SIEMENS

NX Nastran 11

Quick Reference Guide

Contents

Proprietary & Restricted Rights Notice 23

nastran Command and NASTRAN Statement 1-1

The nastran Command 1-1
The NASTRAN Statement 1-13
NASTRAN 1-14

File Management Statements 2-1

Key to Descriptions 2-2
The File Management Section (FMS) 2-3
File Management Statement Summary 2-3
File Management Statement Descriptions 2-5
WHERE and CONVERT Clauses 2-7
$ 2-9
ACQUIRE 2-10
ASSIGN 2-11
CONNECT 2-16
DBCLEAN 2-19
DBDICT 2-20
DBDIR 2-34
DBFIX 2-36
DBLOAD 2-38
DBLOCATE 2-43
DBSETDEL 2-47
DBUNLOAD 2-48
DBUPDATE 2-51
DEFINE 2-53
ENDJOB 2-57
EXPAND 2-58
INCLUDE 2-60
INIT 2-62
PROJ 2-67
RESTART 2-69

Executive Control Statements 3-1

Key to Descriptions 3-2
Executive Control Section 3-2
Executive Control Statement Summary 3-3

Executive Control Statement Descriptions 3-3
$ 3-5
ALTER 3-6
APP 3-10
CEND 3-11
COMPILE 3-12
COMPILER 3-17
DIAG 3-19
DOMAINSOLVER 3-24
ECHO 3-25
ENDALTER 3-26
GEOMCHECK 3-27
ID 3-34
INCLUDE 3-35
LINK 3-37
MALTER 3-40
SOL 3-45
SOL 601,N or SOL 701 3-49
TIME 3-55

Case Control Commands 4-1

Key to Descriptions 4-2
The Case Control Section 4-3
Format of Case Control Command Descriptions 4-3
Case Control Command Summary 4-4
Case Control Command Descriptions 4-17
$ 4-18
A2GG 4-19
ACCELERATION 4-21
ACINTENSITY 4-25
ACPOWER 4-27
ACVELOCITY 4-29
ADAMSMNF 4-31
ADAPTERR 4-44
ADMRECVR 4-47
AECONFIG 4-51
AEROF 4-52
AESYMXY 4-53
AESYMXZ 4-55
ANALYSIS 4-57
APRESSURE 4-62
AUXCASE 4-63
AUXMODEL 4-64
B2GG 4-65
B2PP 4-67
BC 4-69

BCRESULTS 4-70
BCSET 4-74
BEGIN BULK 4-77
BGRESULTS 4-79
BGSET 4-81
BOLTLD (case control) 4-84
BOLTRESULTS 4-87
BOUTPUT 4-89
CKGAP 4-91
CLOAD 4-93
CMETHOD 4-94
CRSTRN 4-95
CSSCHD 4-97
CYCFORCES 4-98
CYCSET 4-100
CZRESULTS 4-101
DEFORM 4-103
DESGLB 4-105
DESOBJ 4-107
DESSUB 4-109
DISPLACEMENT 4-110
DIVERG 4-115
DLOAD 4-116
DRSPAN 4-118
DSAPRT 4-120
DSYM 4-123
DTEMP 4-124
EBDSET 4-126
ECHO 4-127
EDE 4-130
EFLOAD 4-133
EKE 4-136
ELSDCON 4-139
ELSTRN 4-141
ELSUM 4-144
ENTHALPY 4-147
ERP 4-149
ESE 4-153
EXTSEOUT 4-158
FLSFSEL 4-168
FLSPOUT 4-170
FLSTCNT 4-173
FLUX 4-176
FMETHOD 4-178
FORCE 4-179
FREQUENCY 4-183
GCRSTRN 4-184

GELSTRN 4-186
GKRESULTS 4-188
GPFORCE 4-190
GPKE 4-194
GPLSTRN 4-196
GPRSORT 4-198
GPSDCON 4-199
GPSTRAIN 4-201
GPSTRESS 4-203
GROUNDCHECK 4-205
GSTRAIN 4-207
GSTRESS 4-209
GTHSTRN 4-211
GUST 4-213
HARMONICS 4-214
HDOT 4-216
HOUTPUT 4-218
IC 4-220
INCLUDE 4-223
INITS 4-225
JINTEG 4-226
K2GG 4-227
K2PP 4-229
K42GG 4-231
LABEL 4-233
LINE 4-234
LOAD 4-235
LOADSET 4-237
M2GG 4-239
M2PP 4-241
MASTER 4-243
MAXLINES 4-245
MAXMIN 4-246
MBDEXPORT 4-249
MBDRECVR 4-286
MEFFMASS 4-290
METHOD 4-294
MFLUID 4-296
MODALE 4-297
MODCON 4-301
MODES 4-305
MODSEL 4-307
MODTRAK 4-309
MONVAR 4-310
MPC 4-312
MPCFORCES 4-313
MPRES 4-317

NLCNTL 4-319
NLLOAD 4-320
NLPARM 4-322
NLSTRESS 4-323
NONLINEAR 4-325
NOUTPUT 4-326
NSM 4-327
OFREQUENCY 4-328
OLOAD 4-330
OMODES 4-334
OPRESS 4-337
OSTNINI 4-339
OTEMP 4-341
OTIME 4-343
OUTPUT 4-345
P2G 4-347
PANCON 4-349
PARAM 4-353
PARTN 4-355
PFRESULTS 4-356
PLOTID 4-358
PLSTRN 4-359
PRESSURE 4-361
RANDOM 4-362
RCROSS 4-364
REPCASE 4-366
RESVEC 4-368
RIGID 4-372
RMAXMIN 4-375
RMETHOD 4-379
RSMETHOD 4-380
SACCELERATION 4-382
SDAMPING 4-384
SDISPLACEMENT 4-386
SEALL 4-388
SEDR 4-390
SEDV 4-392
SEEXCLUDE 4-394
SEFINAL 4-396
SEKREDUCE 4-398
SELGENERATE 4-400
SELREDUCE 4-402
SEMGENERATE 4-404
SEMREDUCE 4-406
SEQDEP 4-408
SERESP 4-410
SET 4-412

SET 4-415
SETMC 4-418
SETMCNAME 4-420
SETS DEFINITION 4-422
SHELLTHK 4-423
SKIP 4-425
SMETHOD 4-426
SPC 4-427
SPCFORCES 4-428
STATSUB 4-432
STATVAR 4-434
STRAIN 4-436
STRESS 4-442
STRFIELD 4-449
SUBCASE 4-451
SUBCOM 4-453
SUBSEQ 4-455
SUBTITLE 4-457
SUPER 4-458
SUPORT1 4-460
SURFACE 4-461
SVECTOR 4-465
SVELOCITY 4-467
SYM 4-469
SYMCOM 4-470
SYMSEQ 4-472
TEMPERATURE 4-473
TFL 4-479
THERMAL 4-481
THSTRN 4-483
TITLE 4-486
TRIM 4-487
TSTEP 4-488
TSTEPNL 4-490
TSTRU 4-491
VECTOR 4-493
VELOCITY 4-494
VOLUME 4-498
WEIGHTCHECK 4-500
Case Control Applicability Tables 4-501

Case Control X-Y PLOT Commands 5-1

Case Control X-Y PLOT Commands Overview 5-1
X-Y Output Command Summary 5-2
ALLEDGE TICS 5-6
BALL EDGE TICS 5-8

BLEFT TICS 5-9
BRIGHT TICS 5-10
CAMERA 5-11
CLEAR 5-12
CSCALE 5-13
CURVELINESYMBOL 5-14
DENSITY 5-16
LEFT TICS 5-17
LONG 5-18
LOWER TICS 5-19
PENSIZE 5-20
PLOTTER 5-21
RIGHT TICS 5-22
TALL EDGE TICS 5-23
TCURVE 5-24
TLEFT TICS 5-25
TRIGHT TICS 5-26
UPPER TICS 5-27
XAXIS 5-28
XBAXIS 5-29
XBGRID LINES 5-30
XDIVISIONS 5-31
XGRID LINES 5-32
XINTERCEPT 5-33
XLOG 5-34
XMAX 5-35
XMIN 5-36
XPAPER 5-37
XTGRID LINES 5-38
XTITLE 5-39
XYPAPLOT 5-40
XYPEAK 5-41
XYPLOT 5-42
XYPRINT 5-46
XYPUNCH 5-47
XVALUE PRINT SKIP 5-48
YAXIS 5-49
YBDIVISIONS 5-50
YBINTERCEPT 5-51
YBGRID LINES 5-52
YBLOG 5-53
YBMAX 5-54
YBMIN 5-55
YBTITLE 5-56
YBVALUE PRINT SKIP 5-57
YDIVISIONS 5-58
YINTERCEPT 5-59

YGRID LINES 5-60
YLOG 5-61
YMAX 5-62
YMIN 5-63
YPAPER 5-64
YTAXIS 5-65
YTDIVISIONS 5-66
YTGRID LINES 5-67
YTINTERCEPT 5-68
YTITLE 5-69
YTLOG 5-70
YTMAX 5-71
YTMIN 5-72
YTTITLE 5-73
YTVALUE PRINT SKIP 5-74
YVALUE PRINT SKIP 5-75

Parameters and Parameter Applicability Tables 6-1

Parameter Descriptions 6-1
A – Parameters 6-2
B – Parameters 6-9
C – Parameters 6-10
D – Parameters 6-21
E – Parameters 6-26
F – Parameters 6-33
G – Parameters 6-38
H – Parameters 6-40
I – Parameters 6-41
K – Parameters 6-44
L – Parameters 6-45
M – Parameters 6-48
N – Parameters 6-53
O – Parameters 6-58
P – Parameters 6-67
Q & R – Parameters 6-80
S – Parameters 6-88
T – Parameters 6-102
U – Parameters 6-104
V – Parameters 6-108
W & X – Parameters 6-109
Parameter Applicability Tables 6-111

Item Codes 7-1

Item Codes Overview 7-1
Element Stress (or Strain) Item Codes 7-2

Element Force Item Codes 7-84
Fluid Virtual Mass Pressure Item Codes 7-104
Slideline Contact Item Codes 7-105
Element Strain Energy Item Codes 7-106

Degree-of-Freedom Sets 8-1

Degree-of-Freedom Set Definitions 8-1
Degree-of-Freedom Set Bulk Data Entries 8-4
User Defined Degree-of-Freedom Sets 8-5

Bulk Data Entries 9-1

Key to Descriptions 9-2
The Bulk Data Section 9-3
Bulk Data Entry Descriptions 9-3
Bulk Data Syntax Rules 9-4
Continuations 9-8
Bulk Data Entry Summary 9-13
Geometry 9-13
Elements 9-16
Materials 9-27
Constraints and Partitioning 9-29
Loads 9-32
Solution Control 9-37
Miscellaneous 9-42
Element Utilization Summary 9-45
Key to Element Applicability Table 9-46
Element Applicability Table 9-47
Element-Property-Material Cross-Reference Tables 9-49
$ 9-58
/ 9-59

Bulk Data Entries A—B 10-1

ACCEL 10-2
ACCEL1 10-5
ACMODL 10-7
ACSRCE 10-11
ADUMi 10-14
AECOMP 10-15
AECOMPL 10-17
AEDW 10-18
AEFACT 10-20
AEFORCE 10-22
AELINK 10-24
AELIST 10-26
AEPARM 10-27

AEPRESS 10-28
AERO 10-30
AEROS 10-32
AESTAT 10-34
AESURF 10-36
AESURFS 10-39
AMLREG 10-41
ASET 10-43
ASET1 10-45
BAROR 10-47
BCONP 10-49
BCPROP 10-52
BCPROPS 10-54
BCRPARA 10-56
BCTADD 10-59
BCTPARA 10-61
BCTPARM 10-73
BCTSET 10-85
BDYOR 10-89
BEAMOR 10-91
BEDGE 10-93
BFLUID 10-96
BFRIC 10-98
BGADD 10-100
BGPARM 10-102
BGSET 10-108
BLSEG 10-112
BNDFIX 10-115
BNDFIX1 10-117
BNDFREE 10-119
BNDFREE1 10-121
BNDGRID 10-123
BOLT 10-125
BOLTFOR 10-130
BOLTFRC 10-132
BOLTLD (bulk entry) 10-135
BOLTSEQ 10-138
BOUTPUT 10-141
BSET 10-143
BSET1 10-145
BSURF 10-147
BSURFS 10-149
BWIDTH 10-151

Bulk Data Entries CA—CM 11-1

CAABSF 11-2

CAERO1 11-4
CAERO2 11-7
CAERO3 11-10
CAERO4 11-13
CAERO5 11-15
CBAR 11-20
CBARAO 11-26
CBEAM 11-28
CBEAR 11-34
CBEND 11-36
CBUSH 11-40
CBUSH1D 11-45
CCHOCK3 11-47
CCHOCK4 11-51
CCHOCK6 11-55
CCHOCK8 11-59
CDAMP1 11-63
CDAMP2 11-65
CDAMP3 11-67
CDAMP4 11-69
CDAMP5 11-71
CDUMi 11-73
CELAS1 11-75
CELAS2 11-77
CELAS3 11-80
CELAS4 11-82
CFAST 11-84
CGAP 11-87
CHACAB 11-91
CHACBR 11-94
CHBDYE 11-97
CHBDYG 11-102
CHBDYP 11-107
CHEXA 11-112
CHEXCZ 11-117
CLOAD 11-119
CMASS1 11-121
CMASS2 11-123
CMASS3 11-125
CMASS4 11-127

Bulk Data Entries CO—CY 12-1

CONM1 12-2
CONM2 12-4
CONROD 12-7
CONV 12-9

CONVM 12-12
CORD1C 12-14
CORD1R 12-16
CORD1S 12-18
CORD2C 12-20
CORD2R 12-23
CORD2S 12-25
CORD3G 12-28
CPENTA 12-30
CPENTCZ 12-33
CPLSTN3 12-35
CPLSTN4 12-39
CPLSTN6 12-43
CPLSTN8 12-47
CPLSTS3 12-51
CPLSTS4 12-55
CPLSTS6 12-59
CPLSTS8 12-63
CPYRAM 12-67
CQUAD 12-70
CQUAD4 12-73
CQUAD8 12-78
CQUADR 12-84
CQUADX4 12-89
CQUADX8 12-93
CRAC2D 12-97
CRAC3D 12-100
CRAKTP 12-105
CREEP 12-106
CROD 12-113
CSET 12-115
CSET1 12-117
CSHEAR 12-119
CSSCHD 12-121
CSUPER 12-123
CSUPEXT 12-126
CTETRA 12-128
CTRAX3 12-132
CTRAX6 12-136
CTRIA3 12-140
CTRIA6 12-145
CTRIAR 12-150
CTUBE 12-154
CVISC 12-155
CWELD 12-156
CYAX 12-166
CYCADD 12-168

CYCAXIS 12-170
CYCSET 12-171
CYJOIN 12-173
CYSUP 12-175
CYSYM 12-178

Bulk Data Entries D—E 13-1

DAREA 13-2
DCONADD 13-4
DCONSTR 13-5
DDVAL 13-9
DEFORM 13-11
DEFUSET 13-13
DELAY 13-15
DEQATN 13-17
DESVAR 13-23
DIVERG 13-25
DLINK 13-27
DLOAD 13-29
DMI 13-31
DMIG 13-37
DMIG,UACCEL 13-41
DMIJ 13-43
DMIJI 13-47
DMIK 13-51
DOPTPRM 13-55
DPHASE 13-63
DRESP1 13-65
DRESP2 13-77
DRESP3 13-83
DSCREEN 13-88
DTABLE 13-90
DTEMP 13-91
DTEMPEX 13-93
DTI 13-94
DTI,DFRFNC 13-96
DTI,ESTDATA 13-99
DTI,INDTA 13-104
DTI,SELOAD 13-107
DTI,SETREE 13-109
DTI,SPECSEL 13-111
DTI,SPSEL 13-113
DTI,UNITS 13-115
DVBSHAP 13-119
DVCREL1 13-121
DVCREL2 13-123

DVEREL1 13-125
DVGRID 13-128
DVMREL1 13-130
DVMREL2 13-132
DVPREL1 13-134
DVPREL2 13-137
DVSHAP 13-140
EBDADD 13-142
EBDSET 13-143
ECHOOFF 13-145
ECHOON 13-146
EIGB 13-147
EIGC 13-149
EIGP 13-160
EIGR 13-162
EIGRL 13-167
ELIST 13-175
ENDDATA 13-177
EPOINT 13-178
EXTRN 13-180

Bulk Data Entries F—L 14-1

FLFACT 14-2
FLUTTER 14-4
FORCE 14-7
FORCE1 14-9
FORCE2 14-11
FREQ 14-13
FREQ1 14-15
FREQ2 14-17
FREQ3 14-19
FREQ4 14-23
FREQ5 14-26
GENEL 14-28
GRAV 14-32
GRDSET 14-35
GRID 14-37
GROUP 14-39
GUST 14-41
INCLUDE 14-43
INITADD 14-45
INITS 14-47
IPLANE 14-51
ITER 14-53
LOAD 14-59
LOADCYH 14-61

LOADCYN 14-63
LOADCYT 14-65
LSEQ 14-67

Bulk Data Entries M—N 15-1

MAT1 15-2
MAT2 15-8
MAT3 15-11
MAT4 15-14
MAT5 15-16
MAT8 15-18
MAT9 15-22
MAT10 15-25
MAT11 15-28
MATCID 15-31
MATCRP 15-33
MATCZ 15-35
MATDMG 15-38
MATFT 15-41
MATG 15-45
MATHE 15-48
MATHEM 15-54
MATHEV 15-55
MATHP 15-57
MATOVR 15-61
MATPLCY 15-63
MATPOR 15-65
MATS1 15-68
MATSMA 15-76
MATSR 15-79
MATT1 15-81
MATT2 15-85
MATT3 15-88
MATT4 15-91
MATT5 15-93
MATT8 15-95
MATT9 15-98
MATT11 15-101
MATTC 15-104
MATVE 15-106
MFLUID 15-108
MICPNT 15-111
MKAERO1 15-112
MKAERO2 15-114
MODTRAK 15-116
MOMENT 15-118

MOMENT1 15-120
MOMENT2 15-122
MONPNT1 15-124
MPC 15-126
MPCADD 15-128
MUMAT 15-130
NLCNTL 15-132
NLPARM 15-144
NLPCI 15-151
NLRGAP 15-154
NOLIN1 15-157
NOLIN2 15-159
NOLIN3 15-161
NOLIN4 15-164
NSM 15-167
NSM1 15-169
NSMADD 15-171
NSML 15-173
NSML1 15-175
NXSTRAT 15-178

Bulk Data Entries O—P 16-1

OMIT 16-2
OMIT1 16-4
PAABSF 16-6
PACABS 16-8
PACBAR 16-10
PAERO1 16-11
PAERO2 16-12
PAERO3 16-15
PAERO4 16-18
PAERO5 16-22
PANEL 16-26
PARAM 16-28
PBAR 16-30
PBARL 16-33
PBCOMP 16-40
PBEAM 16-46
PBEAML 16-52
PBEAR 16-60
PBEND 16-71
PBUSH 16-79
PBUSH1D 16-82
PBUSHT 16-88
PCHOCK 16-91
PCOMP 16-93

PCOMPG 16-100
PCOMPS 16-106
PCONV 16-112
PCONVM 16-114
PDAMP 16-117
PDAMP5 16-118
PDAMPT 16-119
PDUMi 16-121
PELAS 16-122
PELAST 16-124
PFAST 16-126
PGAP 16-128
PGPLSN 16-133
PHBDY 16-135
PLCYISO 16-137
PLCYKIN 16-139
PLCYRUP 16-140
PLOAD 16-141
PLOAD1 16-145
PLOAD2 16-150
PLOAD4 16-152
PLOADE1 16-156
PLOADX1 16-159
PLOTEL 16-162
PLPLANE 16-164
PLSOLID 16-166
PMASS 16-168
PMIC 16-169
POINT 16-170
PPLANE 16-172
PRAC2D 16-174
PRAC3D 16-176
PROD 16-178
PSHEAR 16-180
PSHELL 16-182
PSHL3D 16-186
PSOLCZ 16-188
PSOLID 16-191
PTUBE 16-206
PVISC 16-208
PWELD 16-209

Bulk Data Entries Q—S 17-1

QBDY1 17-2
QBDY2 17-4
QBDY3 17-6

QHBDY 17-8
QSET 17-10
QSET1 17-12
QVECT 17-14
QVOL 17-17
RADBC 17-19
RADBND 17-21
RADCAV 17-23
RADLST 17-26
RADM 17-28
RADMT 17-30
RADMTX 17-32
RADSET 17-34
RANDPS 17-35
RANDT1 17-37
RBAR 17-39
RBE1 17-42
RBE2 17-45
RBE3 17-48
RCROSS 17-52
RCROSSC 17-55
RELEASE 17-58
RFORCE 17-60
RFORCE1 17-66
RFORCE2 17-73
RLOAD1 17-77
RLOAD2 17-81
ROTORB 17-85
ROTORD 17-87
ROTORG 17-94
ROTPARM 17-96
ROTSE 17-99
RROD 17-101
RSPLINE 17-103
RSSCON 17-105
RTRPLT 17-108
RVDOF 17-110
RVDOF1 17-112
RVEL 17-114
SEBNDRY 17-116
SEBSET 17-118
SEBSET1 17-120
SEBULK 17-122
SECONCT 17-125
SECSET 17-127
SECSET1 17-129
SEELT 17-131

SEEXCLD 17-133
SELABEL 17-135
SELOAD 17-136
SELOC 17-138
SEMPLN 17-140
SENQSET 17-141
SEQEP 17-143
SEQGP 17-144
SEQSEP 17-146
SEQSET 17-148
SEQSET1 17-150
SESET 17-152
SESUP 17-154
SET1 17-156
SET2 17-158
SET3 17-160
SETREE 17-162
SEUSET 17-165
SEUSET1 17-167
SLOAD 17-169
SNORM 17-170
SPC 17-172
SPC1 17-174
SPCADD 17-176
SPCD 17-178
SPCOFF 17-180
SPCOFF1 17-181
SPLINE1 17-183
SPLINE2 17-186
SPLINE3 17-189
SPLINE4 17-192
SPLINE5 17-195
SPOINT 17-198
SUPORT 17-200
SUPORT1 17-202
SWLDPRM 17-204

Bulk Data Entries T—V 18-1

TABDMP1 18-2
TABLED1 18-6
TABLED2 18-11
TABLED3 18-14
TABLED4 18-17
TABLEM1 18-19
TABLEM2 18-23
TABLEM3 18-26

TABLEM4 18-29
TABLES1 18-31
TABLEST 18-34
TABRND1 18-37
TABRNDG 18-41
TABVE 18-43
TEMP 18-45
TEMPBC 18-47
TEMPD 18-50
TEMPEX 18-52
TEMPP1 18-53
TEMPP3 18-56
TEMPRB 18-57
TF 18-61
TIC 18-63
TLOAD1 18-65
TLOAD2 18-69
TMCPARA 18-73
TRIM 18-77
TSTEP 18-79
TSTEP1 18-82
TSTEPNL 18-85
USET 18-91
USET1 18-93
UXVEC 18-95
VCEV 18-97
VIEW 18-99
VIEW3D 18-102

Proprietary & Restricted Rights Notice

© 2016 Siemens Product Lifecycle Management Software Inc. All Rights Reserved.

This software and related documentation are proprietary to Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. NX is a trademark or registered trademark of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries.

NASTRAN is a registered trademark of the National Aeronautics and Space Administration. NX Nastran is an enhanced proprietary version developed and maintained by Siemens Product Lifecycle Management Software Inc.

MSC is a registered trademark of MSC.Software Corporation. MSC.Nastran and MSC.Patran are trademarks of MSC.Software Corporation.

All other trademarks are the property of their respective owners.

TAUCS Copyright and License

TAUCS Version 2.0, November 29, 2001. Copyright (c) 2001, 2002, 2003 by Sivan Toledo, Tel-Aviv University, stoledo@tau.ac.il. All Rights Reserved.

TAUCS License:

Your use or distribution of TAUCS or any derivative code implies that you agree to this License.

THIS MATERIAL IS PROVIDED AS IS, WITH ABSOLUTELY NO WARRANTY EXPRESSED OR IMPLIED. ANY USE IS AT YOUR OWN RISK.

Permission is hereby granted to use or copy this program, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any derivative code must cite the Copyright, this License, the Availability note, and "Used by permission." If this code or any derivative code is accessible from within MATLAB, then typing "help taucs" must cite the Copyright, and "type taucs" must also cite this License and the Availability note. Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. This software is provided to you free of charge.

Availability (TAUCS)

As of version 2.1, we distribute the code in 4 formats: zip and tarred-gzipped (tgz), with or without binaries for external libraries. The bundled external libraries should allow you to build the test programs on Linux, Windows, and MacOS X without installing additional software. We recommend that you download the full distributions, and then perhaps replace the bundled libraries by higher performance ones (e.g., with a BLAS library that

is specifically optimized for your machine). If you want to conserve bandwidth and you want to install the required libraries yourself, download the lean distributions. The zip and tgz files are identical, except that on Linux, Unix, and MacOS, unpacking the tgz file ensures that the configure script is marked as executable (unpack with tar zxvpf), otherwise you will have to change its permissions manually.

Chapter 1: nastran Command and NASTRAN Statement

1.1 The nastran Command

NX Nastran is executed with a command called nastran. (Your system administrator may assign a different name to the command.) The nastran command permits the specification of keywords used to request options affecting NX Nastran job execution. The format of the nastran command is:

```
nastran input_data_file [keyword1 = value1  keyword2 = value2 ...]
```

where input_data_file is the name of the file containing the input data and $keyword_i$=$value_i$ is one or more optional keyword assignment arguments. For example, to run a job using the data file example1.dat, enter the following command:

```
nastran example1
```

Keyword assignments can be specified as command line arguments and/or included in RC files.

In addition to the RC file located in *install_dir*\conf\nastr.rcf, there are two local RC file options.

- You can create an RC file in your home directory.
- You can create an RC file in the same directory as the input file. This RC file is ignored when the “rcf” keyword is used.

The following syntax rules exist for RC files.

- The tilde (~) character is not recognized within RC files.
- Environment variables are only recognized when used in the context of a logical symbol (see “Using Filenames and Logical Symbols” in the *NX Nastran Installation and Operations Guide*).
- When a keyword is specified on the command line, embedded spaces or special characters that are significant to the shell must be properly quoted; quotes should not be used within RC files.

The keywords listed below are the most common ones for various computers, but are not available on all computers. Also, the defaults may be site dependent. Please consult the **Keywords and Environment Variables** in the *NX Nastran Installation and Operations Guide* for keyword applicability, specialized keywords, and further discussion of keywords and their defaults. Also see the *NX Nastran Parallel Processing User’s Guide* for details on all parallel processing keywords. Keywords that use yes/no values accept partial

specification and case-independent values. For example, “yes” may be specified as “y”, “ye”, or “yes” using uppercase or lowercase letters.

after — after=time — Default: None

Holds the job until the time specified by time. See the description of the “at” command in your system documentation for the format of time.

Example: `nastran example after=10:00`

The job is held until 10:00 A.M.

append — append={yes|no} — Default: no

Controls the combining of the F04, F06, and LOG files into a single file after the run completes. If “no” is specified, the files are not combined. If “yes” is specified, the files are combined into one file with the suffix “.out”.

Example: `nastran example append=yes`

The F04, F06, and LOG files are combined into a file named example.out.

authqueue — authqueue=*number of minutes* — Default: 0

When NX Nastran attempts to run, but cannot because a license is unavailable, the job will be retried every minute up to the value of the AUTHQUEUE keyword.

batch — batch={yes|no} — Default: yes

Indicates how the job is to be run. If “yes” is specified, the job is run as a background process. If “no” is specified, the job is run in the foreground. If the “aft” or “queue” keywords are specified, the batch keyword is ignored.

Note

If the job is already running in an NQS or NQE batch job, the default is “no”.

Example: `nastran example batch=no`

The job is run in the foreground.

buffpool — buffpool=*value* — The default is 51 buffers. The value defined in the installation RC file is buffpool=20X.

Specifies the memory size of the buffer pool. The buffer pool is a portion of the total memory used to cache database memory. *value* can be defined as a percentage of the total memory requested with the keyword **memory**, or as a specific value.

- To define as a percentage of the total memory requested with the keyword **memory**, enter

  ```
  buffpool=nX
  ```

  where *n* is a percentage 0<*n*<100, and X represents the total memory requested with the keyword **memory**. For example, if **memory**=5Gb is defined, and **buffpool**=20X is defined, the memory requested for the buffer pool is 20% of 5Gb, or 1Gb.

- When *value* is defined as a specific value, you either enter the number of buffers (default), or a number followed by one of the modifiers below. In the conversion column, the *bytes_per_word* is 4 using the LP-64 executables, and 8 using ILP-64 executables. The modifier may be specified using any case combination.

Modifier	**Conversion to number of words**
G or Gw	Multiply *entered_value* by 1024**3.
Gb	Multiply *entered_value* by (1024**3)/*bytes_per_word*.
M or Mw	Multiply *entered_value* by 1024**2.
Mb	Multiply *entered_value* by (1024**2)/*bytes_per_word*.
K or Kw	Multiply *entered_value* by 1024.
Kb	Multiply *entered_value* by 1024/*bytes_per_word*.
w	Use *entered_value* as is.
b	Divide *entered_value* by *bytes_per_word*.

where *bytes_per_word* is 4 using the LP-64 executables, and 8 using ILP-64 executables. The modifier may be specified using any case combination.

Example: `nastran example buffpool=200`

This example requests 200 buffers for the buffer pool since no modifier exists.

buffsize =*value* The default is 8193 words. The value defined in the installation RC file is buffsize=32769.

Specifies the memory size of a buffer in words.

cl_dcmp =*value*

Enables GPU for FRRD1 module computations.

Example: `nastran.exe cl_dcmp=1 input_file.dat`

=1 Enables GPU for DCMP module computations with the first available AMD or NVIDIA GPU.

=2 Enables GPU for DCMP module computations with the first available AMD GPU.

=1 and sys531=1 Enables GPU for DCMP module computations with the first available NVIDIA GPU (both cl_dcmp=1 and sys531=1 are required for this option).

cl_frrd =*value*

Enables GPU for FRRD1 module computations.

Example: `nastran.exe cl_frrd=1 input_file.dat`

=1 Enables GPU for FRRD1 module computations with the first available AMD or NVIDIA GPU.

=2 Enables GPU for FRRD1 module computations with the first available AMD GPU.

=3 Enables GPU for FRRD1 module computations with the first available NVIDIA GPU.

dbs dbs=*pathname* Default=input_data_file directory

Creates database files using an alternate file prefix. (See “Default DBsets and Their Default Attribute” in the *NX Nastran Installation and Operations Guide*.) If “dbs” is not specified, database files are created in the current directory using the base name of the input data file as the prefix. If the “dbs” value is a directory, database files are created in the specified directory using the base name of the input data file as the filename.

Note

If “dbs” is specified and “scratch=yes” is specified, a warning will be issued and “scratch=no” assumed. This is a change from the behavior of prior releases, which would ignore the “dbs” keyword if “scratch=yes” was set.

In the following examples, assume the following directory contents:

```
example.dat  mydir/  other/
other/example.dat
```

where “mydir” and “other” are directories.

Example: `nastran example`

or: `nastran other/example`

Database files are created in the current directory with name example, e.g., ./example.DBALL.

Example: `nastran example dbs=myfile`

Database files are created in the current directory with name myfile, e.g., ./myfile.DBALL.

Example: `nastran example dbs=mydir`

Database files are created in the mydir directory with name example, e.g., mydir/example.DBALL.

Example: `nastran example dbs=mydir/myfile`

Database files are created in the mydir directory with name myfile, e.g., mydir/myfile.DBALL.

gpgpu =*value*

Enables GPU for both FRRD1 and DCMP module computations.

Example: `nastran.exe gpgpu=any input_file.dat`

=none Disables GPU for both FRRD1 and DCMP module computations.

=any Enables GPU for both FRRD1 and DCMP module computations with the first available AMD or NVIDIA GPU.

=amd Enables GPU for both FRRD1 and DCMP module computations with the first available AMD GPU.

intel_mic =*value*

Enables MKL computations with Intel's MIC architecture.

Example: `nastran.exe intel_mic=1 input_file.dat`

=1 Enables MKL computations with Intel's MIC architecture.

memory

memory=*memory_size* Default=estimate
The value defined in the installation RC file is 0.45*physical.

Specifies the amount of open core memory to allocate, where *memory_size* can be defined as a fraction of the machine physical memory, as "estimate", or as a specific value.

- To define *memory_size* as a fraction of the machine physical memory, enter:

  `memory=n*physical`

  where *n* is a fraction 0<*n*<1.0.

- If memory=estimate is defined, ESTIMATE will be used to determine the memory size.

- When *memory_size* is defined as a specific value, you either enter the number of words, or a number followed by one of the modifiers below. In the conversion column, the *bytes_per_word* is 4 using the LP-64 executables, and 8 using ILP-64 executables. The modifier may be specified using any case combination.

Modifier	**Conversion to number of words**
G or Gw	Multiply *memory_size* by 1024**3.
Gb	Multiply *memory_size* by (1024**3)/*bytes_per_word*.
M or Mw	Multiply *memory_size* by 1024**2.

Mb	Multiply *memory_size* by (1024**2)/*bytes_per_word*.
K or Kw	Multiply *memory_size* by 1024.
Kb	Multiply *memory_size* by 1024/*bytes_per_word*.
w	*memory_size* is the number of words.
b	Divide *memory_size* by *bytes_per_word*.
Example:	`nastran example memory=25mw`
	The job is run using an open core memory size of 25 megawords, or 25 600 kilowords, or 26 214 400 words.

The maximum *memory_size* limits are included in the *NX Nastran Installaton and Operations Guide* in the section titled “Maximum Memory Size”.

news

news={yes|no} Default=yes

Displays the news file in the F06 file. If “yes” is specified, the news file is displayed in the F06 file regardless of when it was last changed. If “no” is specified, the news file is not displayed in the F06 file.

Example: `nastran example news=yes`

The news file is displayed in the F06 file after the title page block.

notify

notify={yes|no} Default=yes

Sends notification when the job is completed. See also the “ncmd” keyword in the *NX Nastran Installation and Operations Guide*.

Note

If the job is queued using the queue keyword, or the job is already running in an NQS batch job, the default is “no”.

Example: `nastran example notify=yes`

A message is sent when the job is completed.

old

old={yes|no} Default=yes

Saves previous copies of the F04, F06, LOG, OP2, OUT, PCH, and PLT output files using sequence numbers. Sequence numbers are appended to the keyword filename and are separated by a period.

If “yes” is specified, the highest sequence number of each of the output files is determined. The highest sequence number found is incremented by one to become the new sequence number. Then, all current output files that do not include sequence numbers are renamed using the new sequence number as a suffix.

Example: `nastran v2401 old=yes`

For example, the user’s directory contains the following files:

v2401.dat	v2401.log
v2401.f04	v2401.log.1
v2401.f04.1	v2401.log.2
v2401.f04.2	v2401.log.3
v2401.f06	v2401.f06.1

Apparently, the user ran the job four times, but deleted some of the files. When the next job is run, the following files are renamed: v2401.f04 is renamed to v2401.f04.4, v2401.f06 is renamed to v2401.f06.4, and v2401.log is renamed to v2401.log.4. The sequence number 4 is used because it is one greater than the highest sequence number of all of the selected files (the highest being v2401.log.3). Using this method, all files related to a single run will have the same sequence number.

out

out=pathname Default=input_data_file directory

Saves the output files using a different file prefix or in a different directory. If “out” is not specified, the output files are saved in the current directory using the base name of the input data file as a prefix.

If the “out” value is a directory, output files are created in the specified directory using the base name of the input data file as the filename. In the following examples, assume the following directory contents:

```
example.data mydir/ other/

other/example.dat
```

where “mydir” and “other” are directories.

Example: `nastran example`

or: `nastran other/example`

Output files are created in the current directory with name example, e.g., ./example.f06.

Example: `nastran example out=myfile`

Output files are created in the current directory with name myfile, e.g., ./myfile.f06.

Example: `nastran example out=mydir`

Output files are created in the mydir directory with name example, e.g., mydir/example.f06.

Example: `nastran example out=mydir/myfile`

Output files are created in the mydir directory with name myfile, e.g., mydir/myfile.f06.

parallel — Same as smp keyword.

rcf — rcf=pathname Default=no

Specifies the name of the local RC file. If this keyword is not specified, then the .nast#rc (# designates release number) file from the data file directory is used.

Example: `$ nastran example rcf=nast.rc`

scratch — scratch={yes|no|mini} Default=no

Deletes the database files at the end of the run. If the database files are not required, “scratch=yes” can be used to remove them. If “mini” is specified, a reduced size database (that can only be used for data recovery restarts) will be created. See “Mini-Database Restarts” in the *NX Nastran User’s Guide* for further details on the “mini” database.

Example: `nastran example scratch=yes`

All database files created by the run are deleted at the end of the job in the same way as the FMS statement INIT MASTER(S).

sdirectory — sdirectory=*directory* Default: See the description below.

Specifies the directory to use for temporary scratch files created during the run. As NX Nastran can create very large scratch files, the scratch directory should contain sufficient space to store any scratch files created during a run. You must have read, write, and execute privileges to the directory.

See “Determining Resource Requirements” in the *NX Nastran Installation and Operations Guide* for information on estimating a job’s total disk space requirements.

The default value is taken from the TMPDIR environment variable if it is set to a non-null value. Otherwise, the computer’s default temporary file directory is chosen; this is usually /tmp, but on SGI systems, it is /var/tmp.

Example: `nastran example sdir=/scratch`

Scratch files are created in the directory /scratch.

smemory

smemory=*value* — The default is 100 buffers. The value defined in the installation RC file is smemory=20X.

Specifies the memory to reserve for scratch memory, where *value* can be defined as a percentage of the total memory requested with the keyword **memory**, or as a specific value. The keyword **smemory** can be overridden by the FMS statement INIT SCRATCH(MEM=*value*).

- To define as a percentage of the total memory requested with the keyword **memory**, enter

  `smemory=nX`

  where *n* is a percentage 0<*n*<100, and X represents the total memory requested with the keyword **memory**. For example, if **memory**=5Gb is defined, and **smemory**=20X is defined, the memory requested for scratch memory is 20% of 5Gb, or 1Gb.

- When *value* is defined as a specific value, you either enter the number of buffers (default), or a number followed by one of the modifiers below. In the conversion column, the *bytes_per_word* is 4 using the LP-64 executables, and 8 using ILP-64 executables. The modifier may be specified using any case combination.

Modifier	Conversion to number of words
G or Gw	Multiply *entered_value* by 1024**3.
Gb	Multiply *entered_value* by (1024**3)/*bytes_per_word*.
M or Mw	Multiply *entered_value* by 1024**2.
Mb	Multiply *entered_value* by (1024**2)/*bytes_per_word*.
K or Kw	Multiply *entered_value* by 1024.
Kb	Multiply *entered_value* by 1024/*bytes_per_word*.
w	Use *entered_value* as is.

b Divide *entered_value* by *bytes_per_word*.

where *bytes_per_word* is 4 using the LP-64 executables, and 8 using ILP-64 executables. The modifier may be specified using any case combination.

Example: `nastran example smem=200`

This example reserves 200 buffers for scratch memory since no modifier exists.

Example: `nastran example smemory=25mw`

This example will use a scratch memory size of 25 megawords, or 25 600 kilowords, or 26 214 400 words.

Note: When you define **smemory** as a percentage of the total memory or in units of words or bytes, the software will convert the value to *number of buffers*. As a result, the value you enter will be adjusted such that **smemory** / **buffsize** is an integer. The value used by the software is reported in the .f04 file. See “Managing Memory” in the *Installations and Operations Guide*.

smp smp=value Default=0

Same as parallel keyword.

Example: `nastran example smp=2`

Specifies the maximum number of CPUs selected for shared-memory parallel (SMP) processing in several numeric modules. SMP processing reduces elapsed time at the expense of increased CPU time. The default is 0, which specifies no SMP processing. If “parallel=1", the parallel algorithms are used on one processor.

Note: If you need to vary the number of SMP CPUs during a job, you must set either the “smp” keyword or SYSTEM(107) on a NASTRAN statement to the maximum number of SMP CPUs that will be requested. Some systems cannot process a DMAP request for CPUs in excess of this initial value.

See the *NX Nastran Parallel Processing User’s Guide* for details on all parallel processing options.

symbol symbol=*symbolic_name*=*string* Default: *None*

Defines a symbolic (or logical) name used on ASSIGN and INCLUDE statements and in command line arguments. This statement can only be specified in initialization or RC files. It *cannot* be specified on the command line (although logical symbols defined using this keyword may be used on the command line). Symbolic names must be 16 characters or less, the value assigned to the symbolic name must be 256 characters or less. If the symbolic name used in ASSIGN or INCLUDE statements or in command line arguments is not defined, it is left in the filename specification as is.

For example, many of the TPL and DEMO input data files have ASSIGN statements, such as the following:

```
ASSIGN  MASTER=DBSDIR:abc.master
```

The string “DBSDIR:” specifies a symbolic name that is to be replaced by another string. The replaced string is defined by the “symbol” keyword in the initialization or RC file or as an environment variable. For example,

```
SYMBOL=DBSDIR=/dbs
```

When the previous ASSIGN statement is processed, the filename assigned to the logical name MASTER is “/dbs/abc.master”. An alternate way of defining symbolic names is through the use of environment variables. For example, issuing the following command at a Bourne or Korn shell prompt:

```
DBSDIR=/dbs; export DBSDIR
```

or the following at a C shell prompt:

```
setenv DBSDIR /dbs
```

is equivalent to the above “symbol” keyword.

Note

If a symbolic name is defined by both an RC file and an environment variable, the symbol statement value will be used.

Keywords and Environment Variables in the *NX Nastran Installation and Operations Guide* contains a list of environment variables that are automatically created by the driver program. Of particular importance to the logical symbol feature are the OUTDIR and DBSDIR variables. These variables refer to the directory that will contain the output files (set using the “out” keyword) and the directory that will contain the permanent database files (set using the “dbs” keyword), respectively.

1.2 The NASTRAN Statement

The NASTRAN statement is used to specify values for certain Executive System operational parameters. These parameters are also called system cells. The NASTRAN statement is used for exceptional circumstances and is therefore not needed in most runs. The NASTRAN statement may also be specified in the Runtime Configuration (RC) files at the system, user, and job level as described in the *NX Nastran Installation and Operations Guide*.

NASTRAN

Executive System Parameter Modification

Specifies values for certain Executive System operational parameters called system cells.

FORMAT:

NASTRAN cellname1=expression1, ..., cellnamen=expression*n*

or

NASTRAN SYSTEM(i)=expression1, ..., SYSTEM(n)=expression*n*

DESCRIBERS

Describer	Meaning
cellnamei	System cell names from Table 1-1.
SYSTEM	Specifies the system cell number.
expression	See DEFINE statement for description.
i	System cell number from Table 1-1.

REMARKS:

1. The NASTRAN statements may appear anywhere in the File Management Section. The NASTRAN statement may also be specified in Runtime Configuration (RC) files. See “Customizing the Runtime Configuration Files” in the *NX Nastran Installation and Operations Guide*.

2. System cell values and their associated cell names may also be set with the DEFINE statement. They may also be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See “PUTSYS, GETSYS” of the *NX Nastran DMAP Programmer’s Guide*.

3. More than one NASTRAN statement and/or DEFINE statement may be present and, if a system cell is specified more than once among these statements, then the last specification takes precedence.

4. The expression will use type conversion rules based on the type (i.e., integer, real, or logical) of the cellname, as defined on a previous DEFINE statement (see the DEFINE statement for conversion rules).

5. If the expression is omitted, the system cell associated with the cellname will be assigned the value as set on a previous DEFINE statement.

6. Only integers are allowed when using the system cell numbers. Thus, when a real number is requested, use the system cell name.

EXAMPLES:

1. Either of the following statements could be used to change the default value for block size.

```
NASTRAN SYSTEM (1)=4097
or
NASTRAN BUFFSIZE=4097
```

Or, if a prior DEFINE statement had defined a keyword MY_SYSBUF to the value 4097 then the following code could be used:

```
NASTRAN SYSTEM(1)=MY_SYSBUF
```

or

```
NASTRAN BUFFSIZE=MY_SYSBUF
```

The following statement is used to request execution of MSGMESH:

```
NASTRAN MESH
```

2. Table 1-1 gives a summary of the recommended system cells.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
1	BUFFSIZE	Specifies the number of words in a physical record. Also called block length.
2	F06	Specifies FORTRAN unit number for standard output file (.f06). (Integer > 0; Default = 6)
9	NLINES	Specifies the number of lines printed per page of output. See LINE.
14	MAXLINES	See MAXLINES.
19	ECHO	See ECHO.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
20	METIME	Minimum time for execution summary table message.
21	APP	Approach Flag. See APP of the Executive Control section. If APP HEAT is specified, then this system cell is set to 1.
22	MACHTYPE	Machine type.
25	DIAGA	Alternate method to set DIAGs 1 through 32. See DIAG.
28	CONFIG	Machine subtype.
31	MESH	Requests execution of MSGMESH.
46 - 54	ADUMi	Dummy element flag., i=1 through 9.
56	HEAT	See APP. 0: Structural analysis. (Default) 1: Heat transfer.
57	HICORE	Working Memory. " See Managing Memory in the *NX Nastran Installation and Operations Guide*.
61	DIAGB	Alternate method to set diagnostics 33 through 64. See DIAG .
64	PUNCH	Specifies FORTRAN unit number for PUNCH file (.f07). (Default = 7)
66	MPYAD	Selects/deselects multiplication methods. See MPYAD in the *NX Nastran DMAP Programmer's Guide*.
69	SOLVE	Controls matrix decomposition. Same as DECOMP in the *NX Nastran DMAP Programmer's Guide* and Option Selection in the *NX Nastran Numerical Methods User's Guide*. 0 or -1: Prints up to 50 messages for null columns and zero diagonals. (Default = -1) 1: Terminates execution when first null column is encountered. 2: Suppresses printing of message when a null column is encountered. 4: Terminates execution when first zero diagonal term is encountered.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
		16: Places 1.0 in diagonal position for all null columns and proceeds with the decomposition. 32: Terminates execution on zero diagonal term. 64: Exits after execution of preface for symmetric decomposition.
70	FBSOPT	Selects forward-backward substitution methods. FBS in the *NX Nastran DMAP Programmer's Guide* and the FBS Method Selection in the *NX Nastran Numerical Methods User's Guide*.
77	DELFF	Deletes form feeds.
81	REAL	Specifies the amount of open core memory that certain numerical modules will be restricted to.
- - -	DBSET	Database neutral file set. SubDMAP DBFETCH in the *NX Nastran DMAP Programmer's Guide* .
82	DMAP	Allows NOGO to operate. See Processing of User Errors in the *NX Nastran DMAP Programmer's Guide*.
86	F04	Specifies FORTRAN unit number for Execution Summary Table (.f04). See "Understanding the .f04 File" in the *NX Nastran User's Guide*. (Integer > 0; Default = 4)
87	RADMTX	Type of radiation exchange coefficient, See RADMTX.
88	RADLST	Print radiation area summary. See RADLST.
107	PARALLEL	Parallel processing deselection (deactivation) for matrix operations. See Keywords in the *NX Nastran Installation and Operations Guide*. 0: Deactivate parallel processing. 1 through 1023: Number of processors. The desired number of processors is summed with the values below in order to deactivate parallel processing methods in the following matrix operations: 1024: Forward-backward substitution. 2048: Decomposition.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
		4096: Multiplication. 8192: Householder in eigenvalue extraction. 65536: Element matrix assembly. 262144: Sparse decomposition. 524288: Sparse forward-backward substitution. If PARALLEL is set simply to the number of processors, then parallel processing is selected in all operations above.
108	NEWHESS	Request complex eigenvalue method. See EIGC.
109	- - -	Controls DMAP execution: 0: Do not execute DMAP instruction if all outputs are previously computed. 1: Always execute DMAP instruction. (Default)
114	BUFFPOOL	Sets bufferpool size. See “Keywords and Environment Variables” in the *NX Nastran Installation and Operations Guide*.
124	ATTDEL	Controls the automatic assignment of the delivery database. See “Delivery Database” in the *NX Nastran User’s Guide* and “Creating and Attaching Alternate Delivery Databases” in the *NX Nastran Installation and Operations Guide*. 0: Enables automatic assigning. (Default) -1: Disables automatic assigning.
125	NOKEEP	Controls NOKEEP option of the RESTART File Management statement. 0: Disable NOKEEP 1: Enable NOKEEP

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
126	SPARSE	Selects the sparse matrix method. For unsymmetrical sparse matrix decomposition method selection, see cell number 209. The following values may be summed in order to select sparse matrix methods in the operations listed below: 0: Deactivate sparse methods. 1: Multiplication. 8: Symmetric decomposition. 16: Forward-backward substitution.
		The default is 25, which is the sum of all values.
128	UPDTTIM	Specifies database directory update time. See DBUPDATE . 0: Do not update. >0: Time, in minutes, between database directory updates.
129	SMPYAD67	Triple Multiply Method Selection in the SMPYAD module. See SMPYAD in the *NX Nastran DMAP Programmer's Guide*. 0: Use current method (Default). 5: Use pre-MSC Nastran Version 67 method.
- - -	MAXDBSET	The maximum number of online DBsets attached to the run.
133	AUTOASGN	Controls auto-assigning of dbsets. Sum the desired values. (Default = 7). See "Database Autoassignment" in the *NX Nastran User's Guide*. 0: No databases are automatically assigned. 1: Only the primary database is automatically assigned. 2: Only the delivery database is automatically assigned. 4: Only located databases are automatically assigned.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
135	TSTAMP	Controls timestamp checking of DBsets.. 0: Do not check. 1: Check. (Default) 2: Same as 1 and print diagnostics.
141	QUADINT	Specifies quadratic or linear interpolation for the line search method in nonlinear analysis. 0: Quadratic interpolation. (Default) 1: Linear interpolation.
142	SCR300	Requests creation of SCR300 partition on SCRATCH DBset. See “INIT” . 1: Do not create SCR300 partition. 2: Create SCR300 partition. (Default)
143	LOCBULK	LOCBULK=1 or 2 specifies that Bulk Data is being obtained via the DBLOCATE FMS statement. NASTRAN LOCBULK=2 is specified when no Bulk Data entries, except for PARAM entries, are to be deleted or added. All PARAM entries must be respecified. All other entries will be ignored and, if present, may increase CPU times in XSORT and IFP. With LOCBULK=2, the XSORT and IFP modules will not reprocess the Bulk Data Section stored in the SEMAP run. Also, GP1, TASNP2, SEP1 and SEP1X modules will be skipped. 0 is the default, which assumes the RESTART FMS statement. This system cell is recognized only in SOLs 101 through 200.
144	- - -	RESTART FMS statement existence flag. Set to 1 if RESTART statement is present.
145	BFGS	Selects strategies of BFGS updates for the arc-length methods in nonlinear analysis. Please see the *NX Nastran Nonlinear Handbook*. 0: Update Δu_R and Δu_P at every iteration with γ^*. (Default) 1: Update Δu_R only with γ. 2: Update Δu_R only with γ^*.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
146	- - -	Reserves (n x BUFFSIZE x 3) memory for faster solution in the Lanczos method of eigenvalue extraction. Default=1. A value of 2 increases the memory reserved by 200%, a value of 3 increases the memory reserved by 300%, etc. See EIGRL . See system cell 229 for a disk space saving feature.
147	UWM	SYSTEM(147)=1 issues User Warning Message for a DMAP parameter appearing on a CALL statement that has an inconsistent authorization in the called subDMAP. 0 is the default, which means no message is issued.
148	DBVERCHK	In general, databases are not compatible between major releases, thus a check is performed in NX Nastran to ensure that the major version which created the database is the same as that being executed. Since specific data on the database may be compatible, SYSTEM(148) allows this check to be circumvented. However, circumventing the check may lead to problems later in the run. 0 (Default): Check is performed. 1: Check is not performed.
150	SCR300DEL	Sets minimum number of blocks of SCR300 partition of SCRATCH DBset at which it is deleted. See "INIT". (Default = 100)
151	- - -	Requests spill or no spill of the SCR300 partition of SCRATCH DBset. See"INIT" . FMS statement. (Default = 1)
155	CBLAMKD	Sets differential stiffness formulation for CBEAM element. 0: Current formulation. (Default) 1: Pre-MSC.Nastran 67 formulation.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
166	- - -	Controls sparse symmetric decomposition. Sum the desired values. (See the section “Matrix Decomposition” in the *Numerical Methods User’s Guide* for more options) 0: No action (Default) 1: If insufficient core is encountered, then switch to conventional decomposition and continue. 2: Print diagnostics 4: Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules. 128: Diagonal scaling.
- - -	OLDQ4K	Requests the pre-MSC.Nastran 68 CQUAD4 element stiffness formulation. No value is required after the keyword. Equivalent to SYSTEM(173)=1.
170	LDQRKD	Selects the differential stiffness method for CQUAD4 and CTRIA3 elements. 0: MSC.Nastran Version 68, improved method. (Default) 1: Pre-MSC Nastran Version 68 method.
173	- - -	0: Default. 1: Requests pre-MSC.Nastran V68 QUAD4 Formulation. 2: Requests MSC.Nastran V68 - V70.5 QUAD4 Formulation.
178		Defines a grid identification number offset for CWELD elements. See the parameter OSWPPT. (Integer > 0; Default = 101,000,000)
182		Defines a constraint element identification number offset for CWELD elements. See the parameter OSWELM. (Integer > 0; Default = 100,001,001)

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
189	Q4TAPER	Specifies the maximum allowable value of taper for the CQUAD4 element. Taper is computed by connecting opposite grid points and computing the area of the enclosed triangles. Another way to think of taper is the ratio of the areas on the two sides of a diagonal. (Real ≥ 0.0; Default = 0.5)
190	Q4SKEW	Specifies the minimum allowable value of skew for the CQUAD4 element. Skew is the angle measured in degrees between the lines that join opposite midsides. (Real ≥ 0.0; Default = 30.0)
191	TETRAAR	Specifies the maximum allowable aspect ratio of the longest edge to the shortest altitude for the CTETRA element. (Real ≥ 0.0; Default = 100.0)
193	DISKSAVE	Lanczos High Performance Option: Controls whether the matrix/vector multiplication is saved in a scratch file or recomputed at every iteration. 0: Save (Default for all machines) 1: No save 2: Save 3: No Save
194	FASTIO	Lanczos High Performance Option: Controls input/output in orthogonalization/renormalization routines. 0: UNPACK/PACK 1: GINO READ/WRITE (Default)
195	FRQSEQ	Lanczos High Performance Option: 100 < Exponent for the rational function used to determine segment boundaries. See also the ALPH field on the See "EIGRL". (Integer; Default = 0, which means equal segments.)
196	SCRSAVE	Lanczos High Performance Option: Controls reuse of scratch files in segment logic. 0: Do not reuse. (Default) 1: Reuse.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
197	NUMSEG	Lanczos High Performance Option: Selects number of segments. See also the NUMS field on the "EIGRL". (Default = 1)
198	MINFRONT	Lanczos High Performance Option: Sets minimum front size. (The default value is machine dependent.) This can also be set with the "rank" keyword. See "Keywords and Environment Variables" for more information on the "rank" keyword, and Computer-Dependent Defaults in the *NX Nastran Installation and Operations Guide* for the machine dependent default values.
199	MASSBUF	Lanczos High Performance Option: Sets half the number of buffers to set aside for storing the mass matrix in core. (Default = 1, which means 2 buffers will be used.)
204	CORDM	Specifies the default value for CORDM field on the PSOLID entry. (Integer ≥ -1; Default = 0)
205	- - -	Defines the number of rows that will be simultaneously updated during a sparse symmetric decomposition. This can also be set with the "rank" keyword. See "Keywords and Environment Variables" and for more information on the "rank" keyword, and Computer-Dependent Defaultsin the *NX Nastran Installation and Operations Guide* for the machine dependent default values.
206	DCMPSEQ	Selects ordering method for sparse matrix decomposition. Note that the EXTREME method has two submethods; BEND and AMF. If EXTREME is used when SYSTEM(206) = 0 (default), then either BEND or AMF is automatically selected by the software depending on the size of the model. If SYSTEM(206) = 4, then BEND is used. 0: EXTREME for 3D, Metis or MMD for 2D (Default) 1: MMD – definite matrices 2: MMD – indefinite matrices 3: No sequencing 4: EXTREME

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
		8: Metis 9: Better of Metis and MMD 32: MLV + 64 (adding 64 to any of the above values): Turns on Supernodal Compression Scheme. For example, sys(206)=72 would indicate Metis (8) + Supernodal Compression Scheme (64).
209	USPARSE	Unsymmetric sparse matrix method selection for the decomposition and forward-backward substitution operations. 0: Deactivate. 1: Activate. (Default)
210	- - -	Option to revert back to the “old” punch algorithm. 0: Writes the “old” punch file. Default until version 5.0. This punch file does not use NDDL definitions and hence some data in the punch file can come out incorrectly. 1: Writes the "new" punch file. Default in NX Nastran version 5.0 and beyond. This uses NDDL definition and hence produces a correct punch file.
213	DISTORT	Element distortion fatal termination override. Applies to the TETRA h-elements. 0: Terminate run. (Default) 1: Do not terminate run.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
216	ITER	Controls execution of global iterative solver in SOLs 101,106,108,111,153 and the element iterative solver in SOL 101. YES: Execute global iterative solver. NO: Do not execute global iterative solver (Default). See bulk card ITER. To run the element iterative solution, see Nastran function ELEMITER.
218	T3SKEW	Allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. See the description of CTRIA3.
229	MEMSAVE	Specifies space-saving method for the old Lanczos method of eigenvalue extraction (system(273)=1). "EIGRL". 0: No space savings. (Default) 1: Does not write factors to disk which reduces scratch space usage by 67%. However, CPU costs will increase.
242	- - -	Controls module BEGN and END messages in .f04 file. 0: Print everything. (Default) 1: Print major modules only. 2: Print sub-modules only. 3: Do not print.
252	- - -	Used to select or deselect triple loop kernels in triple MPYAD operations. 0 = do not use 1 = use triple loop numerical kernels
253 - 262	- - -	SYSTEM(253) to (262) have been set aside for user DMAPS. The software does not use these values in its code in present or future versions. The SSSAlter library may use this range.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
270	QUARTICDLM	A value of 1 selects the new quartic formulation of the doublet lattice kernel (N5KQ) while 0 selects the original quadratic form. (Default=0)
273	- - -	A value of 1 selects the old Lanczos shift logic from V70 and prior systems. (Default=0)
303	MINDEF	Indefinite Mass Matrix Check, the Default=1 does not perform the check. If MINDEF>0, then check is not performed. If MINDEF<0, then epsilon = -1.E(MINDEF). If MINDEF=0, then MINDEF defaults to -6.
304	MPERTURB	Perturbation factor for indefinite mass matrix. The default=1 does not perturb the mass. If MPERTURB>0, then the mass is not perturbed. If MPERTURB<0, then the mass 1.E**(MPERTURB) is added to the diagonal terms of the mass matrix. If MPERTURB=0, then MPERTURB defaults to -6. The perturbed mass matrix is used in the subsequent eigenvalue analysis.
309	- - -	If set to 1, requests the pre-MSC.Nastran Version 70.7 HEXA8 element stiffness formulation. (Default=0)
310	- - -	If set to 1, requests the pre-MSC.Nastran Version 70.7 RBE3 formulation. (Default=0)
319	XMSG	If set to 1, gives extended error messages. If set to 0, only message summaries are reported. (Default=1)
320	- - -	Controls the conversion of DAREA Bulk Data entries for grid and scalar points to equivalent FORCE/MOMENT/SLOAD Bulk Data entries as appropriate. 0: Perform the conversion, but do not give details of the conversion. (Default) N: Perform the conversion and give details of the first N such conversions. -1: Do not perform the conversion.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
321	Q4_WARP	GEOMCHECK check value for CQUAD4 and CQUADR – surface warping factor. Default =0.05
322	Q4_IAMIN	GEOMCHECK check value for CQUAD4 and CQUADR – minimum interior angle. Default =30.0
323	Q4_IAMAX	GEOMCHECK check value for CQUAD4 and CQUADR – maximum interior angle. Default =150.0
325	T3_IAMAX	GEOMCHECK check value for CTRIA3 and CTRIAR – maximum interior angle. Default =160.0
327	BEAM_OFF	GEOMCHECK check value for CBEAM – element offset length ratio. Default =0.15
328	BAR_OFF	GEOMCHECK check value for CBAR – element offset length ratio. Default =0.15
329	MSGLIMIT	Maximum number of messages produced for each element type. Default =100
330	MSGTYPE	Controls the messages that are produced when geometry tests exceed tolerance values. 1: INFORM option produces informative messages 2: WARN option produces warning messages 3: FATAL option produces fatal messages Default =1
332	TET_EPLR	GEOMCHECK check value for CTETRA – edge point length ratio. Default =0.5
334	HEX_AR	GEOMCHECK check value for CHEXA – longest edge to shortest edge aspect ratio. Default =100.0

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
335	HEX_EPLR	GEOMCHECK check value for CHEXA – edge point length ratio. Default =0.5
337	HEX_WARP	GEOMCHECK check value for CHEXA – face warp coefficient. Default =0.707
338	PEN_AR	GEOMCHECK check value for CPENTA – longest edge to shortest edge aspect ratio. Default =100.0
339	PEN_EPLR	GEOMCHECK check value for CPENTA – edge point length ratio. Default =0.5
341	PEN_WARP	GEOMCHECK check value for CPENTA – quadrilateral face warp coefficient. Default =0.707
357	RSEQCONT	Resequence all continuations (useful to eliminate conflicts with auto-continuation capability). 0: Do not resequence continuations. (Default) 1: Resequence all continuations.
359	- - -	= 0: Use the user-requested eigensolution method. <> 0: When LAN is requested, switch to AHOU if the number of DOFs sent to the eigensolver is ≤ “nswitch”, an input parameter to the READ module. This parameter has an MPL default of 20. It may be set to other values in the solution sequences, depending on the context. When HOU, MHOU, or AHOU is selected, switch to the new Householder-QL solution. (Default=1)
368	- - -	Reserved.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
370	QRMETH	Selects the formulation used by the QUADR/TRIAR elements. 0: Selects the QUADR/TRIAR membrane formulation with active out of plane rotational dof. (default). 1: Selects the older, limited QUADR/TRIAR formulation. Complex stresses and strains are not computed. 3: Converts QUADR/TRIAR into QUAD4/TRIA3 formulation. 5: Converts QUAD4/TRIA3 into the newer QUADR/TRIAR formulation.
395-398	- - -	Reserved.
399	ELEMITER	Controls execution of element iterative solver in SOL 101. YES: Execute element iterative solver. NO: Do not execute element iterative solver (Default). Note that both ITER = YES and ELEMITER = YES are required to run element iterative solution.(See also bulk card ITER and Nastran function ITER.)
400-401	- - -	Reserved.
402	DPBLKTOL	Tolerance to determine the removal of duplicate GRID IDs and/or CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S. (Real) Set DPBLKTOL < 0.0 (default) to have the software abort a solve if a model contains duplicate GRID point IDs and/or coordinate system definitions. Set DPBLKTOL = 0.0 to have the software ignore duplicate GRID point IDs and/or coordinate system definitions and proceed with the solve. Set DPBLKTOL > 0.0 to specify a tolerance value that determines whether the software ignores

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
		duplicate GRID IDs and/or coordinate systems and proceeds with the solve. Grids: Duplicates will be removed if their coordinates are within the specified tolerance, one (or none) has a local coordinate system defined, and all belong to the same superelement. If all conditions are met, the GRID using the local coordinate system will be used and the others ignored. If all conditions are not met, the solution stops with an error. Coordinate Systems: Duplicate coordinate system definitions will be removed if the coordinate system types are the same and their transformation matrices are within the specified tolerance and they are on the same superelement. If all conditions are met, the first coordinate system will be kept and subsequent duplicates ignored. If all conditions are not met, the solution will stop with an error.
404	- - -	Debug text file (.txt) generation for RFI/MNF via MBDEXPORT/ADAMSMNF case control commands. 0: Do not generate text file (Default). 1: Generate text file.
405	- - -	Complex eigenvalue solver options using ISRR method. 0: Use ISRR special solver (higher memory requirement). (Default) 1: Use NX NASTRAN general solver (small memory requirement).
406	- - -	Complex eigenvalue solver options using ISRR method. 0: Do not calculate left eigenvectors (better performance). (Default) 1: Calculate left eigenvectors (extra computational cost).

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
407	- - -	Determines if the version stamp header is written on the first line of the OP2 file. 0: Version stamp written. (Default) 1: Turns off the version stamp.
409	- - -	Changes how the RBE3 check for unconnected grid points is done. Normally the check will produce a fatal error when grids referenced by an RBE3 are unconnected, but if this system cell is set to 1, only a warning will be issued and the solution continues. The check for an unconnected node on a RBE3 element is not performed for a superelement or dmp solution.
412	OLDGAPS	Controls how CGAPs are treated in SOL 101 when linear contact is defined. (See the section on linear contact in the *NX Nastran User's Guide* for more information.) 0: CGAPs are treated as contact elements. (Default) 1: CGAPs are treated as linear springs.
413-416		Reserved.
417	REDORTH	Controls if the reduced-orthogonalization option during Lanczos solutions is turned on or off. 0: Reduced-orthogonalization option is turned off. (Default) 1: Reduced-orthogonalization option is turned on.
419	SVDSPC	Determines which AUTOSPC method is used. 0: Use the eigenvalue method (Default) 1: Use the Singular Value Decomposition (SVD) method
421	SPARSEDR	Selects if the sparse data recovery option is used for modal response solutions 0: Sparse data recovery option not used. 1: Sparse data recovery option used. (default)

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
422	ENFMOTN	Controls which formulation is used for enforced motion response analysis (and mode acceleration, if requested). 0: Constraint mode method of enforced motion formulation (and new mode acceleration method). (default) 1: Absolute displacement enforced motion formulation (and old mode acceleration method) (ABS/REL option on DISPLACEMENT, VELOCITY, and ACCELERATION case control entries is disabled). 2: Absolute displacement enforced motion formulation is used, but includes a modal viscous damping coupling term to improve accuracy.
424	SPCHOL	Select if sparse Cholesky decomposition technique is used in linear solutions. 0: Sparse Cholesky technique is not used. 1: Use Sparse Cholesky technique. If fail, fall back to option 0. (default) 4: TAUCS is used. If fail, fall back to option 0. 5: TAUCS is used. If fail, fall back to option 1.
425	SELOPT	Controls which optimizer is selected for design optimization. (Note: previous system cell names are also valid.) 0: Use DOT (default) 1: Use SDO (Siemens Design Optimization)
426	REDMULT	Selects the REDMULT performance option when solving vibration problems with the Lanczos method. 0: REDMULT is not used.(default) 1: REDMULT is used.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
442	- - -	Controls if Solution Monitor files are generated for NX and FEMAP. It is only valid on the platforms supported by NX and FEMAP. 0: Do not generate (default). 1: Generate files and delete after solution completes. -1: Generate files and keep after solution completes.
444	TEMPWARN	Controls if temperature definitions on undefined grid points produce a fatal error or a warning. 0: A warning occurs. (default) 1: A fatal error occurs.
445	NREC	Defines the number of substructures in RDMODES. The efficiency of RDMODES requires carefully choosing m, which should not be too small or too large. A nrec value with power of 2 is preferable for better performance. For a large problem (>1M DOFs), nrec=128 or 256 is likely to be a good choice. When NREC=1, the software will automatically compute a reasonable number of partitions based on your model size. The resulting number of partitions will be reported in the .f06 file. When NREC>1, the software will use the specified value as the number of partitions.
446	RDSCALE	A factor to modify the selected frequency range in the EIGRL specification for eigensolutions of each substructure in RDMODES. The default value is 1.0 for single-level RDMODES, i.e., using the same frequency range as EIGRL specification; and is 2.0 for multilevel RDMODES.
447	QDX_SKEW	GEOMCHECK check value for CQUADX4, CQUADX8, CPLSTS4, CPLSTS8, CPLSTN4, and CPLSTN8 – skew angle. Default =30.0

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
448	QDX_TAPR	GEOMCHECK check value for CQUADX4, CQUADX8, CPLSTS4, CPLSTS8, CPLSTN4, and CPLSTN8 – taper ratio. Default =0.5
449	QDX_IAMN	GEOMCHECK check value for CQUADX4, CQUADX8, CPLSTS4, CPLSTS8, CPLSTN4, and CPLSTN8 – minimum interior angle. Default =30.0
450	QDX_IAMX	GEOMCHECK check value for CQUADX4, CQUADX8, CPLSTS4, CPLSTS8, CPLSTN4, and CPLSTN8 – maximum interior angle. Default =150.0
451	QDX_AR	GEOMCHECK check value for CQUADX4, CQUADX8, CPLSTS4, CPLSTS8, CPLSTN4, and CPLSTN8 – longest edge to shortest edge aspect ratio. Default =100.0
452	QDX_EPLR	GEOMCHECK check value for CQUADX8, CPLSTS8, and CPLSTN8 – edge point length ratio. Default =0.5
454	TRX_IAMN	GEOMCHECK check value for CTRAX3, CTRAX6, CPLSTS3, CPLSTS6, CPLSTN3, and CPLSTN6 – minimum interior angle. Default =30.0
455	TRX_IAMX	GEOMCHECK check value for CTRAX3, CTRAX6, CPLSTS3, CPLSTS6, CPLSTN3, and CPLSTN6 – maximum interior angle. Default =160.0
456	TRX_AR	GEOMCHECK check value for CTRAX3, CTRAX6, CPLSTS3, CPLSTS6, CPLSTN3, and CPLSTN6 – longest edge to shortest edge aspect ratio. Default =100.0
457	TRX_EPLR	GEOMCHECK check value for CTRAX6, CPLSTS6, and CPLSTN6 – edge point length ratio. Default =0.5
459		Reserved.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
461	- - -	For each constraint equation (user defined or generated from a rigid element), the software checks the magnitude of the coefficient of the dependent dof. If the value is relatively small, a warning message is output. If the value is very small, fatal message 5289 is issued. The reason for the check is that this value will be used in a denominator of an equation and, if very small, can lead to numerical round-off errors and poor answers. In rare occasions it might be permissible to allow this to happen and thus if system cell 461 is set to a “1”, the message will only be a warning. 0: A fatal error occurs (default). 1: A warning occurs.
462	- - -	Methods for SOL 111 0 (Default) The software automatically selects either the original FRRD1 method or the in-core FRRDRU method. • The software selects the original FRRD1 method if the number of modes < PARAM,FRUMIN (default=500), or if the model includes any of the following: frequency dependent properties or materials, unsymmetric damping, or consist of only fluid elements. • The software selects the in-core FRRDRU method if the model does not include any of the items listed above and the number of modes ≥ PARAM,FRUMIN. Note: You can optionally set PARAM,FRRU,NO and SYSTEM(462)=0 to turn off the automatic selection and the original FRRD1 method is always used.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
		1: The software automatically selects either the original FRRD1 method, the in-core FRRD1 method, or the in-core FRRDRU method. • The software selects the original FRRD1 method if the model includes any of the following: frequency dependent properties or materials, unsymmetric damping, or consist of only fluid elements. • If the model does not include any of the items listed above and the number of modes < PARAM,FRUMIN (default=500), the software selects the in-core FRRD1 method if their is enough memory to solve in-core. Otherwise, the original FRRD1 method is selected. • The software selects the in-core FRRDRU method if the model does not include any of the items listed above and the number of modes ≥ PARAM,FRUMIN. Note: If you define SYSTEM(262)=YES and the in-core FRRD1 method is selected by the software, an in-core iterative method is used.
		2: Original FRRDRU method is used. This method cannot handle multiple eigenvalues (either repeated roots or rigid body modes). If multiple eigenvalues occur, the method defaults to the original FRRD1 method.
		4: Enhanced FRRDRU method is used. This method allows multiple eigenvalues and sets KDAMP=-1.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
463	- - -	Determines if CBEAM elements with offset vectors are permitted in solutions 106, 129, 153, and 159. See remarks on the CBEAM bulk entry. Also determines if shell elements with ZOFFS should be permitted in nonlinear solutions 106 and 129 in combination with thermal loads. See remarks on the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR bulk entries. 0: A fatal error occurs (default). 1: Solution is permitted to continue, but CBEAM offset vectors remain parallel to their original orientation.
464	- - -	Determines the CGAP normal and transverse stiffness when a BCSET case control command exists (contact conditions exist). 0: (default) Stiffness is determined from the properties on the PGAP bulk data entry. (See remarks on the PGAP bulk data entry.) 1: Stiffness is determined from PENN and PENT on the BCTPARM bulk data entry.
466	PYR_AR	GEOMCHECK check value for CPYRAM – longest edge to shortest edge aspect ratio. Default =100.0
467	PYR_EPLR	GEOMCHECK check value for CPYRAM – edge point length ratio. Default =0.5
469	PYR_WARP	GEOMCHECK check value for CPYRAM – face warp coefficient. Default =0.707

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
474	- - -	Determines the CROD and CBAR element lumped mass calculation. The “old” calculation before NX Nastran 6.1 was inconsistent with the CBEAM. 0: CROD and CBAR lumped mass calculation is consistent with CBEAM element (default). 1: CBAR will use the old method. 2: CROD will use the old method. 3: CBAR and CROD will use the old method.
476	- - -	Determines if a non-converged contact condition in solution 101 continues with a warning, or ends with an error. 0: (default) A warning is reported in the *.f06 file, the solution continues and results creation occurs. 1: A fatal error is reported in the *.f06 file, and the solution ends immediately with no results creation.
480	- - -	Determines the linear contact processing defaults. See the BCTPARM bulk entry. (default): – The elastic modulus of the softer material in each contact pair is used. – PENN and PENT are automatically calculated. – The "REFINE=2" source-refinement algorithm is used. 1: – The modulus averaged over the entire model is used. – The automatic penalty factor calculation is turned off and PENN and PENT are used. – The NX Nastran 6.1 source-refinement algorithm is used.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
489	PERPTOL	Determines the perpendicularity tolerance for shell element material orientations. (Real, Default = 1.0E-8 degrees) A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value of SYSTEM(489).
491	- - -	When the GLUETYPE=2 option is used on the BGPARM bulk entry, reverts the surface-to-surface glue stiffness back to the NX Nastran 6.1 defaults. 3 (default): Use current stiffness defaults. 1: use the NX Nastran 6.1 stiffness defaults.
492	- - -	Determines whether the factor caching for the Lanczos real eigensolver is used. 0: No factor caching (default). 1: The factor caching is used.
495	- - -	Determines if a shell element thickness value of “0.0” causes a fatal error. 0 (default): A fatal error will occur if any Ti value of “0.0” exists on the elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8. This thickness check default began in NX Nastran 7.1. 1: The solution proceeds as it did before version 7.1.
500	- - -	Determines which strains are reported as total strains in the results under the label “NONLINEAR STRESSES”. 0 (default): Report the sum of mechanical strains and thermal strains as total strains. 1: Report only the mechanical strains as total strains. This is the default behavior of versions prior to NX Nastran 8.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
503	- - -	On the ACSRCE bulk entry, determines if the scale factor A is included inside or outside of the radical. See the ACSRCE bulk entry for the source strength equation. 1 (default): The scale factor A is included outside of the radical. 0: The scale factor A is included inside of the radical. This is the behavior before version 8.
509	- - -	When ND is specified on the EIGRL card, determines whether to increase ND automatically to include all roots in a cluster. If ND is not permitted to increase (default), only part of a cluster of multiple eigenvalues is computed, the modal space will not be uniquely defined. In this scenario, results computed from that modal space may be unstable (frequency response, transient response, CMS solution, etc.). 0 (default): Compute ND roots as requested. 1: If necessary, increase ND automatically.
516	- - -	When gluing solid element faces, determines if the solid element face normal check occurs. See remarks on BGSET case control command for more information. 0 (default): Solid element face normal check occurs. 1: Solid element face normal check is off.
519	- - -	0 (default): Automatically supply blank fields 5 through 8 of a wide field card if only fields 1 through 4 are provided. 1: Do not automatically supply fields 5 through 8 (retained to preserve old behavior of NX Nastran).

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
524	RANFRF	Flag to output frequency response in addition to any PSD output 0 (default): Do not output frequency response in addition to PSD output. 1: Output frequency response in addition to PSD output.
525	- - -	Enables integer inputs to be 11 characters. 0 (default): Integer inputs are limited to 8 digits, even when using the ILP-64 executable and large field formatted bulk entries. The maximum ID of grid points or elements is then 99999999. 1: Integer inputs can be defined up to 11 digits when using the ILP-64 executable and large field formatted bulk entries. The maximum ID of grid points or elements is then 99999999999. Note: When using the ILP-64 executable and SYSTEM(525)=1 is defined, the OP2FMT parameter is ignored, and the software writes the OP2 file as 64-bit. See OP2FMT.
526	- - -	In NX Nastran 8.1, the torsional constant calculation for the cross-section types “CHAN1” and “H” on the PBARL/PBEAML bulk entries was updated. 0 (default): The new torsional constant calculation occurs. 1: Reverts to the previous torsional constant calculation.
528	- - -	Allows you to revert to the glue stiffness calculations before NX Nastran 8.5. 0 (default): Use the average modulus (E) in the source region when computing the glue stiffness. Also use the improved linear shell glue stiffness. 1: Use the average modulus (E) of the entire model when computing the glue stiffness. Also use the old linear shell glue stiffness. This is the behavior before version 8.5.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
539	- - -	Selects optimization enhancement level. 0: Selects NX Nastran 8.1 enhancement level. 1: Selects NX Nastran 8.5 enhancement level. 2: Selects NX Nastran 8.1 enhancement level plus selective modification of design variable bounds. 3 (default): Selects NX Nastran 8.5 enhancement level plus selective modification of design variable bounds. For details on these enhancement levels, consult the appropriate *NX Nastran Release Guides*.
542	- - -	Selects SSG1 method. 0: Original method. (Expect slow performance when this method is used with models that contain very large numbers of PLOAD2 bulk entries.) 3 (default): Enhanced method. (Better suited for models that contain very large numbers of PLOAD2 bulk entries.)
544	HEX_TP	GEOMCHECK check value for CHEXA – Ratio of longest edge to shortest edge in stacking direction for CHEXA elements that reference PCOMPS entries only. Default =1.2
545	PEN_TP	GEOMCHECK check value for CPENTA – Ratio of longest edge to shortest edge in stacking direction for CPENTA elements that reference PCOMPS entries only. Default =1.2
548	HEX_TK	GEOMCHECK check value for CHEXA – Ratio of the difference between thickness in stacking direction as defined by grids and as defined by ply thickness specification to the thickness in the stacking direction as defined by grids for CHEXA elements that reference PCOMPS entries only. Default =0.05

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
549	PEN_TK	GEOMCHECK check value for CPENTA – Ratio of the difference between thickness in stacking direction as defined by grids and as defined by ply thickness specification to the thickness in the stacking direction as defined by grids for CPENTA elements that reference PCOMPS entries only. Default =0.05
550	RMSVM	Determines if the RMS von Mises stress is computed when the stress RMS output is requested with the RMS describer on the STRESS case control command. The data blocks OESXRMS1 and OESXRMS2 are written when the RMS von Mises stress is computed. 0 (default): The RMS von Mises stress is computed, and the data blocks OESXRMS1 and OESXRMS2 are written. 1: The RMS von Mises stress is not computed, and the data blocks OESRMS1 and OESRMS2 are written.
552	F56	Defines the Fortran unit number for the .f56 output file. The .f56 file is requested with the PARAM,F56,YES setting in the bulk data. Default=56
553	QAD_AR	GEOMCHECK check value for CQUAD4 element - longest edge to shortest edge aspect ratio. Default =100.0
560	PCHLN	Determines if the line numbers are included in the PUNCH file. 0 (default): The line numbers are on. 1: The line numbers are turned off.
563	Q8_TAPER	GEOMCHECK check value for CQUAD8 element - taper ratio. Default =0.5
564	Q8_IAMIN	GEOMCHECK check value for CQUAD8 element - minimum interior angle. Default =30.0

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
565	Q8_IAMAX	GEOMCHECK check value for CQUAD8 element - maximum interior angle. Default =150.0
566	Q8_AR	GEOMCHECK check value for CQUAD8 element - longest edge to shortest edge aspect ratio. Default =100.0
567	Q8_EPLR	GEOMCHECK check value for CQUAD8 element - edge point length ratio. Default =0.5
568	Q8_SKEW	GEOMCHECK check value for CQUAD8 element - skew angle. Default =30.0
569	TA6_AR	GEOMCHECK check value for CTRIA6 – longest edge to shortest edge aspect ratio. Default =100.0
570	TA6_EPLR	GEOMCHECK check value for CTRIA6 – edge point length ratio. Default =0.5
571	TA6_IAMN	GEOMCHECK check value for CTRIA6 – minimum interior angle. Default =10.0
572	TA6_IAMX	GEOMCHECK check value for CTRIA6 – maximum interior angle. Default =160.0
579	FREQVM	Determines whether the von Mises stress and strain are computed for a deterministic frequency response analysis in SOL 108 or SOL 111. 0 (default): von Mises stress and strain are computed and, along with the other stress and strain results, written to the OESVM and OSTRVM output data blocks. 1: von Mises stress and strain are not computed. Other stress and strain results are written to the OES and OSTR output data blocks.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
580	- - -	Suppresses warning messages from the DMAP compiler that arise because of undefined input or output to DMAP commands. 0 (default): Allow warning messages 1: Suppress warning messages
581	- - -	Turns off the error trap which occurs when an SPCD entry is defined with a bolt preload. 0 (default): A fatal error occurs when an SPCD bulk entry and a bolt preload are both defined. 1: No fatal error occurs. Solution proceeds when an SPCD is defined with a bolt preload.
583	- - -	Controls the maximum number of flutter iterations in SOL 145. Default = 50.
587	- - -	Determines the sectional properties for the axisymmetric elements CTRAX3, CQUADX4, CTRAX6, and CQUADX8. 0 (default): Stiffness, mass and loads are based on a 2*PI section (beginning with NX Nastran 10). 1: Stiffness, mass and loads are computed on a per radian basis (behavior prior to NX Nastran 10), except for heat transfer solutions with CTRAX3, CQUADX4, CTRAX6, and CQUADX8 which remain on a 2*PI basis. See “Axisymmetric Element Sectional Properties” in the *Element Library Reference*.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
589	- - -	Controls the drilling degree-of-freedom on QUADR/TRIAR elements. 0: The drilling degree-of-freedom is active for all QUADR/TRIAR elements in the model (Default). 1: The drilling degree-of-freedom is deactivated for all QUADR/TRIAR elements in the model. 2: The drilling degree-of-freedom is deactivated for the QUADR/TRIAR elements which have membrane stiffness only (MID2 and MID3 are blank on the PSHELL entry).
612	- - -	Requests the form of the mass matrix for a static subcase in SOL401. 0 (default): Use a consistent mass matrix. 1: Use a lumped mass (form used before version 10).
617	ACFORM	Selects acoustic formulations. 0 = NX Nastran base acoustic capabilities including element formulation are used. The features such as AML boundary condition, infinite plane, porous materials and microphone elements are NOT available. Set the system cell to this value to obtain results identical to prior versions or releases of NX Nastran. 1 (default) = Allows extended acoustic capabilities such as AML, porous materials and microphone elements. All extended capabilities are available only in SOLs 108 and 111.

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
634	- - -	Controls the number of times the software prints the message number 2083.3 into the .f06 file. Message number 2083.3 is printed for a shell element which incorrectly has an associated PSHELL with the MID3 field referencing a MATS1 bulk entry. Only the MID1 and MID2 should reference a MATS1. > 0 Limits the number of messages. Default = 100. = 0 The message is not written. = -1 The message is written for all.
635	Q8T6_ANG	Controls the maximum allowable angle between normals to corner grids for CQUAD8 and CTRIA6 elements. USER WARNING MESSAGE 5276 is issued if the the maximum allowable angle is exceeded. Default =30.0
636	TET_EPIA	GEOMCHECK check value for CTETRA – edge point included angle. Default =30.0
637	HEX_EPIA	GEOMCHECK check value for CHEXA and CHEXCZ – edge point included angle. Default =30.0
638	PEN_EPIA	GEOMCHECK check value for CPENTA and CPENTCZ – edge point included angle. Default =30.0
639	PYR_EPIA	GEOMCHECK check value for CPYRAM – edge point included angle. Default =30.0

Table 1-1. System Cell Summary

System Cell Number	System Cell Name	Function and Reference
642		In a SOL 401 analysis which includes a sliding contact condition, the SEPDIS describer on the BCRESULTS case control command can be defined to request the total and incremental slide or slip distance. When system 642 = 1 (default), the output is the slide distance. When system 642 = 0, the output is the slip distance. See the BCRESULTS case control command and the *Multi-Step Nonlinear User's Guide* for details.

Chapter 2: File Management Statements

2.1 Key to Descriptions

Brackets [ ] indicate that a choice of describers is optional.

A brief sentence about the function of the statement is given.

If the describers are stacked vertically, then only one may be specified.

RESTART Reuse Database From a Previous Run

Requests that data stored in a p run.

Describers in uppercase letters are keywords that must be specified as shown.

The default describers are shaded.

Format:

RESTART [PROJECT = 'project-ID' VERSION = { version-ID / LAST } { KEEP / NOKEEP }]

Describers in lower case are variables.

Braces { } indicate that a choice of describers is mandatory.

Describers:

project-ID Project identifier. Se t be e marks. (Character string, maximum of 40 characters; specified on the PROJ FMS statement).

version-ID Version number. (Integer > 0).

LAST Specifies the last version under project-ID.

ERSION will remain on the database after the run is

ERSION will be deleted from the database after the run is

Each of the describers is discussed briefly. Further details may be discussed under Remarks.

Remarks:

1. There may be only on
2. A new version-ID is a
3. If project-ID or versio Message will be issue
4. The RESTART statement is required to perform restarts in all structured solu- 0) and in unstructured solution sequences (21,

en the run will restart from the project-ID speci- ee Example 2 below.)

If the describer is in lower case, then it is a variable and the describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the describers.

1. RESTART VERSION=7

   Version number 7 will be retrieved for this run (version 8). At the end of the run version 7 will be deleted.

2. PROJ='FENDER'
   RESTART

   The last version under project-ID FENDER will be used in the current run.

The File Management Section (FMS)

The File Management Section (FMS) is primarily intended for the attachment and initialization of Database sets (DBsets) and FORTRAN files. The initialization of DBsets includes specification of their maximum size, member names, and physical filenames. The initialization of FORTRAN files includes the specification of their filenames, FORTRAN unit numbers, and FORTRAN attributes.

In most classes of problems that use NX Nastran solution sequences (SOL), no File Management statements are required because a default File Management Section is executed at the beginning of every run. If a restart is desired, then the RESTART statement is required in SOL 101 through 200. All other solutions may not be restarted. If the problem is large in terms of requiring significant amounts of memory or disk space, then the INIT, ASSIGN, and EXPAND statements may be required. If any FORTRAN files are required, then the ASSIGN statement is required; for example, the OUTPUT2 DMAP module. The ASSIGN statement is also required to assign databases for DBLOCATE, DBLOAD, and DBUNLOAD. Special database operations are performed by the DBLOCATE, DBLOAD, DBUNLOAD, DBLCLEAN, ACQUIRE, DBDIR, DBDICT, DBFIX, DBSETDEL, DBUPDATE, and PROJECT statements.

File Management Statement Summary

The following is a summary of all File Management statements:

$	Comment statement.
ACQUIRE	Selects NDDL schema and NX Nastran Delivery Database.
ASSIGN	Assigns physical files to DBset members or special FORTRAN files.
CONNECT	Groups geometry data by evaluator and database.
DBCLEAN	Deletes selected database version(s) and/or projects.
DBDICT	Prints the database directory in user-defined format.
DBDIR	Prints the database directory.
DBFIX	Identifies and optionally corrects errors found in the database.
DBLOAD	Loads a database previously unloaded by DBUNLOAD.
DBLOCATE	Obtains data blocks and parameters from databases.
DBSETDEL	Deletes DBsets.
DBUNLOAD	Unloads a database for compression, transfer, or archival storage.
DBUPDATE	Specifies the time between updates of the database directory.
ENDJOB	Terminates a job upon completion of FMS statements.
EXPAND	Concatenates additional DBset members to an existing DBset.
INCLUDE	Inserts an external file in the input file.

INIT	Creates a temporary or permanent DBset.
NASTRAN	Specifies values for system cells.
PROJ	Defines the current or default project identifier.

The FMS statements are executed in the following order regardless of their order of appearance in the input file:

NASTRAN, DEFINE

RFINCLUDE, INCLUDE

ASSIGN, INIT, EXPAND, DBUPDATE

PROJECT

DBCLEAN

DBFIX

DBDIR (1), DBDICT(1)

DBSETDEL

ACQUIRE

RESTART

DBLOCATE

DBUNLOAD

DBLOAD

DBDIR (2), DBDICT(2)

ENDJOB

If DBDIR or DBDICT is specified before any of the FMS statements DBSETDEL through DBLOAD, then the directory printout will reflect the processing of DBCLEAN and DBFIX only. If DBDIR or DBDICT is specified after DBSETDEL through DBLOAD, then the directory printout will reflect the processing of all statements in the FMS Section. It is recommended that the DBDIR and DBDICT statements be specified last in the FMS Section. Multiple DBLOCATE, DBLOAD, or DBUNLOAD statements are processed in the order in which they appear. If the ENDJOB statement is specified, then only the File Management Section is processed and the Executive Control, Case Control, and Bulk Data Sections are ignored.

2.2 File Management Statement Descriptions

File Management statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

Description

A brief sentence about the function of the statement is given.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma as long as the comma is preceded by one or more spaces. For example:

```
DBLOCATE DATABLK=(KAA)  ,
   WHERE(PROJECT='FRONT BUMPER' AND  ,
   SEID>0 AND VERSION=4)  ,
   LOGI=MASTER3
```

However, if a filename is to be continued on the next line, no space must precede the comma, and the continuation line must have no leading spaces.

For example:

```
ASSIGN     SDB='/jw/johannes/Projects/secret/Aero/Tests/wing/Modes/w,
ing_modal.MASTER'
```

Note that all quotation marks shown under formats and examples are right-handed single quotation marks and must be entered as such. For example:

```
PROJ='MYJOB'
```

Examples

One or more examples are given.

Describers

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The remarks are generally arranged in order of importance and indicate such things as the FMS statement's relationship to other commands, restrictions and recommendations on its use, and further descriptions of the describers.

WHERE and CONVERT Clauses

The WHERE clause is used in the selection of items (data blocks and parameters) on the DBDICT, DBLOCATE, DBLOAD, and DBUNLOAD statements. The CONVERT clause modifies qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements.

The WHERE and CONVERT clauses specify values for PROJECT, VERSION, qualifiers, and DBSET. PROJECT specifies the project-ID that is originally defined on the PROJECT FMS statement at the time the project is created. VERSION specifies the desired version-ID under the project-ID. Qualifiers are used to uniquely identify items on the database with the same name. For example, data block KAA has SEID as one of its qualifiers, which is the superelement ID. An item may have more than one qualifier and the collection of all qualifiers assigned to an item is called a path.

All data blocks and parameters with qualifiers are defined in the NDDL (NASTRAN Data Definition Language) sequence. See the *NX Nastran DMAP Programmer's Guide*. Data blocks and parameters are defined on the DATABLK and PARAM NDDL statements. The DATABLK and PARAM statements specify the name of the data block, parameter, and also its pathname. The pathnames are defined on the PATH NDDL statement, which lists the qualifiers assigned to the path. Qualifiers are defined on the QUAL NDDL statement. DBSET specifies the desired DBset. The DBset of an item is specified after the LOCATION keyword on the DATABLK and PARAM NDDL statement.

The format of the WHERE clause is:

WHERE (where-expr)

where-expr is a logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. If the result of the logical expression is TRUE for an item on the database then the item is selected. For example, WHERE(VERSION=4 AND SEID < > 2 AND SEID > 0) selects all items under version 4 for all values of SEID greater than 0 except 2.

A simple where-expr is a comparison using the following relational operators = , > , < , < = , > =, >< or <>. For example, SEID > 0 means if SEID is greater than zero, then the logical expression is true. Several simple where expressions may be joined into a single where-expression using the following logical operators: AND, OR, XOR, and EQV. The NOT operator may be used to negate a where expression. For example, NOT(SEID > 0) is the same as SEID < = 0. Arithmetic operations and DMAP functions may also be specified in the where-expression. (See the *NX Nastran DMAP Programmer's Guide* .)

If a qualifier in a where-expr is not a qualifier in the path of a specified item, then the where-expr is set to FALSE. If the where-expr does not contain a specification for all qualifiers in the path of an item, then the unspecified qualifiers will be wildcarded, (i.e., quali= * , all values will be selected). The default values of qualifiers, PROJECT, VERSION, and DBSET are described under the statement in which the WHERE clause is specified.

Examples of the WHERE clause are:

1. Select all items in the database for all superelements except 10 and 30 from Version 1.

```
WHERE (VERSION=1 AND SEID>=0 AND NOT(SEID=10 OR SEID=30))
```

2. Select all entries in the database on DBSET=DBALL from all projects and versions.

```
WHERE(PROJECT=PROJECT AND VERSlON>0 AND DBSET='DBALL')
```

The CONVERT clause modifies project- and version-ID, DBset-name (see INIT statement), and qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements. It contains one or more assignment statements separated by semicolons. The format of the CONVERT clause is:

CONVERT(PROJECT=project-expr; VERSION=version-expr; ,

The PROJECT and VERSION statements modify the project-ID (see PROJECT FMS statement) and version-ID. The DBSET statement modifies the DBset-name. The value of quali will be replaced by qual-expri for selected items that have quali in their path. qual-expri is any valid expression (see “Expressions and Operators” in the *NX Nastran DMAP Programmer’s Guide*) containing constants or any qualifier name defined in the path of the item. If qual-expri contains names of qualifiers not in the path of the selected item, then a fatal message is issued. If project-expr and/or version-expr produces a project- or version-ID which does not exist, then one will be created. Also, all version-IDs less than version-expr that do not exist will be created; but they will be “empty.”

Examples of the CONVERT clause are:

1. Set qualifiers SEID, PEID, and SPC to constants 10, 20, 102 respectively.

```
CONVERT(SEID=10;PEID=20;SPC=102)
```

   If more than one value of a qualifier is found for an item by the WHERE clause, then each value is processed in qual-expri to define the new qualifier value for each of the selected items. In the example below, if the original values of PEID were 1, 2, and 3; then the new values for the SEID qualifier will be 2, 4, and 6.

2. Set all values of qualifier SEID to be twice that of the values of qualifier PEID.

```
CONVERT(SEID=2*PEID)
```

$

Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

FORMAT:

$ followed by any characters out to column 80.

EXAMPLES:

```
$ TEST FIXTURE-THIRD MODE
```

REMARKS:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

ACQUIRE

Selects NDDL Schema

Selects the NDDL schema and NX Nastran delivery database to be used for primary database creation.

FORMAT:

$$\text{ACQUIRE NDDL} = \left\{ \begin{array}{c} \text{NDDL} \\ \text{nddl-name} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
NDDL	NX Nastran NDDL schema.
nddl-name	Name of a user NDDL schema specified on a COMPILE NDDL statement when the user NDDL was stored.

REMARKS:

This statement is used to specify the delivery database when the user wishes to create his or her own solution sequence, yet use the subDMAP objects or NDDL schema or both from the delivery database supplied with the software.

EXAMPLES:

The following requests the NX Nastran NDDL schema to be used when creating a new database.

```
ACQUIRE NDDL
SOL MYDMAP
COMPILE DMAP=MYDMAP,SOUOUT=USROBJ
.
.
.
LINK MYDMAP,SOLOUT=USROBJ
```

ASSIGN

Assigns Physical File

Assigns physical file names to DBset members or special FORTRAN files that are used by other FMS statements or DMAP modules.

FORMAT:

Format 1

Assign a DBset member name

ASSIGN log-name='filename1' [TEMP DELETE SYS='sys-spec']

Format 2

Assign a FORTRAN file

$$\text{ASSIGN} \quad \text{logical-key} = \text{'filename2'}$$

$$\left[\text{STATUS} = \left\{\begin{array}{c}\text{NEW}\\ \text{OLD}\\ \text{UNKNOWN}\end{array}\right\}\right]$$

$$\left[\text{FORM} = \left\{\begin{array}{c}\text{FORMATTED}\\ \text{UNFORMATTED}\\ \text{LITTLEENDIAN}\\ \text{BIGENDIAN}\end{array}\right\}\right]$$

$$[\text{UNIT} = u]\,[\text{TEMP}]\,[\text{DELETE}]\,[\text{SYS} = \text{'sys-spec'}]$$

EXAMPLES:

1. Assign the DBALL DBset:

```
ASSIGN DB1='filename of member DB1'
```

```
INIT DBALL LOGI=(DB1)
```

2. Assign FORTRAN file 12 to the OUTPUT4 module using the ASCII option:

```
ASSIGN OUTPUT4='filename of FORTRAN file'
UNIT=12, FORM=FORMATTED
```

DESCRIBERS:

Describer	Meaning
log-name	The name of a DBset member name. log-name is also referenced on an INIT statement after the LOGICAL keyword.
filenamei	The physical filename assigned to the DBset member.
TEMP	Requests that filenamei is deleted at the end of the run.
DELETE	Requests that filenamei, if it exists before the start of the run, is to be deleted.
logical-key	Specifies defaults for STATUS, UNIT, and FORM of FORTRAN files for other FMS statements, DMAP modules, punching and plotting operations.
filename2	The physical file name assigned to the FORTRAN file.
STATUS	Specifies whether the FORTRAN file is being created (STATUS=NEW) or has been created prior to the run (STATUS=OLD). If its status is not known, then STATUS=UNKNOWN is specified.
UNIT=u	u is the FORTRAN unit number of the FORTRAN file. See Remark 9.
FORM	Indicates whether the FORTRAN file is written in ASCII (FORM=FORMATTED), binary (FORM=UNFORMATTED), little endian binary (LITTLEENDIAN), or big endian binary (BIGENDIAN) format.
DEFER	The specified file will not be opened/created during NX.Nastran initialization. The file must be opened by the module or DMAP accessing the file.

Describer	Meaning
sys-spec	System specific or machine-dependent controls. See Remark 1 .

REMARKS:

1. The ASSIGN statement and its applications are discussed further in “Introduction to Database Concepts” in the *NX Nastran User’s Guide*.

2. The logical-key names and their default attributes that may be assigned by the user are as follows:

logical-key Name	Default Status	Default Unit	Default Form	Application
DBC	NEW	40	UNFORMATTED	DBC module — PARAM,POST,0
INPUTT2	OLD	none	UNFORMATTED	INPUTT2 module
INPUTT4	OLD	none	UNFORMATTED	INPUTT4 module
OUTPUT2	NEW	none	UNFORMATTED	OUTPUT2 module
OUTPUT4	NEW	none	UNFORMATTED	OUTPUT4 module
DBUNLOAD	NEW	50	UNFORMATTED	DBUNLOAD FMS statement
DBLOAD	OLD	51	UNFORMATTED	DBLOAD FMS statement
USERFILE	none	none	none	User-defined

   The defaults may be overridden on the ASSIGN statement.

3. Certain reserved names may not be used for log-names or logical-key names: SEMTRN, LNKSWH, MESHFL, LOGFL, INPUT, PRINT, INCLD1, and CNTFL. If they are used, then a fatal message is issued. Also, unit numbers 1 through 10, 14 and 16 should not be assigned. PUNCH and PLOT may be used but are not recommended.

4. If one of the logical-key names indicated in Remarks 2. and 3. is not specified on this statement, then it is assumed to be a DBset member log-name as shown in Format 1.

5. The logical-key names DBUNLOAD and DBLOAD may be assigned only once in the FMS section. The others may be assigned as many times as needed for the application. However, in all logical-key assignments, the unit number u must be unique. If it is necessary to execute the INPUTT4 and OUTPUT4 modules on the

same unit, then specify ASSIGN OUTPUT4 only. The same is recommended for the INPUTT2 and OUTPUT2 modules.

6. STATUS, UNIT, and FORM are ignored if assigning a log-name (DBset member name).

7. FORM=FORMATTED must be specified for a unit when:

   - Ascii output is desired on the INPUTT4 and OUTPUT4 DMAP modules that process the unit. See the *NX Nastran DMAP Programmer's Guide*.

   - FORMAT=NEUTRAL is selected on the DBUNLOAD and DBLOAD FMS statements that process the unit. See "Database Archival and Retrieval" in the *NX Nastran User's Guide*.

   - The neutral file format is desired for the INPUTT2 and OUTPUT2 modules.

8. See the *NX Nastran Installation and Operations Guide* for further information on machine-dependent aspects of the ASSIGN statement.

9. The following unit numbers are reserved for specific tasks. Only use these numbers when performing the associated task. For example, UNIT=7 is used for punch files.

```
ASSIGN PUNCH='results_output.pch',NEW,UNIT=7
```

UNIT NO	DESCRIPTION
1	Default UNIT for SCR.F01
2	Default UNIT for SCR.F02
3	Mesh file SCR.F03
4	Log file (typically standard out)
5	Input (deck) file
6	Print file (f06 file)
7	Punch file
8	Node-locked license file
9	Include files
10	Control file or the ASG file
11	INPUTT2 unit
12	OP2 file
14	Plot file
15	Mesh file
16	Assembly file
18	Acoustics coupling file - 1

19	Acoustics coupling file - 2
20	Data base migration files
21	Adina input file
22	Adina output file - 1
23	Adina output file - 2
24–29	BCS linear solver
24–33	BCS eigensolver
31–35	Reserved for AMLS files
40	DBC (xdb) File1
41	Reserved for SML (SAMCEF Material Library)
42	Reserved for SML (SAMCEF Material Library)
43	Reserved for SML (SAMCEF Material Library)
44	Reserved for SML (SAMCEF Material Library)
45	Reserved for SML (SAMCEF Material Library)
46	Reserved for AMLS files
50	DBC (xdb) File2
51	DBC (xdb) File3
56	Print file (f56 file)
66	Linear contact refinement file
70	Reserved for Akusmod files
71-80	Reserved for BUN files (TEMPEX and DTEMPEX)
81-90	Reserved for SML (SAMCEF Material Library)
151–154	Solution monitor files

CONNECT

Group Evaluator Data

Defines a group of beam cross-section entities, or a DRESP3 second order response function external evaluator. These entities should belong to the same evaluator-class (set of routines that process them). The PBARL, PBEAML, and DRESP3 Bulk Data entries refer to the groups defined here.

FORMAT:

$$\text{CONNECT} \begin{bmatrix} \text{BEAMEVAL} \\ \text{DRESP3} \end{bmatrix} \quad \text{group} \quad \text{evaluator} \quad \text{'path'} \quad \text{'data'}$$

DESCRIBERS:

Describer	Meaning
group	Group name referenced by the GROUP field on PBARL, PBEAML, and DRESP3 Bulk Data entries.
evaluator	Identifies the particular class of evaluator to which the beam cross-section entities belong, or is an alias for the path name to the DRESP3 evaluator. Entities belonging to one evaluator-class are handled by the same set of routines (either provided with the software or user-provided). For beam cross-sections, the class MSCBML (NX Nastran Beam Library) is provided internally. For the DRESP3 response evaluation, template subroutines are provided with NX Nastran, so that the users can customize them accordingly, modifying them and/or adding more subroutines called from these. Users may develop custom evaluator libraries for beam cross-sections, or DRESP3 functions and configure them for use with NX Nastran. See Remarks 2, 3 and, 4.
path	Optional pathname or filename used by evaluator. Path must be enclosed by single quotation marks if it contains lowercase characters.

Describer	Meaning
data	Optional character string passed to the evaluator. Data must be enclosed within single quotation marks if it contains lowercase characters or embedded blanks.

REMARKS:

1. CONNECT requests:
   - an external database or evaluator,
   - a user defined grouping for beam cross-section data defined by PBARL and PBEAML entries.
2. A single reserved group name, MSCBML0 has been predefined for beam cross-section entities. It may be used in the PBARL and PBEAML entries, without being defined explicitly by means of a CONNECT FMS statement. It corresponds to the MSCBML (Beam-Library) evaluator.
3. Custom beam cross-section or DRESP3 evaluator libraries developed by users should comply with the guidelines outlined in the *NX Nastran User's Guide*.
4. Once developed, an evaluator may be configured as:
   - internal, where the evaluator routines are linked with the rest of the NX Nastran object modules to comprise the NX Nastran executable program,
   - or external, where the evaluator routines are linked with a provided server program to constitute an independent geometry server, or DRESP3 server.
5. The NX Nastran provided DRESP3 external program development capability will result in an external program with the default name dr3serv.exe. Refer to Version 5.1 Release Guide and the DRESP3 bulk data definition on how to build and run this external program together with NX Nastran.
6. The file which has the resolution of the alias for the DRESP3 evaluator path name should be referred to from the nastran command line by the gmconn keyword. For example, if the file name is xyz, then gmconn = xyz should be added to the nastran command. Thus, the nastran command refers to the file which contains the resolution of the alias for the path to the evaluator, and the alias is contained in the CONNECT statement, completing the information necessary to refer to the evaluator program from inside the data file.

7. An example for the content of the file that resolves the alias for the DRESP3 evaluator path name is:

   EXTRESP,-,/network_drive/user_dir/dr3srv/dr3serv

   where, for this example, EXTRESP is the evaluator alias used in the CONNECT statement (see also under Examples).

EXAMPLES:

1. CONNECT BEAMEVAL HOIST, NEWBEAMS

   In this case, the user is requesting that all calculations on PBARL and PBEAML Bulk Data entries that are grouped as HOIST use the NEWBEAMS evaluator. The user must supply the NEWBEAMS beam cross-section evaluator library, and configure it to function with the NX Nastran executable program.

2. CONNECT DRESP3 TESTGRP EXTRESP

   In this case, the user is requesting that all DRESP3 calculations that are grouped as TESTGRP use the evaluator pointed at by the evaluator alias EXTRESP. EXTRESP is defined as in Remark 7 above, and the file containing this definition is pointed at as defined in Remark 6 above. The external program, with the default name dr3serv, contains NX Nastran supplied routines which the user may modify, and any user added routines called by the first set of routines.

DBCLEAN

Delete Database Versions and/or Projects

Deletes one or more versions and/or projects from the database.

FORMAT:

DBCLEAN VERSION = {version-ID,*} [PROJECT={'project-ID',*}]

DESCRIBERS:

Describer	Meaning
version-ID	Version identifier of the database to be deleted.
*	Wildcard. All versions or projects to be deleted.
project-ID	Project identifier of the project to be deleted. (See the FMS statement, "PROJ" .)

REMARKS:

1. There may be up to ten DBCLEAN statements in the FMS Section.
2. If no project-ID is given, the current project-ID is assumed.

EXAMPLES:

```
DBCLEAN VERS=7 PROJ = 'OUTER WING - LEFT'
```

The above example would delete from the database all data blocks and parameters stored under Version 7 of the project identified as OUTER WING – LEFT.

DBDICT

Prints Database Directory Tables:

Prints database directory tables.

DATABASE DIRECTORY TABLES

- Data blocks described by an NDDL DATABLK statement.
- Parameters described by an NDDL PARAM statement.
- All unique paths (KEYs) and their qualifiers values.
- Qualifiers and their current values.
- Data blocks not described by an NDDL DATABLK statement.
- Parameters not described by an NDDL PARAM statement.
- Project and version information.

BASIC FORMAT:

The basic format of DBDICT specifies which tables to print and prints all items (data blocks and parameters) found in the directory. Also, the attributes (colnames) to be printed and the print format are predefined. Note that more than one table may be specified on the same DBDICT statement.

DBDICT [DATABLK PARAM PROJVERS QUALCURR QUALIFIERS]

EXAMPLES:

```
DBDICT
DBDICT PARAM PROJVERS
```

FULL FORMAT:

The full format permits the selection of items by name and/or by the WHERE describer. The full format also permits the attributes to be printed using the SELECT describer. In addition, the print format can be specified with the SORT, FORMAT, and LABEL describers. Note that the full format only allows the specification of a single table on a DBDICT statement.

$$\text{DBDICT}\left(\left[\begin{array}{l}\left(\left[\begin{array}{c}\text{DATABLK}\\ \text{DATABLK(LOCAL)}\end{array}\right]=\left[\begin{array}{c}*\\ \text{(datablk-list)}\end{array}\right]\right)\\ \left(\left[\begin{array}{c}\text{PARAM}\\ \text{PARAM(LOCAL)}\end{array}\right]=\left[\begin{array}{c}*\\ \text{(param-list)}\end{array}\right]\right)\\ \text{PROJVERS}\\ \text{QUALCURR}\\ \text{QUALIFIERS}\end{array}\right]\text{WHERE(where-expr),}\right)$$

SELECT(colname[- ‘ col-label’]. . .),

FORMAT (FWIDTH = w [.d] DWIDTH = w [.d] AWIDTH = a IWIDTH = i,
LWIDTH = k COLSPACE = c VALUE = w,
colname = col-width, . . .),

$$\text{SORT}\left(\text{colname} = \left[\begin{array}{c}\text{A}\\ \text{D}\end{array}\right], \ldots\right),$$

$$\text{LABEL}\left(\text{'page - title'}\left[\begin{array}{c}\text{RIGHT}\\ \text{CENTER}\\ \text{LEFT}\end{array}\right]\right)$$

DESCRIBERS:

Describer	Meaning
DATABLK	Print the data blocks. datablk-list specifies a list of NDDL-defined data blocks separated by commas. If LOCAL is specified, the non-NDDL-defined data blocks are printed.
PARAM	Print the parameter table. param-list specifies a list of parameters separated by commas. If LOCAL is specified, the non-NDDL-defined parameters are printed.
PROJVERS	Print the project-version table.
QUALIFIERS	Print the qualifier table.
QUALCURR	Print the current values of the qualifiers. SORT is ignored.
where-expr	Logical expression that specifies the desired values of colnames described below. For example, WHERE(VERSION=4 AND SEID < > 2 AND SEID > 0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for a further description. The default for VERSION is the last version, and PROJECT is the current project. The default for qual is *, which is all qualifier values found on the database. See also Remark 12.
SELECT	Specifies a list of column names to be printed. The order of the specified colnames will be printed from left to right. If a colname is not specified, then all columns will be printed.
colname	Column name. colname specifies a particular attribute of the database item; such as, data block name (NAME), creation date (CDATE), number of blocks (SIZE), or qualifier name (SEID, SPC, etc.). The allowable colnames are given in the Remarks.
col-label	The label to be printed above the column identified by colname. The default for col-label is the colname. col-label may not be specified for colnames QUALSET, QUALALL, and TRAILER.
FWIDTH=w.d	Specifies the default width for single-precision real numbers in real and complex qualifiers. (Integers: w > 0 and d > 0, Default = 12.5)

Describer	Meaning
DWIDTH=w.d	Specifies the default width for double-precision real numbers in real and complex qualifiers. (Integers: w > 0 and d > 0, Default = 17.10)
AWIDTH=a	Specifies the default width for character string qualifiers. Character strings are printed with enclosing single quotation marks, even if the string is blank. (Integer > 0, Default = 8)
IWIDTH=i	Specifies the default width for integer qualifiers. (Integer > 0.) See Remarks for defaults.
LWIDTH=k	Specifies the default width for logical qualifiers. Logical values are printed as either “T” for TRUE or “F” for FALSE. (Integer > 0, Default = 1)
COLSPACE=c	Specifies the default number of spaces between columns. (Integer > 0) See Remarks for defaults.
VALUE=w	Specifies the default width for parameter values. The values are printed as character strings with left justification. (Integer > 0, Default = 40)
col-width	The print width of the data under colname or qual-name. For real numbers, specify w.d where w is the width of the field and d is the number of digits in the mantissa. For integers and character strings, specify w where w is the width of the field. col-width may not be specified for colnames QUALSET, QUALALL, and TRAILER.
SORT	Specifies how the rows are sorted. The sort is performed in order according to each colname specified in the list. A “D” following the colname causes the sort to be in descending order. An “A” following the colname causes the sort to be in ascending order. Colnames QUALSET, QUALALL, and TRAILER may not be specified under SORT. Each colname specified in SORT must be separated by commas.
page-title	A title to be printed on each page of the directory output.
RIGHT, CENTER, LEFT	Print justification of the page title.

REMARKS:

1. DBDICT prints seven different tables according to a default or a user-defined format. The tables are:

Table 2-1. DBDICT Tables

Describer	Description	Default Page-Title	See Remark
DATABLK	Data blocks described by a NDDL DATABLK statement.	NDDL DATABLOCKS	2
PARAM	Parameters described by a NDDL PARAM statement.	NDDL PARAMETERS	3
QUALCURR	Current Qualifiers and their values.	CURRENT QUALIFIERS	4
QUALIFIERS	Qualifiers and their values for each key number.	QUALIFIERS	5
DATABLK(LOCAL)	Data blocks not described by a NDDL DATABLK statement.	LOCAL DATABLOCKS	6
PARAM(LOCAL)	Parameters not described by a NDDL PARAM statement.	LOCAL PARAMETERS	7
PROJVERS	Project-Version.	PROJECT-VERSION	8

If DBDICT is specified without any describers then the NDDL Data blocks Table will be printed. See Remark 2.

DATABLK(LOCAL) and PARAM(LOCAL) produce no output, and QUALCURR produces the default values specified on the NDDL QUAL statement.

The defaults and allowable colnames for SELECT, FORMAT, SORT, and LABEL depend on the table. The defaults are described in the following remarks and tables.

2. The default print of the NDDL Data Blocks Table is obtained by:

```
DBDICT
```

or

```
DBDICT DATABLK
```

and is equivalent to:

```
DBDICT DATABLK ,
     SELECT(NAME,DATABASE,DBSET,PROJ,VERS,CDATE,CTIME,
          SIZE,KEY,PURGED='PU',EQUIVD='EQ',
          POINTER='FILE',QUALSET) ,
     FORMAT(NAME=8,DBSET=8,CDATE=6,CTIME=6,SIZE=5,
          KEY=4 ,PURGED=4,EQUIVD=4,POINTER=8,
          IWIDTH=5,COLSPACE=1) ,
     SORT(PROJ=A,VERS=A,DBSET=A,NAME=A) ,
     LABEL('NDDL DATABLOCKS' CENTER)
```

and looks like:

```
**** D I C T I O N A R Y   P R I N T ****

 EXECUTION OF DMAP STATEMENT NUMBER    20
  MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     16

                              NDDL DATABLOCKS
NAME   DATABASE DBSET  PROJ VERS  CDATE  CTIME  SIZE  KEY  PU  EQ   FILE  SEID  PEID  LOAD   SPC   MPC  METH
-----------------------------------------------------------------------------------------------------------------
AGG    MASTER  DBALL    1     1 930805  72340   0    326   1   0  132484   0    0
AXIC   MASTER  DBALL    1     1 930805  72336   0    315   1   0   65764
BGPDTS  MASTER  DBALL    1     1 930805  72338   1    324   0   2  131332       0
BGPDTX  MASTER  DBALL    1     1 930805  72338   1    324   0   1  131332       0
BJJ    MASTER  DBALL    1     1 930805  72341   0    332   1   0  132612       0
BULK   MASTER  DBALL    1     1 930805  72336   2    315   0   0   65700
CASECC  MASTER  DBALL    1     1 930805  72336   1    316   0   2   67428
```

Figure 2-1. DBDICT DATABLK Example

Table 2-2 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-2. DBDICT DATABLK Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	4	PROJ NO	Project number associated with PROJECT
VERS	4	VERSION	Version number
CDATE	6	CDATE	Creation date
CTIME	6	CTIME	Creation time
NAME	8	NAME	Parameter name
DATABASE	8	DATABASE	MASTER DBset name
DBSET	8	DBSET	DBset name
RDATE	6	RDATE	Revision date
RTIME	6	RTIME	Revision time
SIZE	5	SIZE	Number of blocks
qual-name	See Note.	qualifier name	Qualifier name
KEY	4	KEY	Key number
TRLi	8	TRLi	i-th word in the trailer
TRAILER	8	TRLi	All 10 trailer words
EXTNAME	8	EXTNAME	Extended name
EQUIVD	4	EQ	Equivalenced flag
PURGED	4	PU	Purged flag
EQFLAG	4	EF	Scratch equivalenced flag

Table 2-2. DBDICT DATABLK Colnames

colname	Default col-width	Default col-label	Description
SCRFLAG	4	SF	Scratch DBSET flag
POINTER	8	POINTER	Directory pointer
DBENTRY	8	DBENTRY	Database entry pointer
FEQCHAIN	8	FEQCHAIN	Forward equivalence chain
BEQCHAIN	8	BEQCHAIN	Backward equivalence chain
DBDIR20	9	DBDIR(20)	Directory word 20
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note

Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, AWIDTH=8, and FWIDTH=12.5.

3. The default print of the NDDL Parameter Table is obtained by:

```
DBDICT PARAM
```

and is equivalent to:

```
DBDICT PARAM,
     SELECT(NAME,DATABASE,DBSET,PROJ,VERS,CDATE,CTIME,
          KEY,VALUE,QUALSET),
     FORMAT(NAME=8,DATABASE=8,DBSET=8,CDATE=6,CTIME=6,
          KEY=4,VALUE=40,IWIDTH=5,COLSPACE=1),
     SORT(PROJ=A,VERS=A,DBSET=A,NAME=A),
     LABEL('NDDL PARAMETERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

EXECUTION OF DMAP STATEMENT NUMBER     21
 MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     17

                                                NDDL PARAMETERS
NAME    DATABASE   DBSET  PROJ VERS  CDATE  CTIME  KEY VALUE              SEID  PEID  LOAD   SPC   MPC  METH

-----------------------------------------------------------------------------------------------------------
ACOUSTIC MASTER   MASTER   1    1   930805  72338  323 0                   0     0
ALTRED   MASTER   MASTER   1    1   930805  72338  319 NO
BCHNG    MASTER   MASTER   1    1   930805  72337  325 FALSE                     0
DBALLX   MASTER   MASTER   1    1   930805  72336  318 DBALL              -1    -1
EPSBIG   MASTER   MASTER   1    1   930805  72339  323 1.000000E+12        0     0
ERROR    MASTER   MASTER   1    1   930805  72338  319 -1
FIXEDB   MASTER   MASTER   1    1   930805  72338  323 0                   0     0
```

Figure 2-2. DBDICT PARAM Example

Table 2-3 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-3. DBDICT PARAM Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	5	PROJ	Project number associated with PROJECT
VERS	4	VERS	Version number
CDATE	6	CDATE	Creation date
CTIME	6	CTIME	Creation time
NAME	8	NAME	Parameter name
DATABASE	8	DATABASE	MASTER DBset name
DBSET	8	DBSET	DBset name
RDATE	6	RDATE	Revision date
RTIME	6	RTIME	Revision time
POINTER	8	POINTER	Directory pointer
VALUE	40	VALUE	Parameter value
KEY	4	KEY	Key number
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

> **Note**
>
> Default widths for qualifiers are DWIDTH=17.10, AWIDTH=8, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5.

4. The default print of the Qualifier Table is obtained by:

```
DBDICT QUALIFIERS
```

and is equivalent to:

```
DBDICT QUALIFIERS ,
     SELECT(KEY QUALALL) ,
     FORMAT(DWIDTH=17.10 AWIDTH=8 IWIDTH=5 LWIDTH=1 ,
          FWIDTH=12.5  COLSPACE=2) SORT(KEY=A) ,
     LABEL('QUALIFIERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *
EXECUTION OF DMAP STATEMENT NUMBER     22

MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     18
                                          QUALIFIERS
KEY APRCH  B2GG   B2PP   BMETH  CMETH  CONFIG  DEFORM  DELTA  DESITER  DLOAD  DRMM  DYRD  EXTRCV  FMETH  FREQ
   FSCOUP GUST HIGHQUAL HINDEX IC IKBAR  IMACHNO  IPANEL  IQ    ISA  ISOLAPP  K2GG        K2PP         LOAD
   M2GG    M2PP  MACHINE METH  METHF  MFLUID MODEL   MPC  MTEMP  NCASE   NL99  NLOAD  NLOOP  NOQUAL  OPERALEV
   OPERASYS P2G   PEID  PVALID  SDAMP  SEDWN   SEID  SOLAPP SOLID SPC STATSUB   SUBDMAP      SUBMODEL SUPORT
   TEMPLD TFL  TSTEP  ZNAME       ZUZR1  ZUZR2  ZUZR3
-------------------------------------------------------------------------------------------------------------
335  '     '                                                    F   0
                    0                                                   '        '
     '     '                                                     0
           '        '   0       0                   0

-------------------------------------------------------------------------------------------------------------
336  '     '                                                      0
                    0                                                   '        '
                                                        0      0                    -1
                      0       0                   0
```

Figure 2-3. DBDICT QUALIFIERS Example

QUALALL selects all qualifiers to be printed. The qualifiers will be printed in alphabetic order. QUALSET selects the qualifiers SEID, PEID, SPC, MPC, LOAD, and METH to be printed.

Table 2-4 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers. QUALALL and QUALSET may not be specified in the FORMAT or SORT describers. The qualifier names and values are not printed one per row, but rather from left to right as one logical line that is allowed to wrap after 132 columns.

Table 2-4. DBDICT QUALIFIERS Colnames

colname	Default col-width	Default col-label	Description
KEY	5	KEY	Key number

Table 2-4. DBDICT QUALIFIERS Colnames

colname	Default col-width	Default col-label	Description
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note

Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

5. The default print of the Current Qualifier Table is obtained by:

```
DBDICT QUALCURR
```

and is equivalent to:

```
DBDICT QUALCURR SELECT(QUALALL),
     FORMAT(AWIDTH=8,IWIDTH=5,LWIDTH=1,COLSPACE=2),
     LABEL=('CURRENT QUALIFIERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *
EXECUTION OF DMAP STATEMENT NUMBER     24

MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     20
                                           CURRENT QUALIFIERS
APRCH  B2GG     B2PP     BMETH CMETH CONFIG DEFORM DELTA DESITER DLOAD DRMM DYRD EXTRCV  FMETH   FREQ  FSCOUP
       GUST  HIGHQUAL HINDEX  IC IKBAR IMACHNO IPANEL IQ  ISA ISOLAPP K2GG   K2PP        LOAD  M2GG
       M2PP  MACHINE  METH  METHF MFLUID MODEL  MPC MTEMP  NCASE NL99 NLOAD NLOOP NOQUAL  OPERALEV  OPERASYS
       P2G      PEID  PVALID SDAMP SEDWN SEID SOLAPP SOLID    SPC  STATSUB  SUBDMAP         SUBMODEL  SUPORT
       TEMPLD TFL TSTEP  ZNAME     ZUZR1 ZUZR2 ZUZR3
-------------------------------------------------------------------------------------------------------------
'    '  '     '  '       '   0     0     0     0    F        0    0     F     0       0      0      0       F
          0     0      0    0     0     0     0    0        0      1  '       '  '      '    300  '        '
        '     '      0    0    0     0     0   100    0       0    0     0     -1      0        0         0
        '     '     0    0    0     0     0  '      '   0     400      0  '           '        0       0
          0     0      0  '  '     0     0    0
-------------------------------------------------------------------------------------------------------------
```

Figure 2-4. DBDICT QUALCURR Example

Table 2-5 gives the allowable colnames along with a description that may be specified in the SELECT describers.

Table 2-5. DBDICT QUALCURR Colnames

colname	Default col-width	Default col-label	Description
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note

Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

6. The default print of the Local Data Block Table is obtained by:

```
DBDICT DATABLK(LOCAL)
```

and is equivalent to:

```
DBDICT DATABLK(LOCAL),
     SELECT(NAME,SUBDMAP,SIZE='BLOCKS',PURGED='PU',
          EQUIVD='EQ',POINTER,TRL1,TRL2,TRL3,TRL4,
          TRL5,TRL6,TRL7),
     FORMAT(NAME=8,SUBDMAP=8,IWIDTH=8,COLSPACE=2),
     SORT(NAME=A) LABEL('LOCAL DATABLOCKS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *
 EXECUTION OF DMAP STATEMENT NUMBER     23
  MODULE NAME = DBDICT    , SUBDMAP SEKRRS   , OSCAR RECORD NUMBER     19
                                  LOCAL DATABLOCKS
NAME    SUBDMAP     BLOCKS   PU   EQ   POINTER   TRL1   TRL2   TRL3   TRL4   TRL5   TRL6   TRL7
-----------------------------------------------------------------------------------------------
--
CASEW   PHASE1DR      1      0    0    131780     201    4       0     308     0      0       0
```

Figure 2-5. DBDICT DATABLK(LOCAL) Example

TRLi specifies the data block trailer word i where 1 ≤ i ≤ 10. TRAILER selects all 10 data block trailer words.

Table 2-6 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-6. DBDICT DATABLK(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name
SUBDMAP	8	SUBDMAP	SubDMAP name
SIZE	8	BLOCKS	Number of blocks
EQUIVD	8	EQ	Equivalenced flag
PURGED	8	PU	Scratch flag
POINTER	8	POINTER	Directory pointer
TRLi	8	TRLi	i-th word in the trailer
TRAILER	8	TRLi	All 10 trailer words
EXTNAME	8	EXTNAME	Extended name

7. The default print of the Local Parameter Table is obtained by:

```
DBDICT PARAM(LOCAL)
```

and is equivalent to:

```
DBDICT PARAM(LOCAL) SELECT(NAME,SUBDMAP,VALUE),
     FORMAT(COLSPACE=4,VALUE=40,AWIDTH=8),
     SORT(NAME=A) LABEL(' LOCAL PARAMETERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

 EXECUTION OF DMAP STATEMENT NUMBER     24
  MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     20
                                                        LOCAL PARAMETERS
NAME        SUBDMAP     VALUE
-----------------------------------------------------------------
AERO        SESTATIC    FALSE
AERO        PHASE1DR    FALSE
ALTRED      SESTATIC    NO
ALTRED      PHASE1DR    NO
ALTSHAPE    SESTATIC    0
ALWAYS      PHASE1DR    -1
ALWAYS      PHASE1C     -1
ALWAYS      SEKRRS      -1
ALWAYS      SESTATIC    -1
APP         PHASE1DR    STATICS
APP         PHASE1C     STATICS
APP         SESTATIC    STATICS
APRCH       SESTATIC
ASING       PHASE1DR    0
ASING       SEKRRS      0
ASING       PHASE1C     0
ASING       SESTATIC    0
```

Figure 2-6. DBDICT PARAM(LOCAL) Example

Table 2-7 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-7. DBDICT PARAM(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name
SUBDMAP	8	SUBDMAP	SubDMAP name
VALUE	40	VALUE	Parameter name

8. The default print of Project-Version Table is obtained by:

```
DBDICT PROJVERS
```

and is equivalent to:

```
DBDICT PROJVERS ,
     SELECT(PROJECT='PROJECT NAME',PROJ='PROJ NO.' ,
          VERS='VERSION',DELFLG='DELETED' ,
          CDATE='CREATION DATE' CTIME='CREATION
          TIME') ,
     FORMAT(PROJECT=40,PROJ=10,VERS=10,DELFLG=7,
          COLSPACE=1 ,CDATE=13,CTIME=13) ,
     LABEL('PROJECT-VERSION',CENTER) ,
          SORT(PROJ=A,VERS=A)
```

Table 2-8 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-8. DBDICT PROJVERS Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	10	PROJ NO	Project number associated with PROJECT
VERS	10	VERSION	Version number
DELFLG	7	DELETED	Flag indicating whether this project/version has been deleted by the RESTART NOKEEP or DBCLEAN statements.
CDATE	13	CREATION DATE	Creation date
CTIME	13	CREATION TIME	Creation time

CDATE is printed as YYMMDD where YY, MM, and DD are the year, month, and date, respectively. CTIME is HHMMSS where HH, MM, and SS are the hour, minute, and second, respectively.

9. If a parameter or qualifier value is defined to be character string, then the value will be printed with enclosing single quotation marks. Blank strings will also be printed with single quotation marks.

10. If a given qualifier is not in the path of a given data block or parameter, then blank spaces will be printed.

11. A line will wrap if additional columns need to be printed and not enough space is available on the output. The first column of each additional line is to be indented by the width of the first column printed for the entry.

12. The where-expr has the following rules:

    - If the where-expr specifies a colname that is not assigned to the data block or parameter, then no directory information will be printed for that data block or parameter. For example, given that SPC is not a qualifier for KGG, then the following DBDICT statement will produce no output:

      ```
      DBDICT DATABLK=KGG WHERE(SPC=10)
      ```

    - If the where-expr does not specify a colname that is assigned to the data block (or parameter), then the qualifier is wildcarded. For example, given that SEID is a qualifier for KAA, then the following DBDICT statements are equivalent:

      ```
      DBDICT DATABLK=KAA
      DBDICT DATABLK=KAA WHERE(SEID = *)
      ```

13. A colname specified in the where-expr must be specified in the SELECT clause if the SELECT clause is also specified.

EXAMPLES:

1. Print the Project Version Table with a title.

   ```
   DBDICT  PROJVERS SORT(PROJ,VERSION) LABEL('PROJECT
        VERSION TABLE' LEFT)
   ```

2. Print a directory of all data blocks qualified with PEID=10 or SEID=10. Print columns for the NAME and DBSET, and the qualifiers SPC, MPC, and LOAD.

   ```
   DBDICT  DATABLK SELECT(NAME,SPC,MPC,LOAD,DBSET,SIZE,
             SEID,PEID) ,
        SORT(NAME,SIZE=D)  WHERE(SEID=10 OR PEID=10)
   ```

DBDIR

Prints Database Directory Tables

Prints the database directory.

FORMAT:

DBDIR [VERSION={version-ID,*}] [PROJECT={'project-ID',*}] [FORMAT={format-no}]

DESCRIBERS:

Describer	Meaning
version-ID	Version identifier.
project-ID	Project identifier. (See PROJ.)
*	Wildcard. All versions or projects will be printed.
format-no	Specifies the type(s) of directory information to be printed. If several items are desired, then the sum of the following values is required. 63 is the default, which is the sum of the first six values below: 1 Project Version Table 2 NDDL Data Blocks 4 NDDL Parameters 8 NDDL Empty Data Blocks 16 NDDL Data Block Trailers 32 NDDL Path Value Table for NDDL entries 64 Scratch Data Blocks

REMARKS:

1. Trailers are not printed unless NDDL data blocks, empty data blocks, or local scratch data blocks are also printed.

2. The path value table is not printed unless NDDL data blocks, empty data blocks, or parameters are also printed.

3. If no version-ID is given, all versions will be printed. If no project-ID is given, the current project will be printed.

4. The directory print includes all items found on the primary database and all databases attached by the DBLOCATE and DBLOAD FMS statements.

5. Only one DBDIR statement is permitted in the FMS Section, and this statement should appear last in the FMS Section.

EXAMPLES:

```
DBDIR
```

The above example would print all versions of the current project-ID in the database.

DBFIX

Database Directory Error Detection

Detects and optionally corrects errors in the database directory.

FORMAT:

$$\text{DBFIX}\left[\left\{\begin{array}{c}\text{LIST}\\ \text{NOLIST}\end{array}\right\}\left\{\begin{array}{c}\text{CORRECT}\\ \text{NOCORRECT}\end{array}\right\}\right]$$

DESCRIBERS:

Describer	Meaning
LIST	Requests a debug listing of the database directory pointers.
NOLIST	Suppresses a debug listing of the database directory.
CORRECT	Corrects the database if any errors are found.
NOCORRECT	Suppresses the correction of the database.

REMARKS:

1. It is recommended that a backup copy of the database be made before this statement is used since corrections of the database are achieved through the deletion of data. Data blocks and parameters are deleted from the database if they have (1) incorrect paths (different from those listed in the NDDL), (2) incorrect names (two or more names that are not equivalenced and reference the same data), or (3) incorrect directory pointers.

2. NOLIST does not suppress the listing of any corrections made to the database.

EXAMPLES:

```
DBFIX LIST,NOCORRECT
```

The example above requests a printout of the directory pointers and any errors, but not the corrections.

DBLOAD

Load a Database from a FORTRAN File

Recovers data blocks or parameters from a database created by the DBUNLOAD statement.

FORMAT:

```
DBLOAD [DATABLK= [ * | (datablk-list) ] PARAM= [ * | (param-list) ] WHERE(where-expr) ,
CONVERT(convert-expr)UNIT = unit FORMAT { BINARY | NEUTRAL } { OVRWRT | NOOVRWRT } ]
```

EXAMPLES:

1. Load the database stored in ASCII format on FORTRAN unit 12.

```
DBLOAD      UNIT=12 FORMAT=NEUTRAL
ASSIGN      DBLOAD='physical file name of unloaded database'UNIT=12
            FORMATTED
```

2. Load version 1 of KAA under project FRONT BUMPER and store it on the primary database under version 5 and project BUMPER. Overwrite duplicates found on the primary database.

```
DBLOAD      DATABLK=(KAA) WHERE(PROJECT='FRONT
            BUMPER' AND ,SEID=10 AND VERSION=1) CONVERT(VERSION=5;
            , PROJECT='BUMPER') OVRWRT
ASSIGN      DBLOAD='physical file name of unloaded
            database'
```

DESCRIBERS:

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is * which selects all data blocks. The loaded data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The loaded parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.
where-expr	A logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section on WHERE and CONVERT clauses. The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is * for all DBsets. The default for qual is *, which is all qualifier values found on the loaded database. See also Remark 8 .

Describer	Meaning
convert-expr	Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is: PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expri[;..] For example, CONVERT (SEID=100 + SEID; SPC=102). See the beginning of this section on WHERE and CONVERT clauses. The default action for VERSION and PROJECT is to use the same version IDs and project IDs; i.e., CONVERT(PROJECT=PROJECT; VERSION=VERSION). But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. The default action for qualifiers and DBSET is to use the same values as long as they are defined in both databases. If not, see Remark 8 .
unit	Specifies the FORTRAN unit number of the database to be loaded. The unit must be specified on an ASSIGN statement that references the physical filename of the loaded database. The default is 51.
OVRWRT NOOVRWRT	By default, if duplicate data blocks or parameters exist on the loaded and primary databases, then a fatal message is issued. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as a data block or parameter on the primary database.
NEUTRAL, BINARY	The database to be loaded may be in BINARY or NEUTRAL format. BINARY indicates the database to be loaded is in binary or FORTRAN unformatted format. NEUTRAL indicates the database to be loaded is in ASCII format. The default is BINARY.

REMARKS:

1. The DBLOAD statement and its applications are discussed further in “Database Archival and Retrieval” in the *NX Nastran User’s Guide*.

2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be loaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be loaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be loaded.

3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.

4. The database to be loaded is attached as read-only. In other words, items can only be fetched and not stored on this database.

5. If more than one DBLOAD statement is specified, then the statements will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOAD statements, then the last duplicate will be used.

6. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.

7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.

8. If the database to be loaded and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:

   - If a qualifier in the NDDL of the database to be loaded is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double-precision, complex or character, then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not loaded.

   - If a DBset-name in the NDDL of the database to be loaded is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the database to be loaded.

9. Data blocks that are equivalenced on the database to be loaded remain equivalenced as long as they are loaded in the same DBLOAD statement or in consecutive DBLOAD statements with the same unit number. Otherwise, a separate copy is produced for the secondary data block.

10. It is not possible to restart from a database created by DBLOAD in the same run.

11. SOL 190 (or DBTRANS) is also required with DBLOAD if:
    - The database to be loaded has a different BUFFSIZE.
    - The database to be loaded is in neutral format or is being transferred between different machine types.

      See also Migrating Databases in the *NX Nastran User's Guide*.

DBLOCATE

Attaches Secondary Databases

Obtains data blocks or parameters from prior versions of the primary database, or other databases. DBLOCATE may also be used to compress the primary database and to migrate databases created in prior NX Nastran versions.

FORMAT:

$$\text{DBLOCATE}\left[\text{DATABLK}=\left[\begin{matrix}*\\(\text{datablk-list})\end{matrix}\right]\text{PARAM}=\left[\begin{matrix}*\\(\text{param-list})\end{matrix}\right]\text{WHERE}(\text{where-expr})\ ,\right.$$
$$\left.\text{CONVERT}(\text{convert-expr})\text{LOGICAL}\ =\ \text{dbname}\left[\begin{matrix}\text{OVRWRT}\\\text{NOOVRWRT}\end{matrix}\right]\text{COPY}\right]$$

EXAMPLE:

1. Locate in version 4 of MASTER3 all data blocks named KAA for all superelements with IDs greater than 0.

```
DBLOCATE    DATABLK=(KAA) WHERE(PROJECT='FRONT
            BUMPER'  ,AND SEID>0 AND VERSION=4) LOGI=MASTER3
ASSIGN      MASTER3='physical file name of master
            DBset'
```

2. Copy all data blocks and parameters from the last version of MASTER3 to the primary database. For all items with the qualifier SEID, change the SEID to twice the old ID number.

```
DBLOCATE    CONVERT(SEID=2*SEID) COPY LOGI=MASTER3

ASSIGN      MASTER3='physical file name of master
            DBset'
```

3. Compress a database with multiple versions. All versions under the current project-ID (see PROJ statement) will be copied from the database OLDDB to NEWDB.

```
ASSIGN       MASTER3='physical filename of new
             master DBset'
ASSIGN       OLDDB='physical filename of old master
             DBset'
DBLOCATE     LOGI=OLDDB COPY WHERE(VERSION=*) ,
             CONVERT(VERSION=VERSION;PROJECT=PROJECT)
```

DESCRIBERS:

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is *, which selects all data blocks. The located data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK = (KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The located parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.
where-expr	A logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID < > 2 AND SEID > 0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section on WHERE and CONVERT clauses. The default for VERSION is the last version-ID and PROJECT is the current project-ID. The default for qual is *, which is all qualifier values found on the located database. See also Remark 9 .
convert-expr	Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is: PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expri[;...] For example, CONVERT (SEID=100+SEID; SPC=102). See the beginning of this section on WHERE and CONVERT clauses. The default action for VERSION and PROJECT is to convert to the current version-ID and current project-ID. But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. See Example 26-3. The default action

Describer	Meaning
	for qualifiers and DBSET is to use the same values as long as both databases have the same NDDL scheme. If not, see Remark 9 .
dbname	Specifies the logical name of the master directory DBset of the located database. dbname must be specified on an ASSIGN statement, which references the physical file name. By default, the located database is also the primary database. (If dbname is specified for the primary database, then dbname must be MASTER.)
OVRWRT NOOVRWRT	By default, duplicate data blocks or parameters on the located database will take precedence over those on the primary database. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as the data block or parameter on the primary database. If NOOVRWRT is specified, then a fatal message is issued.
COPY	Requests that the located data blocks or parameters be copied to the primary database.

REMARKS:

1. The DBLOCATE statement and its applications are discussed further in “Database Concepts” in the *NX Nastran User’s Guide*.

2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be located. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be located. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be located.

3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.

4. If more than one DBLOCATE statement is specified, then the statements will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOCATE statements, then the last duplicate will be used.

5. If the located database is not the primary database, then it is attached for read-only purposes. In other words, items can only be fetched and not stored on the located database.

6. If the RESTART FMS statement is also specified, then located data blocks and parameters are treated as if they exist in the restart version. In other words, restart equivalences will be made on located items at the beginning of the run and can be subsequently broken as a result of regeneration and/or NDDL dependencies.

7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.

8. If LOGICAL refers to the primary database and one version is to be copied to another, then the items are equivalenced.

9. If the located database and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:

   - If a qualifier in the NDDL of the located database is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double precision, complex or character, then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not located.

   - If a dbset-name in the NDDL of the located database is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the located database.

DBSETDEL

Deletes DBsets

Deletes DBsets, all of its members, and associated physical files.

FORMAT:

DBSETDEL dbsetnamei

DESCRIBERS:

Describer	Meaning
dbsetnamei	Specifies the name(s) of DBset(s) to be deleted. The DBset names MASTER, OBJSCR, or SCRATCH may not be specified.

REMARKS:

1. The DBSETDEL statement and its applications are discussed further in “Deleting DBsets” in the *NX Nastran User’s Guide*.
2. If dbsetnamei does not exist, then no action is taken.
3. After a DBset has been deleted with this statement, it may be recreated with the INIT statement in a subsequent run.

EXAMPLES:

Delete DBset DBUP20 from the database.

```
DBSETDEL DBUP20
```

DBUNLOAD

Unload a Database to a FORTRAN File

Stores data blocks or parameters from the primary database onto a FORTRAN file in a binary or neutral format, for purposes of database compression or database transfer between different computers.

FORMAT:

$$\text{DBUNLOAD}\left[\text{DATABLK}=\left[\begin{matrix}*\\(\text{datablk-list})\end{matrix}\right]\text{PARAM}=\left[\begin{matrix}*\\(\text{param-list})\end{matrix}\right]\text{WHERE}(\text{where-expr})\right.$$
$$\left.\text{UNIT}=\text{unit FORMAT}=\left\{\begin{matrix}\text{BINARY}\\\text{NEUTRAL}\end{matrix}\right\}\left\{\begin{matrix}\text{REWIND}\\\text{NOREWIND}\end{matrix}\right\}\right]$$

EXAMPLES:

1. Unload the database in ASCII format onto FORTRAN unit 12.

```
DBUNLOAD    UNIT=12 FORMAT=NEUTRAL

ASSIGN      DBUNLOAD='physical file name of
            FORTRAN unit 12'

            UNIT=12 FORMATTED
```

2. Unload version 1 of KAA under project FRONT BUMPER.

```
DBUNLOAD     DATABLK=(KAA) WHERE(PROJECT='FRONT
             BUMPER' ,AND SEID=10 AND VERSION=1)
ASSIGN       DBUNLOAD='physical file name of
             FORTRAN unit 50'
```

DESCRIBERS:

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is * which selects all data blocks.

Describer	Meaning
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters.
where-expr	Logical expression which specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section on WHERE and CONVERT Clauses. The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is * for all DBsets. The default for qual is *, which is all qualifier values found on the primary database.
unit	Specifies the FORTRAN unit number to unload the database. The unit must be specified on an ASSIGN statement, which references its physical filename. The default is 50.
NEUTRAL BINARY	The database may be unloaded in BINARY or NEUTRAL format. BINARY indicates the database is to be unloaded in binary or FORTRAN unformatted. NEUTRAL indicates the database is to be unloaded in ASCII format. The default is BINARY.
NOREWIND REWIND	By default, if DBUNLOAD is executed more than once for the same unit, then the unit is not rewound. REWIND requests that the unit be rewound prior to unloading.

REMARKS:

1. The DBUNLOAD statement and its applications are discussed further in “Database Archival and Retrieval” in the *NX Nastran User’s Guide*.

2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be unloaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be unloaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be unloaded.

3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.

4. If more than one DBUNLOAD statement is specified, then the statements will be processed in the order in which they appear.

5. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.

6. If NEUTRAL is specified, then only data blocks with an NDDL description are unloaded. (See the *NX Nastran DMAP Programmer's Guide* under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be unloaded in BINARY if TYPE=UNSTRUCTURED, KDICT, or KELM.

7. Data blocks that are equivalenced on the primary database remain equivalenced as long as they are unloaded in the same DBUNLOAD statement or in consecutive DBUNLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.

DBUPDATE

Specify Database Directory Update Interval

Specifies the maximum length of CPU time between database directory updates to the MASTER DBset. This statement is intended to be used if the INIT MASTER(RAM=r) option is specified.

FORMAT:

DBUPDATE [=] update-time

DESCRIBERS:

Describer	Meaning
update-time	CPU time interval in minutes (real or integer) between database directory updates.

REMARKS:

1. The difference in CPU time from the last update is checked after the execution of each DMAP instruction. The database directory is updated if this difference is greater than update-time. Update-time and CPU time are accurate to the nearest whole second only.

2. If update-time < 0, then database directory updates are only performed at the end of the run.

3. Defaults for update-time are machine dependent and may be found in the *NX Nastran Installation and Operations Guide*.

4. Periodic updates of the directory tables to MASTER DBset increase the integrity of the database during system crashes (for example, crashes due to insufficient time or space).

5. In addition to those specified by DBUPDATE, directory updates are performed automatically at various points in the execution of the DMAP. An asterisk appears after the word “BEGN” in the Executive Summary Table whenever an update occurs. These updates occur whenever a permanent data block or parameter DMAP equivalence or restart equivalence is broken. Updates also occur upon

deletions. Additions to the database do not automatically cause a directory update to take place.

6. This statement is in effect only when INIT MASTER(RAM=r) is being used. INIT MASTER(S) and INIT MASTER(NORAM) disable periodic and automatic updates.

7. Update-time may also be changed with the DMAP instruction PUTSYS(update-time, 128) or the NASTRAN SYSTEM(128)=update-time statement. (The update-time must be a real, single-precision value specified in minutes.)

EXAMPLES:

```
DBUPDATE=5.5
```

The above example would call for a database directory update at the end of a DMAP module execution, after five and one-half minutes of CPU time have elapsed from the last update.

DEFINE

Parameter Definition

Assigns user defined keywords (or cellnames) to NASTRAN system cells. (See the NASTRAN statement for a description of “cellname”). In addition, the DEFINE statement provides a mechanism to set default values for system cells.

FORMAT:

DEFINE keyword [=expression] [LOCATION=SYSTEM(i)] [TYPE=type]

DESCRIBERS:

Describer	Meaning
keyword	User defined name, 1 through 24 characters in length. The first character must be alphabetic. Only the following characters can be used for keywords: A-Z, 0-9, ’ , _ , and “. All other characters are invalid.
expression	Expression produces a single value from a set of constant and/or variable parameters separated by operators. The value is assigned to the “keyword” and is also used to set the value for the NASTRAN system cell specified by “LOCATION”. The TYPE determines both the type of the result and the type conversions that will be applied to the constants and variables within the expression—mixed mode expressions are allowed (see Remark 6). The parentheses can be used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses. The variable parameters within the expression must be keywords previously defined on a DEFINE statement. The following operations are allowed

Describer **Meaning**

Parameter Type	Operator	Operation
Integer or Real	+	Addition
	−	Subtraction
	×	Multiplication
	/	Division
Logical	+	Bit-wise OR
Logical	-	Bit clear. For example, the result of *a-b* is equal to the value of *a* with the bits associated with *b* set to 0.

SYSTEM(I) Specifies the NASTRAN system cell number to be associated with the keyword.

type The type of the expression result and the type conversions that will be applied to the constants and variables within the expression. Allowable data types are as follows:

Description	Type
Integer (default)	I
Real	R
Logical	LOGICAL

REMARKS:

1. If TYPE, LOCATION and EXPRESSION are omitted, the default data type is integer and the default value is zero.

2. If expression is omitted, an internal default will be assigned to the keyword/cellname based on the LOCATION. (See “NASTRAN” for a list of internal default values).

3. A DEFINE statement that specifies a LOCATION sets the default for a NASTRAN system cell and therefore it is not necessary to set the system cell value on a subsequent NASTRAN statement, unless the user wishes to override the previous DEFINE statement setting. Also, since more than one DEFINE statement may

be present for the same “keyword” the last specification takes precedence. “Keywords” referenced on a NASTRAN statement or in an expression on the DEFINE statement are automatically substituted by the last specification of the “keyword” prior to the current statement being processed.

4. DEFINE statements may also be specified in Runtime Configuration (RC) files. See the *NX Nastran Installation and Operations Guide.*

5. System cells may also be set with the NASTRAN statement. In addition, they may be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See the *NX Nastran DMAP Programmer’s Guide*

6. Each operand within the expression will be converted to the result type prior to the arithmetic operation. For example: the statement “DEFINE JJ=2.5 + 3.6 TYPE=I” would result in 2.5 and 3.6 being converted to 2 and 3 respectively and the result of 5 would be assigned to JJ.

EXAMPLES:

1. Change the default value for block size.

```
DEFINE BUFFSIZE=4097  LOCATION=SYSTEM(1)
```

2. Set the sparse matrix selection to forward-backward substitution only.

```
DEFINE SPARSE=16 LOCATION=SYSTEM(126)
```

3. Define the system cell keyword and default value for the maximum output line count and then reset it to another value on a NASTRAN statement. Note: The DEFINE statement would typically be placed in an RC file and the NASTRAN statement would be placed in the File Management Section whenever the user wants to override the DEFINE statement default setting.

```
DEFINE        MAXLINES=999999999 LOCATION=SYSTEM(9)
NASTRAN       MAXLINES=100000
```

4. Define system cells that behave like “toggles”, turning some feature on or off.

```
DEFINE                MESH=2 LOCATION=(31)
DEFINE                NOMESH=0 LOCATION=(31)


NASTRAN               MESH
```

Note: Since each subsequent DEFINE statement redefines the default value, the second DEFINE of system cell location 31 sets the default value to 0. A NASTRAN statement can then be inserted in the input file to reset the MESH system cell back to a value of 2. This same technique can be used with any system cell where the user wishes to simply refer to the system cell keyword and have the system cell set to a previous DEFINE statement default.

5. Invalid usage of the DEFINE and NASTRAN statement:

```
DEFINE          BUFFSIZE=4097
NASTRAN         BUFFSIZE=2048
```

Valid usage:

```
DEFINE        BUFFSIZE=4097 LOCATION=SYSTEM(1)
NASTRAN       BUFFSIZE=2048
```

ENDJOB

Terminate Job

Terminates the job at a user-specified location in the FMS Section.

FORMAT:

ENDJOB

REMARKS:

ENDJOB is normally used after a DBDICT or DBDIR statement or after database initialization.

EXAMPLES:

```
DBDICT
ENDJOB
```

EXPAND

Concatenate New DBset Members

Concatenates additional DBset members on an existing permanent DBset previously defined with an INIT statement.

FORMAT:

EXPAND dbset-name LOGICAL=(log-namei [(max-sizei)]...)

DESCRIBERS:

Describer	Meaning
dbset-name	The name of a DBset previously defined with an INIT statement.
log-namei	Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement which refers to the physical file name of the DBset member.
max-sizei	Specifies the maximum size, in blocks, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the unit keywords below

Unit Keyword	Storage Unit
KW	Kilowords (thousand)
KB	Kilobytes (thousand)
MW	Megawords (million)
MB	Megabytes (million)
GW	Gigawords (billion)
GB	Gigabytes (billion)

For example, 100MB is 100 megabytes. The size of a block in words is defined by BUFFSIZE.

REMARKS:

1. On all computers with dynamic file allocation, the physical filename of a DBset member may be specified on an ASSIGN statement:

   ```
   ASSIGN log-name='physical filename'
   ```

   If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in “Introduction to Database Concepts” in the *NX Nastran User’s Guide*.

EXAMPLES:

```
ASSIGN        DBMEM02='physical file name'
EXPAND        DBALL LOGICAL=(DBMEM02)
```

This would create and add the DBset member DBMEM02 to the already existing DBset DBALL.

INCLUDE

Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

FORMAT:

INCLUDE 'filename'

DESCRIBERS:

Describer	Meaning
filename	External file to be inserted. The '*directory_path/filename*' must begin and end with the single quote character.

EXAMPLES:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD=100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

REMARKS:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.

2. Each line has a 72 character limit. Multiple lines can be used when file names are long. The entire string must begin and end with the single quote character.

   For example,

D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat

can be defined on multiple lines:

```
INCLUDE ‘D:\folder1\folder2\folder3\folder4\folder5\folder6\
folder7\folder8\folder9\mydata.dat'
```

The following input format is also supported:

```
INCLUDE ‘D:\folder1\folder2\folder3\
         folder4\folder5\
     folder6\folder7\
        folder8\folder9\mydata.dat'
```

INIT

Create a DBset

Creates a temporary or permanent DBset. For the SCRATCH and MASTER DBsets, all or some of their space may be allocated to real memory.

FORMAT 1:

Initialize any DBset except MASTER and SCRATCH:

INIT DBset-name [LOGICAL=(log-namei(max-sizei),...)

BUFFSIZE=b CLUSTER=c]

FORMAT 2:

Initialize the MASTER DBset:

$$\text{INIT MASTER}\left[\left(\begin{array}{c}\text{RAM} = \text{r}\\ \text{NORAM}\end{array}, \text{S}\right)\text{LOGICAL=(log-name(max-sizei),...)}\right.$$

BUFFSIZE = b CLUSTER = c

FORMAT 3:

Initialize the SCRATCH DBset:

$$\text{INIT SCRATCH}\left[\left(\begin{array}{c}\text{MEM} = \text{m}\\ \text{NOMEM}\end{array}\right)\text{LOGICAL = (log-name(max-sizei),...),}\right.$$

SCR300 = (log-namei(max-sizei),...) BUFFSIZE = b CLUSTER=c]

EXAMPLES:

1. Modify the default allocation of the DBALL DBset to 50000 blocks.

```
INIT DBALL LOGI=(DBALL(50000))
```

2. Do not allocate any real memory for the MASTER and SCRATCH DBsets.

```
INIT            MASTER(NORAM)
INIT            SCRATCH(NOMEM)
```


3. Create a new DBset called DBUP with two members DBUP1 and DBUP2.

```
INIT            DBUP LOGI=(DBUP1, DBUP2)
ASSIGN          DBUP1 ='physical filename 1'
ASSIGN          DBUP2='physical filename 2'
```

DESCRIBERS:

Describer	Meaning
dbset-name MASTER SCRATCH	The name of a temporary or permanent DBset.
log-namei	Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement, which refers to the physical file name of the DBset member. If no log-namei is specified, then the DBset will have one member and the log-name will be the same as the DBset-name. A maximum of twenty log-names may be specified. For the SCRATCH DBset see also Remark 8 . SCR300 is a special keyword that indicates that the log-names are members reserved for DMAP module internal scratch files.
max-sizei	Specifies the maximum size, in blocks, words, or bytes, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the unit keywords below.

Unit Keyword	Storage Unit
KW	Kilowords (1024)
KB	Kilobytes (1024)
MW	Megawords (1048576)
MB	Megabytes (1048576)

Describer	Meaning
	GB — Gigabytes (1073741824) GW — Gigawords (1073741824) For example, 100MB is 100 megabytes. The size of a block in words is defined by BUFFSIZE. The default for DBALL and SCRATCH is 250,000 blocks.
RAM NORAM	RAM=r requests that r words of real memory are to be allocated for the MASTER DBset. The default is RAM or RAM=30000. NORAM or RAM=0 specifies no real memory is to be allocated.
S	If the primary database is being created in the run, this option requests that all DBsets in the primary database will be automatically deleted at the end of the run. INIT MASTER(S) is equivalent to specifying scr = yes on the “nastran” command. See the “nastran Command and NASTRAN Statement” . If the run is a restart, then this option is ignored.
MEM NOMEM	MEM=m specifies that m blocks of real memory are to be allocated for the SCRATCH DBset. The default m is machine dependent and may be found in the *NX Nastran Installation and Operations Guide.* NOMEM or MEM = 0 requests that no real memory is to be allocated.
BUFFSIZE	BUFFSIZE = b specifies the number of words per block in the DBset and will override the value specified by the BUFFSIZE keyword on the NASTRAN statement. The default for b is obtained from the NASTRAN BUFFSIZE statement. See the *NX Nastran Installation and Operations Guide*.
CLUSTER	CLUSTER = c specifies the number of blocks per cluster in the DBset. The default is 1, and any other value is not recommended.

REMARKS:

1. The INIT statement and its applications are discussed further in “Introduction to Database Concepts” in the *NX Nastran User’s Guide*.

2. There are four DBsets that are predefined and automatically allocated by the program. Their DBset-names are MASTER, DBALL, SCRATCH, and OBJSCR, and they are described in “Introduction to Database Concepts” in the *NX Nastran User’s Guide*.

3. On all computers with dynamic file allocation, the physical filename of a DBset member may specified on an ASSIGN statement:

   ```
   ASSIGN log-name='physical filename'
   ```

   If an ASSIGN statement is not specified for the member, then a name is automatically assigned.

4. It is recommended that there be sufficient physical space to hold a DBset member should it reach its maximum size (max-sizei). The max-sizei may be converted to words by multiplying by b. A summary of space usage and allocation is printed at the end of the Execution Summary Table.

5. In restart runs, the INIT statement is ignored for preexisting permanent DBsets. The INIT statement is intended to be specified only in the run in which the DBset is being created. If more DBset members need to be added to the DBset, then the EXPAND statement is used.

6. If RAM or RAM = r is specified and the run terminates because the computer operating system detects insufficient space or time or the computer halts due to a power outage or operator interruption, then it may not be possible to restart from the database. See the DBUPDATE FMS statement.

7. BUFFSIZE = b and CLUSTER = c must satisfy the following inequality:

   $$b \leq \frac{64000}{c} + 5$$

8. By default, the SCRATCH DBset is divided into two partitions: LOGICAL and SCR300. The LOGICAL partition, log-names after the LOGICAL keyword, are reserved for DMAP scratch data blocks and the SCR300 partition for DMAP module internal scratch files.

   - The maximum total number of log-names for LOGICAL and SCR300 is 20. For example, if LOGICAL has 8 log-names, then SCR300 can have no more than 12 log-names.

- If NASTRAN SYSTEM(142) = 1 is specified, then the SCR300 partition is not created and internal scratch files, along with DMAP scratch data blocks, will reside on the LOGICAL partition. The default is SYSTEM(142) = 2.

- If NASTRAN SYSTEM(151)=1 is specified and the LOGICAL partition has reached its maximum size, then the SCR300 partition will be used. The default is SYSTEM(151) = 1.

- By default, the space specified for the SCR300 partition is released to the operating system after each DMAP module is executed as long as the module used more than 100 blocks for internal scratch files. If 100 blocks is not a desirable threshold, then it may be changed by specifying NASTRAN SYSTEM(150) = t, where t is the number of blocks for the threshold.

9. BUFFSIZE = b is predefined for DBset-names MSCOBJ, OBJSCR, and USROBJ and may not be changed by BUFFSIZE on this statement or the NASTRAN BUFFSIZE bmax statement. The default for b is recommended for all except very large problems. bmax must reflect the maximum of b specified for all DBsets attached to the run, including the delivery database. See *NX Nastran Installation and Operations Guide* for the defaults of b and bmax.

10. If INIT MASTER(RAM = r) and INIT SCRATCH(MEM = m) are specified, then BUFFSIZE for these DBsets must be the same. If not, a warning message is issued, and the BUFFSIZE for the SCRATCH DBset is reset to that of the MASTER DBset.

11. Only one INIT statement per dbset-name may be specified in the File Management Section.

PROJ

Define Database Project-Identifier

Defines the current or default project identifier, project-ID.

FORMAT:

PROJ [=] ’project-ID’

DESCRIBERS:

Describer	Meaning
project-ID	Project identifier. Must be enclosed in single quotes. (Character string, maximum of 40 characters; Default = blank)

REMARKS:

1. There may be only one PROJECT statement in the File Management Section. The PROJECT statement must be specified before all DBCLEAN, DBDIR, DBDICT, RESTART, DBLOCATE, and DBLOAD statements where project-ID is not specified by the user.

2. This statement is optional and specifies that all data blocks and parameters stored on or accessed from the database in the current run shall also be identified by project-ID. Therefore, in subsequent runs that may access this data through other FMS statements such as RESTART, the project-ID must be specified.

3. Project-ID is the default on DBCLEAN, DBDIR, DBDICT, and RESTART FMS statements and in the WHERE and CONVERT clause of the DBLOCATE statement.

4. Leading blanks and trailing blanks enclosed within single quotes are ignored. All other blanks are considered part of the project-ID.

5. Project-ID is saved with only the first 40 characters specified.

EXAMPLES:

1. PROJ = ’MY JOB’
2. The following project-ID will be truncated to 40 characters:

   ```
   PROJ ’CAR MODEL 1999 BODY FRAME SYM - PROTYP B RUN’
   ```

   and all subsequent restarts must have the statement.

   ```
   PROJ ’CAR MODEL 1999 BODY FRAME SYM - PROTYP B’
   ```

RESTART

Reuse Database From a Previous Run

Requests that data stored in a previous run be used in the current run.

FORMAT:

$$\text{RESTART}\left[\text{PROJECT} = \text{'project-ID'}\ \ \text{VERSION} = \left[\begin{matrix}\text{version-ID}\\ \text{LAST}\end{matrix}\right]\ \left[\begin{matrix}\text{KEEP}\\ \text{NOKEEP}\end{matrix}\right]\ \ \text{LOGICAL} = \text{dbname}\right]$$

DESCRIBERS:

Describer	Meaning
project-ID	Project identifier. See PROJ FMS statement. Must be enclosed in single right-hand quotation marks (’). (Character string, maximum of 40 characters; default is the project-ID specified on the PROJ FMS statement).
version-ID	Version number. (Integer > 0)
LAST	Specifies the last version under project-ID.
KEEP	Data stored under VERSION will remain on the database after the run is completed.
NOKEEP	Data stored under VERSION will be deleted from the database after the run is completed.
dbname	Specifies the logical name of an existing MASTER (master directory) DBset to be used for restart purposes. This MASTER and its associated database will be opened in a read-only mode to perform the restart, any new data will be written to the database for the current run.

REMARKS:

1. There may be only one RESTART statement in the File Management Section.

2. A new version-ID is automatically assigned whenever a restart is performed.

3. If project-ID or version-ID or both are specified and cannot be found, then a fatal message will be issued.

4. The RESTART statement is required to perform restarts in solution sequences 101 through 200.

5. If PROJECT is not specified, then the run will restart from the project-ID specified on the PROJ statement. (See Example 2 below.)

6. Databases created in one version cannot be directly restarted into a different version. Restrictions are typically documented in the current release guide; however, a DBLOCATE type restart might work.

EXAMPLES:

1. ```
   RESTART VERSION=7
   ```

   Version number 7 will be retrieved for this run (version 8). At the end of the run, version 7 will be deleted.

2. ```
   PROJ='FENDER'
   RESTART
   ```

   The last version under project-ID FENDER will be used in the current run.

3. ```
   ASSIGN RUN1='run1.MASTER'
   RESTART LOGICAL=RUN1
   ```

   The run1.MASTER and its associated database will be used (read only) for restart purposes.
   ```

Chapter 3: Executive Control Statements

3.1 Key to Descriptions

Braces { } indicate that a choice of describers is mandatory.

A brief sentence about the function of the statement is given.

Describers in uppercase letters are keywords that must be specified as shown.

LINK Link a Main SubDMAP

Links a main subDMAP to form a solution sequence.

Format:

LINK { *n* | subDMAP-name } [SOLO = SOLOUT-DBset

Default describers are shaded.

INCL-DBset [MAP | NOMAP], SOLN

If the describers are stacked vertically, then you can only specify one.

Brackets [ ] indicate that a choice of describers is optional.

Descri

n solution number of the main subDMAP. See the SOL state- nt description for the list of valid numbers. (Integer > 0).

subDM name of a main subDMAP. See the *NX Nastran DMAP Mod- ule Dictionary*.

Describers that are variables are shown in lower case. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

solout-DBset The name solution se

exeout-DBset The name only the O

incl-DBset The name See Remark 2.

newname A new name assigned to the OSCAR. The default is sub DMAP-

Each of the describers is discussed briefly.

MAP Prints th e sub- DMAPs

NOMAP Suppresses printing of the link map.

Remarks:

Remarks are arranged in order of importance. For example, they indicate the statement's relationship to other statements, restrictions and recommendations on use, and provide further details about the describers.

1. All DBsets s ZE. See the INIT FMS st

2. SubDMAP n the current run or obtai bjects in the following order:

   a. Objects created with the COMPILE statement in the current run.

   b. Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.

3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.

Executive Control Section

This section describes the Executive Control statements. These statements select a solution sequence and various diagnostics.

Most Executive Control statements are order independent. The exceptions are the COMPILE, COMPILER, ALTER, ENDALTER, and LINK statements. If used, the LINK statement must appear after all COMPILE statements. The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. The COMPILER statement also sets the defaults for subsequent COMPILE statements.

Executive Control Statement Summary

The Executive Control statements are summarized as follows:

ALTER	Specifies deletion and/or insertion of following DMAP statements.
APP	Specifies an approach in a solution sequence.
CEND	Designates the end of the Executive Control statements.
COMPILE	Requests compilation of specified subDMAPs or the NDDL file.
COMPILER	Specifies DMAP compilation diagnostics.
DIAG	Requests diagnostic output or modifies operational parameters.
ECHO	Controls the echo of Executive Control statements.
ENDALTER	Designates the end of a DMAP sequence headed by an ALTER.
GEOMCHECK	Specifies tolerance values and options for optional finite element geometry tests.
ID	Specifies a comment.
LINK	Requests the link of a main subDMAP.
SOL	Requests execution of a solution sequence or DMAP program.
TIME	Sets the maximum allowable execution time.

3.2 Executive Control Statement Descriptions

You can abbreviate Executive Control statements down to their first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

Description

A brief sentence about the function of the statement is given.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
COMPILE  SEDRCVR  SOUIN=MSCSOU,
  NOREF    NOLIST
```

Examples

One or more examples are given.

Describers

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), its allowable range, and its default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The remarks in the Remarks section are generally arranged in order of importance and indicate such things as the Executive Control statement's relationship to other statements, restrictions and recommendations on its use, and further descriptions of the describers.

$

Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

FORMAT:

$ followed by any characters out to column 80.

EXAMPLES:

```
$ TEST FIXTURE-THIRD MODE
```

REMARKS:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

ALTER

Insert and/or Delete DMAP Statements

Inserts and/or deletes DMAP statements in a subDMAP.

FORMAT:

```
ALTER k1 [,k2]
```

or

```
ALTER 'string1'[(occurrence,offset)] ,['string2'[(occurrence,offset)] ]
```

or

```
ALTER k1 , ['string2'[(occurrence,offset)] ]
```

or

```
ALTER 'string1'[(occurrence,offset)] , [k2]
```

DESCRIBERS:

Describer	Meaning
k1	If k2 or 'string2' is not specified, the subsequent DMAP statements will be inserted after either the statement number k1 or the 'string1', [(occurrence,offset)] reference point.
k1, k2	DMAP statements numbered k1 through k2 will be deleted and may be replaced with subsequent DMAP statements.
'string1'	If 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.
'string1', 'string2'	DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequent DMAP statements.
occurrence	This flag indicates which occurrence of the proceeding string is to be used, starting at the beginning of the subDMAP. (Integer > 0; Default = 1)

Describer	Meaning
offset	This flag indicates the offset from the reference DMAP statement. Depending on the sign, the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement. (Integer; Default = 0)

REMARKS:

1. The ALTER statement must be used in conjunction with the COMPILE Executive Control statement. Note: ALTER statements cannot be used in conjunction with a MALTER statement, and therefore, should never immediately follow this statement.

2. If a MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.

3. The ALTERs can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:

   - K2 or ‘string2’(occurence, offset) references must refer to a DMAP statement number that is greater than or equal to the k1 or ‘string1’(occurrence,offset) reference within a single ALTER statement.

   - K1 or ‘string1’ and k2 or ‘string2’ cannot overlap DMAP line positions with another ALTER that references the same subDMAP.

4. The ‘string1’ or ‘string2’ used as a search pattern will apply to one complete DMAP statement-i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string. All blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained. However, comments are ignored for the following type of alter:

   ```
   alter '^ *gp0'
   ```

5. Within a SUBDMAP, both ‘string1’ and ‘string2’ will be used to search for a pattern match starting at the beginning of the subDMAP-not at the current position of the last string match.

6. The special characters (meta characters) used for string searching are described in Remark 9. The characters <,>, and $, which are common DMAP characters, are also special meta characters. If they are to be used in the search string as

regular characters, then they must be preceded by a backward slash (\). For example, to find the string

```
IF(DDRMM >=-1)
```

the command is

```
ALTER 'IF (DDRMM \>=-1)' $
```


7. The ALTER statement must not exceed 72 characters (no continuations are allowed).

8. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (''). For example, the alter statement

```
ALTER 'string1'(r1,01),''
```

is equivalent to

```
ALTER 'string1'(r1,01),'string1'(r1,01)
```

The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

As another example, the alter statement

```
ALTER 'string1'(r1,01),''(r2,02)
```

is equivalent to

```
ALTER 'string1'(r1,01),'string1'(r2,02)
```

9. Meta characters:

.	Matches any *single* character except *newline*.
*	Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".*" means "match any number of characters".
[...] or < >	Matches any *one* of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some meta characters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character *not* in the list.

^ or ! or -|	Requires that the following regular expression be found at the beginning of the line. Note, that these meta characters may lead to UFM 802 if the preceding line is a comment.
$	Requires that the preceding regular expression be found at the end of the line.
\	Treats the following special character as an ordinary character. For example " \ ." stands for a period and " *" for an asterisk. Also, to search for a tic ('), the search string must be single quotes.
'	Marks the beginning and end of a pattern to be matched.
	Note: Nonportable characters such as ^ and [ ] should be replaced (e.g., ^ → ! and [] → <>) if portability is required. However, all the above characters are recognized by NX Nastran.

10. If a string-based alter uses the "!" in the expression (find occurrence at the beginning of line), it is possible NX Nastran will fail with USER FATAL MESSAGE 802.

EXAMPLES:

1. The following alter will insert a MATPRN DMAP statement after the first occurrence of the string 'SDR2' in subDMAP DSASTAT.

```
SOL 101
COMPILE DSASTAT $
ALTER 'SDR2' $
MATPRN OESDS1//$
CEND
```

2. The following alter will delete the second occurrence of the OUTPUT4 DMAP statement in subDMAP DSASTAT and replace it with a MATPRN DMAP statement.

```
SOL 101
COMPILE DSASTAT $
ALTER 'OUTPUT4'(2),'OUTPUT4'(2) $
$ OR
$ ALTER 'OUTPUT4'(2),'' $
MATPRN OESDS1//$
CEND
```

APP

Specify Solution Sequence Approach

Selects heat transfer analysis in the linear static SOL 101 or a coupled analysis combining heat transfer and structural analysis in SOL 153.

FORMAT:

APP approach.

DESCRIBERS:

Describer	Meaning	
approach	Specifies one of the following:	
	HEAT	Indicates that heat transfer is to be performed in SOL 101.
	COUPLED	Indicates a coupled analysis combining heat transfer and structural analysis in SOL 153.

REMARKS:

1. The APP statement is optional.
2. The APP HEAT statement applies only to linear static SOL 101. The APP HEAT statement is not required in SOLs 153 and 159, or in SOL 101 if PARAM,HEATSTAT,YES is specified.
3. The NASTRAN HEAT=101 statement is an alternate specification of APP HEAT. See “nastran Command and NASTRAN Statement”.

EXAMPLES:

The following requests a heat transfer rather than a structural analysis in Solution Sequence 101.

```
SOL 101
APP HEAT
```

CEND

End of Executive Control Delimiter

Designates the end of the Executive Control Section.

FORMAT:

CEND

REMARKS:

CEND is required unless an ENDJOB statement appears in the File Management Section.

COMPILE

Compile DMAP Statements

Requests the compilation of a subDMAP, subDMAP alter, or NDDL sequence.

FORMAT 1:

Compiles a subDMAP or subDMAP alter sequence

$$\text{COMPILE}\ \left[\begin{matrix}\text{SUBDMAP}\\ \text{DMAP}\end{matrix}\right]\text{subDMAP-name}\ [\text{SOUIN} = \text{souin-DBset}\ \text{SOUOUT} = \text{souout-DBset}$$

$$\text{OBJOUT} = \text{objout-DBset}\left[\begin{matrix}\text{LIST}\\ \text{NOLIST}\end{matrix}\right]\left[\begin{matrix}\text{REF}\\ \text{NOREF}\end{matrix}\right]\left[\begin{matrix}\text{DECK}\\ \text{NODECK}\end{matrix}\right]\Bigg]$$

FORMAT 2:

Compiles an NDDL sequence

$$\text{COMPILE NDDL} = \text{nddl-name}\left(\left\{\begin{matrix}\text{SOUIN} = \text{souin-dbset}\\ \text{SOUOUT} = \text{souout-dbset}\end{matrix}\right\}\left\{\begin{matrix}\text{LIST}\\ \text{NOLIST}\end{matrix}\right\}\left\{\begin{matrix}\text{REF}\\ \text{NOREF}\end{matrix}\right\}\left\{\begin{matrix}\text{DECK}\\ \text{NODECK}\end{matrix}\right\}\right)$$

EXAMPLES:

1. The following compiles an alter in subDMAP PHASEIDR.

```
COMPILE PHASE1DR
ALTER 'CALL PHASE1A'
CEND
```

2. The following compiles a subDMAP called MYDMAP. (SUBDMAP and END are DMAP statements; see *NX Nastran DMAP Programmer's Guide*.)

```
COMPILE MYDMAP LIST REF
SUBDMAP MYDMAP $
.
.
```

```
.
END $
CEND
```

3. The following obtains a listing of the NDDL.

```
ACQUIRE NDDL
COMPILE NDDL=NDDL LIST
CEND
```


DESCRIBERS:

Describer	Meaning
subDMAP-name	The name of a subDMAP sequence. SubDMAP-name must be 1 to 8 alphanumeric characters in length and the first character must be alphabetic. The keywords DMAP and SUBDMAP are optional and do not have to be specified.
nddl-name	The name of an NDDL sequence. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The keyword NDDL must be specified.
souin-DBset	The name of a DBset from which the subDMAP or NDDL source statements will be retrieved. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is MSCSOU if the next statement is not a subDMAP statement.
souout-DBset	The name of a DBset on which the subDMAP or NDDL source statements will be stored. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is the SCRATCH DBset.
objout-DBset	The name of a DBset on which the subDMAP object code will be stored. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is the OBJSCR DBset.
LIST, NOLIST	LIST requests a compiled listing of the subDMAP or NDDL sequence. NOLIST suppresses the listing. NOLIST is the default.

Describer	Meaning
REF, NOREF	REF requests a compiled cross reference of the subDMAP or NDDL sequence. NOREF suppresses the cross reference. NOREF is the default.
DECK, NODECK	DECK requests the subDMAP or NDDL source statements to be written to the PUNCH file. NODECK suppresses the writing to the PUNCH file. NODECK is the default.

REMARKS:

1. SubDMAP names for NX Nastran solution sequences are given in the SOL statement description. The “COMPILER LIST REF” statement may be used to determine the appropriate subDMAP-name.

2. If a subDMAP is being compiled and SOUIN=souin-DBset is specified, then an ALTER Executive Control statement or an INCLUDE statement which contains an ALTER statement as the first non comment line, must appear immediately after these. If not, then the SUBDMAP DMAP statement must appear immediately after these statements. See the *NX Nastran DMAP Programmer’s Guide*.

3. DBsets USRSOU and USROBJ are no longer automatically created. They must be initialized by the INIT FMS statement and then may be specified for souin-dbset (or souout-dbset) and objout-dbset, respectively. They may be used to store the subDMAP source statements and object code on the primary database for re-execution in a subsequent run. For example:

   In the first run, the following COMPILE statement compiles and stores a subDMAP called MYDMAP.

   ```
   COMPILE MYDMAP SOUOUT=USRSOU OBJOUT=USROBJ
   SUBDMAP MYDMAP $
   .
   .
   END $
   CEND
   ```

   In the second run, the SOL statement is used to execute the MYDMAP stored in the previous run. The LINK statement is required to retrieve the object code from the USROBJ DBset.

   ```
   SOL MYDMAP
   LINK MYDMAP INCL=USROBJ
   CEND
   ```

In the third run, the COMPILE statement is used to alter MYDMAP and execute.

```
SOL MYDMAP
COMPILE MYDMAP SOUIN=USRSOU
ALTER...
.
.
.
CEND
```


4. If SOUOUT or OBJOUT is specified and a subDMAP with the same name as the subDMAP-name already exists on the database, then its source statements or object code will be replaced.

5. A COMPILE statement is required for each subDMAP to be compiled. If two or more COMPILE statements reference the same subDMAP name, then only the last is used in the linking of the object code. If the COMPILE statement is being used only to alter a subDMAP and two or more COMPILE statements reference the same subDMAP name, then the multiple alters are assembled and the subDMAP is compiled only once.

6. Only one COMPILE statement for an NDDL sequence may be specified in the input file.

   - SOUIN=souin-DBset requests only a compilation of the NDDL sequence stored on souin-DBset for purposes of obtaining a listing or a cross reference, and it cannot be modified with the ALTER statement. See Remark 3 . COMPILE NDDL=NDDL SOUIN=MSCSOU LIST requests a listing of the NX Nastran NDDL sequence. The ACQUIRE FMS statement or the SOL statement must be specified in order to attach the corresponding Delivery Database.

   - To alter the NX Nastran NDDL sequences, the entire modified NDDL sequence is included after the COMPILE statement and SOUIN=souin-DBset is not specified.

   - SOUOUT=souout-DBset requests the storage of the NDDL source statements on the souout-DBset and may not be specified with SOUIN=souin-DBset.

7. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, and NODECK. In other words, if LIST or NOLIST, REF or NOREF, or DECK or NODECK is not specified, then the corresponding option on the COMPILER statement will be used. In the following example, REF on the COMPILER statement will override the default of NOREF on the COMPILE statement.

```
COMPILER REF
COMPILE MYDMAP
```

8. MSCSOU and MSCOBJ, specified with SOUOUT and OBJOUT, are special DBsets similar to USRSOU and USROBJ except that they are used in the creation or modification of a delivery database. For an example application, see the *NX Nastran Installation and Operations Guide.*

COMPILER

DMAP Compiler Output Options

Requests compilation of a DMAP sequence and/or overrides defaults on the COMPILE statement.

FORMAT:

$$\text{COMPILER}[=]\begin{bmatrix}\text{LIST}\\ \text{NOLIST}\end{bmatrix}\begin{bmatrix}\text{DECK}\\ \text{NODECK}\end{bmatrix}\begin{bmatrix}\text{REF}\\ \text{NOREF}\end{bmatrix}\begin{bmatrix}\text{GO}\\ \text{NOGO}\end{bmatrix}\begin{bmatrix}\text{SORT}\\ \text{NOSORT}\end{bmatrix}$$

DESCRIBERS:

Describer	Meaning
LIST, NOLIST	LIST requests the compilation listing of the solution sequence. NOLIST suppresses the listing.
DECK, NODECK	DECK requests that the DMAP source statements of the solution sequence be written to the PUNCH file. NODECK suppresses the DECK option.
REF, NOREF	REF requests a compilation cross reference. NOREF suppresses a compilation cross reference.
GO, NOGO	GO requests the execution of the solution sequence following compilation. NOGO requests termination following compilation.
SORT, NOSORT	SORT compiles subDMAPs in alphabetical order. NOSORT compiles subDMAPs in calling sequence order.

REMARKS:

1. REF is equivalent to DIAG 4. LIST is equivalent to DIAG 14. DECK is equivalent to DIAG 17.

2. NOGO is an alternative to NOEXE on the SOL statement.

3. This statement provides the user a means of obtaining a compilation or source listing, or both, of a complete solution sequence, including all the component subDMAPs.

4. See the COMPILE statement to compile a single subDMAP.

5. This statement also requests the automatic link of the solution sequence. Therefore, all objects must be created in the current run or obtained from the DBset such as USROBJ. See the COMPILE statement for how to create and store objects.

6. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, NODECK on the COMPILE entry when they are not explicitly specified. However, COMPILER LIST produces a list of the entire solution sequence. If a listing of only specific subdmaps are desired, then COMPILER LIST should not be specified and the LIST request should be made on the COMPILE entry.

```
COMPILER REF
COMPILE MYDMAP
```

EXAMPLES:

```
COMPILER=LIST
```

DIAG

Request Diagnostic Output

Requests diagnostic output or special options.

FORMAT:

DIAG [=] k1[k2, ..., kn]

DESCRIBERS:

Describer	Meaning
ki	A list separated by commas and/or spaces of desired diagnostics.

REMARKS:

1. The DIAG statement is optional.
2. Multiple DIAG statements are allowed.
3. The following table lists the possible values for ki and their corresponding actions:

k=1	Dumps memory when a nonpreface fatal message is generated.
k=2	Prints database directory information before and after each DMAP statement. Prints bufferpooling information.
k=3	Prints “DATABASE USAGE STATISTICS” after execution of each functional module. This message is the same as the output that appears after the run terminates.
k=4	Prints cross-reference tables for compiled sequences. Equivalent to the COMPILER REF statement.
k=5	Prints the BEGIN time on the operator’s console for each functional module.
k=6	Prints the END time for each functional module in the log file or day file and on the operator’s console. Modules that consume less time than the threshold set by SYSTEM(20) do not create a message.

k=7 Prints eigenvalue extraction diagnostics for the Complex Determinate method.

k=8 Prints matrix trailers as the matrices are generated in the Execution Summary Table.

k=9 Not used.

k=10 Uses alternate nonlinear loading in linear transient analysis. Replaces N_{n+1} with $(N_{n+1} + N_n + N_{n-1})/3$.

k=11 DBLOAD, DBUNLOAD, and DBLOCATE diagnostics.

k=12 Prints eigenvalue extraction diagnostics for complex Inverse Power, complex Lanczos and ISRR methods.

k=13 Prints the open core length (the value of REAL on VAX computers).

k=14 Prints the solution sequence. Equivalent to the COMPILER LIST statement.

k=15 Prints table trailers.

k=16 Traces real inverse power eigenvalue extraction operations.

k=17 Punches solution sequences. Equivalent to the COMPILER DECK statement.

k=18 In aeroelastic analysis, prints internal grid points specified on SET2 Bulk Data entries.

k=19 Prints data for MPYAD and FBS method selection in the Execution Summary Table.

k=20 Similar to DIAG 2 except the output appears in the Execution Summary Table and has a briefer and more user-friendly format. However, the .f04 file will be quite large if DIAG 20 is specified with an NX Nastran solution sequence. A DMAP Alter with DIAGON(20) and DIAGOFF(20) is recommended. DIAG 20 also prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics. See the *NX Nastran DMAP Programmer's Guide*.

k=21 Prints diagnostics of DBDIR and DBENTRY table.

k=22 Outputs EQUIV and EQUIVX module diagnostics.

k=23 Not used.

k=24 Prints files that are left open at the end of a module execution. Also prints DBVIEW diagnostics.

k=25 Outputs internal plot diagnostics.

k=26 Outputs dynamic file allocation diagnostics.

k=27 Prints Input File Processor (IFP) table. See section 4.3 of the *NX Nastran Programmer's Guide* for more information on IFP.

k=28 Punches the link specification table (XBSBD). The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

k=29 Process link specification table update. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

k=30 In link 1, punches the XSEMii data (i.e., set ii via DIAG 1 through 15). The Bulk Data and Case Control Sections are ignored, and no analysis is performed. After link 1, this turns on BUG output. Used also by MATPRN module. See also Remark 5 on the "TSTEP" Bulk Data entry.

k=31 Prints the link specification table and module properties list (MPL) data. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

k=32 Prints diagnostics for XSTORE and PVA expansion.

k=33 Not used.

k=34 Turns off plot line optimization.

k=35 Prints diagnostics for 3-D slideline contact analysis in SOLs 106 and 129.

k=36 Prints extensive tables generated by the GP0 module in p-version analysis.

k=37 Disables the superelement congruence test option and ignores User Fatal Messages 4277 and 4278. A better alternative is available with PARAMeter CONFAC. See "Parameters and Parameter Applicability Tables".

k=38 Prints material angles for CQUAD4, CQUAD8, CTRIA3, and CTRIA6 elements. The angle is printed only for elements that specify MCID in field 8 of the connection entry.

k=39 Traces module FA1 operations and aerodynamic splining in SOLs 145 and 146.

k=40 Prints constraint override/average information for edges and faces in p-adaptive analysis.

k=41 Traces GINO OPEN/CLOSE operations.

k=42 Not used.

k=43 Not used.

k=44 Prints a mini-dump for fatal errors and suppresses user message exit.

k=45 Prints the same database directory information as DIAG 2 except that it prints only after each DMAP statement.

k=46 Used by Siemens development for GINO printout.

k=47 Prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics.

k=48 Used by Siemens development for GINO printout.

k=49 Prints the DMAP execution time summary table.

k=50 Traces the nonlinear solution in SOLs 106, 129, 153, and 159. Prints subcase status; echoes NLPARM, NLPCI, and TSTEPNL entry fields; and prints initial arc-length. Prints iteration summary only in SOLs 129, and 159.

In static aeroelastic analysis (SOL 144), prints transformation information associated with the generation of the DJX matrix in the ADG module and intermediate solutions information in the ASG module.

k=51 Prints intermediate displacement, load error vectors, and additional iteration information helpful to debugging in SOLs 106, 129, 153, and 159.

k=52 Disables the printing of errors at each time step in SOLs 129 and 159.

k=53 Prints the MESSAGE module output in the execution summary table.

k=54 Print linker debug diagnostics.

k=55 Output performance timing.

k=56 Prints all DMAP statements and RESTART deletions. (Extended print of Execution Summary table).

k=57 Outputs executive table (XDIRLD) performance timing and last-time-used (LTU) diagnostics.

k=58 Outputs data block deletion debug and timing constants echo.

k=59 Prints buffpool debug.

k=60	Prints diagnostics for data block cleanup at the end of each module execution in subroutines DBCLN, DBEADD, and DBERPL.
k=61	GINO block allocator diagnostics.
k=62	GINO block manager diagnostics.
k=63	Prints each item checked by the RESTART module and its NDDL description.
k=64	Not used.

EXAMPLES:

```
DIAG 8,53
```

or

```
DIAG 8
DIAG 53
```

See the “NX Nastran Output Files” in the *NX Nastran User’s Guide.*

DOMAINSOLVER

Domain Decomposition Solution Method

This is an obsolete interface to various domain decomposition methods. If you are looking for FDMODES or GDMODES, use the numseg and dmp keywords. For GDSTAT/LDSTAT, use the dmp and dstat keywords. For ACMS, use the RDMODES method instead with the nrec keyword. See the *Parallel Processing Guide* for details.

ECHO

Control Printed Echo

Controls the echo (printout) of the Executive Control Section.

FORMATS:

```
ECHOOFF
ECHOON
```

REMARKS:

1. The ECHO statement is optional.
2. ECHOOFF suppresses the echo of subsequent Executive Control statements. ECHOON reactivates the echo after an ECHOOFF statement.

ENDALTER

End of DMAP Alter

Designates the end of an alter.

FORMAT:

ENDALTER

REMARKS:

The ENDALTER statement is required when using an alter unless the alter package ends with the CEND, COMPILE, or LINK statement.

GEOMCHECK

Specifies Geometry Check Options

Specifies tolerance values for (optional) finite element geometry tests.

FORMAT:

$$\text{GEOMCHECK test_keyword [= tol_value], [MSGLIMIT = n],}$$

$$\left[\text{MSGTYPE} = \begin{matrix}\text{FATAL}\\ \text{INFORM}\\ \text{WARN}\end{matrix}\right], [\text{SUMMARY}], [\text{NONE}]$$

DESCRIBERS:

Describer	Meaning
test_keyword	A keyword associated with the particular element geometry test. See Remark 2 for a list of acceptable selections.
tol_value	Tolerance value to be used for the specified test. See Remark 2 for default values of the test tolerances.
n	The maximum number of messages that will be produced. The default is 100 messages for each element type. See Remark 3.
FATAL	Geometry tests that exceed tolerance values produce fatal messages. See Remark 4.
INFORM	Geometry tests that exceed tolerance values produce informative messages. See Remark 4.
WARN	Geometry tests that exceed tolerance values produce warning messages. See Remark 4.
SUMMARY	A summary table of the geometry tests performed is produced. No individual element information messages are output.
NONE	None of the optional element geometry tests will be performed.

EXAMPLES:

1. Adjust the tolerance for the CQUAD4 element skew angle test to 15.0 degrees and limit messages to 50.

```
GEOMCHECK   Q4_SKEW=15.0,MSGLIMIT=50
```

2. Limit messages to 500 for each element type.

```
GEOMCHECK  MSGLIMIT=500
```

3. Adjust the tolerance for the CQUAD4 element taper and set the message type to fatal for all tests.

```
GEOMCHECK  Q4_TAPER=0.4,MSGTYPE=FATAL
```

4. Request summary table output only using default tolerance values.

```
GEOMCHECK  SUMMARY
```

REMARKS:

1. There are two categories of element checks in an NX Nastran solution:

   - The system controlled checks will always occur. As the name implies, there is no user control to these checks. The system controlled checks will always produce a fatal error if a condition is found which prevents the analysis to proceed.

     See “System Element Checks” in the *NX Nastran User’s Guide* for descriptions of these checks.

   - GEOMCHECK are the optional, user controlled checks, and can optionally generate a fatal error if it finds an element which fails the user defined criteria. By default, GEOMCHECK does not produce fatal errors. Element checks which are controlled with the GEOMCHECK statement are summarized in Remark 2. See “User Controlled Element Checks” in the *NX Nastran User’s Guide* for detailed descriptions of all checks. The GEOMCHECK element checks will occur using default values when the GEOMCHECK statement is not included. GEOMCHECK controls are available for the shell elements CQUAD4, CQUADR, CTRIA3, CTRIAR, the solid elements CHEXA, CPENTA, CPYRAM, CTETRA, CHEXCZ, CPENTCZ the axisymmetric elements CTRAX3, CTRAX6, CQUADX4, CQUADX8, the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, the chocking elements CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8, the CBAR

element, and the CBEAM element. Multiple GEOMCHECK directives may be present. Continuations are acceptable.

> **Note**
>
> Be aware that when MSGTYPE=inform, GEOMCHECK may flag a poor quality element with an informational message, the element may pass the system check, and the solve may complete. MSGTYPE should be set appropriately if you expect fatal errors in these cases. Fatal errors can also be forced by assigning a negative value to MSGLIMIT, for example, MSGLIMIT=-100. This is also useful since a specific "FAIL" message will be reported for each element failing the GEOMCHECK criteria. See Remark 3 for more information on MSGLIMIT.

2. The **test_keyword** describer can optionally be used to modify the default thresholds. The following table lists the **test_keywords** and their defaults. The **test_keyword** inputs are Real ≥ 0.0. See "User Controlled Element Checks" in the *NX Nastran User's Guide* for detailed descriptions of all element checks.

test_keyword	Default Threshold	Summary
CQUAD4 and CQUADR Shell Element Checks		
Q4_SKEW	Failure when < 30.0	Skew angle in degrees
Q4_TAPER	Failure when > 0.5	Taper ratio
Q4_WARP	Failure when > 0.05	Surface warping factor
Q4_IAMIN	Failure when < 30.0	Minimum Interior Angle in degrees
Q4_IAMAX	Failure when > 150.0	Maximum Interior Angle in degrees
QAD_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio (CQUAD4)
CTRIA3 and CTRIAR Shell Element Checks		
T3_SKEW	Failure when < 10.0	Skew angle in degrees
T3_IAMAX	Failure when > 160.0	Maximum Interior Angle in degrees
CQUAD8 Shell Element Checks		
Q8_SKEW	Failure when < 30.0	Skew angle in degrees
Q8_TAPER	Failure when > 0.5	Taper ratio
Q8_IAMIN	Failure when < 30.0	Minimum Interior Angle in degrees
Q8_IAMAX	Failure when > 150.0	Maximum Interior Angle in degrees

test_keyword	Default Threshold	Summary
Q8_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
Q8_EPLR	Failure when < 0.5	Edge point length ratio
CTRIA6 Shell Element Checks		
TA6_IAMN	Failure when < 10.0	Minimum Interior Angle in degrees
TA6_IAMX	Failure when > 160.0	Maximum Interior Angle in degrees
TA6_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
TA6_EPLR	Failure when < 0.5	Edge point length ratio
CTETRA Solid Element Checks		
TET_AR	Failure when >100.0	Longest edge to shortest height aspect ratio
TET_EPIA	Failure when > 30.0	Edge point included angle
TET_EPLR	Failure when < 0.5	Edge point length ratio
CHEXA Solid Element Checks		
HEX_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
HEX_EPIA	Failure when > 30.0	Edge point included angle
HEX_EPLR	Failure when < 0.5	Edge point length ratio
HEX_TK	Failure when > 0.05	Ratio of the difference between thickness in stacking direction as defined by grids and as defined by ply thickness specification to the thickness in the stacking direction as defined by grids for CHEXA elements that reference PCOMPS entries only
HEX_TP	Failure when > 1.2	Ratio of longest edge to shortest edge in stacking direction for CHEXA elements that reference PCOMPS entries only
HEX_WARP	Failure when < 0.707	Face warp coefficient
CPYRAM Solid Element Checks		
PYR_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
PYR_EPIA	Failure when > 30.0	Edge point included angle
PYR_EPLR	Failure when < 0.5	Edge point length ratio
PYR_WARP	Failure when < 0.707	Face warp coefficient
CPENTA Solid Element Checks		

test_keyword	Default Threshold	Summary
PEN_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
PEN_EPIA	Failure when > 30.0	Edge point included angle
PEN_EPLR	Failure when < 0.5	Edge point length ratio
PEN_TK	Failure when > 0.05	Ratio of the difference between thickness in stacking direction as defined by grids and as defined by ply thickness specification to the thickness in the stacking direction as defined by grids for CPENTA elements that reference PCOMPS entries only
PEN_TP	Failure when > 1.2	Ratio of longest edge to shortest edge in stacking direction for CPENTA elements that reference PCOMPS entries only
PEN_WARP	Failure when < 0.707	Quadrilateral face warp coefficient
CHEXCZ Solid Element Checks		
HEX_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
HEX_EPIA	Failure when > 30.0	Edge point included angle
HEX_EPLR	Failure when < 0.5	Edge point length ratio
HEX_WARP	Failure when < 0.707	Face warp coefficient
CPENTCZ Solid Element Checks		
PEN_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
PEN_EPIA	Failure when > 30.0	Edge point included angle
PEN_EPLR	Failure when < 0.5	Edge point length ratio
PEN_WARP	Failure when < 0.707	Face warp coefficient
Checks for the Axisymmetric Elements CTRAX3 and CTRAX6, the Plane Stress Elements CPLSTS3 and CPLSTS6, the Plane Strain Elements CPLSTN3 and CPLSTN6, the Chocking Elements CCHOCK3 and CCHOCK6		
TRX_IAMN	Failure when < 30.0	Minimum Interior Angle in degrees
TRX_IAMX	Failure when > 160.0	Maximum Interior Angle in degrees
TRX_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
TRX_EPLR	Failure when < 0.5	Edge point length ratio (CTRAX6, CPLSTS6, CPLSTN6, CCHOCK6)

test_keyword	Default Threshold	Summary
Checks for the Axisymmetric Elements CQUADX4 and CQUADX8, the Plane Stress Elements CPLSTS4 and CPLSTS8, the Plane Strain Elements CPLSTN4 and CPLSTN8, the Chocking Elements CCHOCK4 and CCHOCK8		
QDX_IAMN	Failure when < 30.0	Minimum Interior Angle in degrees
QDX_IAMX	Failure when > 150.0	Maximum Interior Angle in degrees
QDX_AR	Failure when > 100.0	Longest edge to shortest edge aspect ratio
QDX_SKEW	Failure when < 30.0	Skew angle in degrees
QDX_TAPR	Failure when > 0.5	Taper ratio
QDX_EPLR	Failure when < 0.5	Edge point length ratio (CQUADX8, CPLSTS8, CPLSTN8, CCHOCK8)
CBEAM and CBAR Element Checks		
BEAM_OFF	Failure when > 0.15	CBEAM element offset length ratio
BAR_OFF	Failure when > 0.15	CBAR element offset length ratio

3. A single line summarizing the results of all tests for an element will be output if any of the geometry tests exceeds the test tolerance. Only the first n of these messages will be produced. A summary of the test results indicating the number of tolerances exceeded as well as the element producing the worst violation is also output. If the SUMMARY keyword has been specified, only the summary table is produced and none of the single line element messages will be output.

   Solutions which fail with numerical problems, i.e. negative Jacobian, report differently based on the sign of MSGLIMIT. When troubleshooting a failed analysis, you may find it useful to switch between the following two strategies:

Value of MSGLIMIT	Meaning when numerical fatal errors occur
> 0 (positive)	If numerical problems are found resulting from poor quality elements, a FATAL ERROR message is generated without the offending element ID, the analysis terminates, but the GEOMCHECK criteria *is* processed and output for all elements. This strategy works when only a few elements fail the GEOMCHECK. However, if there are hundreds or thousands of failures, diagnosing the critical elements could become difficult. In addition, the offending element ID may or may not have failed the GEOMCHECK criteria depending on the criteria you entered.

< 0 (negative)	If numerical problems are found resulting from poor quality elements, a FATAL ERROR message is generated which includes the offending element ID, the analysis terminates, yet GEOMCHECK *is not* processed for all elements. The downside of using a negative MSGLIMIT is if the model contains multiple offending elements, the procedure of fixing/rerunning needs to be repeated until all offending elements have been corrected.

4. When SUMMARY is not specified, each geometry test that exceeds the tolerance will be identified in the single line output summary by an indicator based on the specification for MSGTYPE. For the FATAL option, the indicator is “FAIL”; for the INFORM option, it is “xxxx”; for the WARN option, it is “WARN”. If the FATAL option is specified and any test fails, the run is terminated.

ID

Comment

Specifies a comment.

FORMAT:

ID [=] i1, i2

DESCRIBERS:

Describer	Meaning
i1, i2	Character strings (1 to 8 characters in length and the first character must be alphabetic).

REMARKS:

The ID statement is optional and not used by the program.

INCLUDE

Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

FORMAT:

INCLUDE ‘filename’

DESCRIBERS:

Describer	Meaning
filename	External file to be inserted. The ‘*directory_path/filename*’ must begin and end with the single quote character.

EXAMPLES:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA.

```
SOL 101
INCLUDE ‘MYEXEC.DATA’
CEND
TITLE = STATIC ANALYSIS
LOAD=100
BEGIN BULK
ENDDATA
```

REMARKS:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.

2. Each line has a 72 character limit. Multiple lines can be used when file names are long. The entire string must begin and end with the single quote character.

   For example,

D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat

can be defined on multiple lines:

```
INCLUDE 'D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\fold
er8\folder9\mydata.dat'
```

The following input format is also supported:

```
INCLUDE 'D:\folder1\folder2\folder3\
         folder4\folder5\
     folder6\folder7\
        folder8\folder9\mydata.dat'
```

LINK

Link a Main SubDMAP

Links a main subDMAP to form a solution sequence.

FORMAT:

$$\text{LINK} \left\{ \begin{array}{c} \text{n} \\ \text{subDMAP-name} \end{array} \right\} [\text{SOLOUT} = \text{solout-DBset} \quad \text{EXECOUT - exeout-DBset}$$

$$\text{INCLUDE - incl-DBset} \left[\begin{array}{c} \text{MAP} \\ \text{NOMAP} \end{array} \right] \text{SOLNAME} = \text{newname} \;]$$

DESCRIBERS:

Describer	Meaning
n	The solution number of the main subDMAP. See “SOL” for the list of valid numbers. (Integer>0)
subDMAP-name	The name of a main subDMAP. See the *NX Nastran DMAP Programmer’s Guide*. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
solout-DBset	The name of a DBset where the solution sequence executable and the link table of the solution sequence may be stored. See Remark 6 . (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
exeout-DBset	The name of an alternate DBset different than solout-DBset where only the solution sequence executable may be stored. See Remark 6 . (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)

Describer	Meaning
incl-DBset	The name of a DBset where other subDMAP objects are obtained. See Remark 2 . (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
newname	A new name which is referenced by the SOL statement. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic; default is subDMAP-name.)
MAP	Prints the link map. A link map will give the name of all the subDMAPs that make up the solution sequence.
NOMAP	Suppresses printing of the link map.

REMARKS:

1. All DBsets specified on this statement must have the same BUFFSIZE. See "INIT" .

2. SubDMAP objects are created with the COMPILE statement either in the current run or obtained from previous runs. The LINK statement collects objects in the following order:

   - Objects created with the COMPILE statement in the current run.

   - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.

3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.

4. The LINK statement must appear after all the COMPILE packages, if any. A compile package begins with the COMPILE statement and is delimited by the ENDALTER, CEND, LINK, or another COMPILE statement.

5. The link table is necessary for COMPILER (or DIAG 4, 14, 17) Executive statement requests and the automatic link process.

6. EXEOUT is useful in building delivery databases where executables are not to be saved. EXEOUT defaults to the same DBset as specified by SOLOUT.

EXAMPLES:

1. `LINK STATICS`

   Links the STATICS main subDMAP. The program links any subDMAPs compiled in this run, with any other subDMAP objects called in STATICS and stored on the MSCOBJ DBset.

2. `LINK MYDMAP,SOLNAM=STATICS,SOLOUT=USROBJ,`
   `NOMAP,INCLUDE=USROBJ`

   Links MYDMAP and renames the solution sequence executable to STATICS. The executable will be saved on the USROBJ DBset. The order of search for subDMAP objects is:

   - Compiled subDMAP in this run.
   - USROBJ DBset.

MALTER

Insert and/or Delete DMAP Statements in Solution Sequences

Inserts or deletes DMAP statements by allowing a global “string” search across all subDMAPs within the current solution sequence.

FORMAT:

```
MALTER 'string1'[(occurrence,offset)] , ['string2'[(occurrence,offset)] ]
```

or

```
MALTER 'string1'[(occurrence,offset)] , [k2]
```

DESCRIBERS:

Describer	Meaning
‘string1’	If ‘string2’ or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of ‘string1’.
‘string1’, ‘string2’	DMAP statements beginning with the first occurrence of ‘string1’ through DMAP statements containing the first occurrence of ‘string2’ will be deleted and may be replaced with subsequent DMAP statements.
k2	If k2 is specified, it is applied to the subDMAP in which ‘string1’ was found. (Integer > 0)
occurrence	This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP. (Integer > 0, Default = 1)
offset	This flag indicates the offset from the referenced DMAP statement. Depending on the sign, the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement. (Integer, Default = 0)

REMARKS:

1. If a MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.

2. The MALTER statement can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:

   - k2 or ‘string2’(occurence,offset) references must refer to a DMAP line number that is greater than or equal to the k1 or ‘string1’(occurrence,offset) reference within a single MALTER statement.

   - ‘string1’ and k2 or ‘string2’ cannot overlap DMAP line positions with another MALTER that references the same subDMAP.

3. The ‘string1’ or ‘string2’ used as a search pattern will apply to one complete DMAP statement-i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string. All blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained.

4. The special characters used for string searching are described in Remark 9 . The characters <, >, and $, which are common DMAP characters, are also special meta characters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string

   ```
   IF (DDRMM >=-1)
   ```

   the command is

   ```
   ALTER ‘IF (DDRMM \>=-1)’ $
   ```

5. ‘string2’ (r2,02) can be defaulted to ‘string1’ (r1,01) by using a null string (‘’). For example, the alter statement

   ```
   MALTER ‘string1’(r1,01),‘’
   ```

   is equivalent to

   ```
   MALTER ‘string1’(r1,01),‘string1’(r1,01)
   ```

   The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

   As another example, the alter statement

   ```
   MALTER ‘string1’(r1,01),‘’(r2,02)
   ```

   is equivalent to

```
MALTER 'string1'(r1,01),'string1'(r2,02)
```

6. The existing COMPILE statement options, such as LIST, XREF, SOUIN, etc., cannot be directly specified on the new MALTER statement. They are obtained as follows:

   - If a COMPILE statement exists for the subDMAP referenced by the MALTER, then options from this COMPILE statement will be used.

   - Else, they will be taken from the COMPILER statement, provided the LIST, and SORT option is always on.

7. The MALTER string search order is as follows:

   - All COMPILE statement references that are part of the existing solution sequence (i.e., SOL=) are searched first.

   - Then, all remaining subDMAPs in the solution sequence are searched in ascending alphabetical order.

   - Within a subDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP, not at the current position of the last string match.

8. The MALTER statement must not exceed 72 characters (no continuations are allowed).

9. Meta characters:

.	Matches any *single* character except *newline*.
*	Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".*" means "match any number of characters."
[...] or < >	Matches any *one* of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some meta characters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character *not* in the list.
^ or ! or ¬|	Requires that the following regular expression be found at the beginning of the line.

$ Requires that the preceding regular expression be found at the end of the line.

\ Treats the following special character as an ordinary character. For example " \." stands for a period and " *" for an asterisk. Also, to search for a tic ('), the search string must be " \' ".

' Marks the beginning and end of a pattern to be matched.

Note

Nonportable characters such as ^ and [ ] should be replaced (e.g., ^ → ! and [] → <>) if portability is required. However, all the above characters are recognized by NX Nastran.

10. Labels for use with the MALTER have been included in the solution sequences. See Table 3-1. These labels will be maintained in future versions and it is strongly suggested that alters which use the MALTER command take advantage of the unique MALTER labels. Use of the MALTER labels will significantly reduce the time required to convert alters between versions.

Table 3-1. DMAP Labels and Corresponding SubDMAP Positions

DMAP MALTER Labels
$MALTER:AFTER PREFACE MODULES
$MALTER:TOP OF PHASE 1 SUPERELEMENT LOOP, AFTER PARAMETERS AND QUALIFIERS SET
$MALTER:AFTER SUPERELEMENT STIFFNESS, VISCOUS DAMPING, MASS, AND ELEMENT STRUCTURAL DAMPING GENERATION (KJJZ, BJJZ, MJJZ, K4JJ)
$MALTER:AFTER X2GG MATRICES READ (K2JJ, M2JJ, B2JJ)
$MALTER:AFTER TOTAL SUPERELEMENT STIFFNESS, VISCOUS DAMPING, AND MASS FORMULATED, STRUCTURAL + DIRECT INPUT
$MALTER:AFTER SUPERELEMENT LOAD GENERATION (PJ)
$MALTER:AFTER UPSTREAM SUPERELEMENT MATRIX AND LOAD ASSEMBLY (KGG, BGG, MGG, K4GG, PG)
$MALTER:AFTER SUPERELEMENT MATRIX AND LOAD REDUCTION TO A-SET, STATIC AND DYNAMIC (KAA, KLAA, MAA, MLAA, BAA, K4AA, PA)
$MALTER:BOTTOM OF PHASE 1 SUPERELEMENT LOOP
$MALTER:AFTER X2PP MATRICES READ (K2PP, M2PP, B2PP)
$MALTER:AFTER SUPERELEMENT DISPLACEMENT RECOVERY (UG)
$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT1 (OUGV1, OES1, OEF1, ETC.)

Table 3-1. DMAP Labels and Corresponding SubDMAP Positions
DMAP MALTER Labels
$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT2 (OUGV2, OES2, OEF2, ETC.)
$MALTER:BOTTOM OF SUPERELEMENT DATA RECOVERY LOOP
$MALTER:USERDMAP - AFTER CALL PREFACE

EXAMPLES:

1. The following MALTER will insert a MATPRN DMAP statement to print the KJJ matrix for each superelement.

```
SOL 101
MALTER 'MALTER:AFTER SUPERELEMENT STIFFNESS .* GENERATION'
MESSAGE //'SEID='/SEID $
MATPRN KJJZ/ $
```

2. The following MALTER will add a user DMAP after the PREFACE modules in SOL 100 (USERDMAP).

```
SOL 101
MALTER 'AFTER CALL PREFACE'
.
.
.
```

SOL

Execute a Solution Sequence

Specifies the solution sequence or main subDMAP to be executed.

FORMAT:

$$\text{SOL} \left\{ \begin{array}{c} \text{n} \\ \text{subDMAP-name} \end{array} \right\} [\text{SOLIN} = \text{obj-DBset} \quad \text{NOEXE}]$$

DESCRIBERS:

Describer	Meaning
n	Solution number. See Remark 6 . for the list of valid numbers. (Integer > 0)
subDMAP-name	The name of a main subDMAP. See the *NX Nastran DMAP Programmer's Guide*. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
obj-DBset	The character name of a DBset where the OSCAR is stored. See Remarks 1 and 2. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
NOEXE	Suppresses execution after compilation and/or linkage of the solution is complete. Also, the Bulk Data and Case Control Sections are not read or processed.

REMARKS:

1. If a SOLIN keyword is not given and if there are no LINK statements within the input data, the program will perform an automatic link. The program will first collect the objects created in the current run by the COMPILE statement and the

remaining objects stored in the MSCOBJ DBset. The program will then perform an automatic link of the collected objects.

2. If the SOLIN keyword is not given but a LINK statement is provided, the SOLIN default will be obtained from the SOLOUT keyword on the LINK statement.

3. The OSCAR (Operation Sequence Control ARray) defines the problem solution sequence. The OSCAR consists of a sequence of entries with each entry containing all of the information needed to execute one step of the problem solution. The OSCAR is generated from information supplied by the user's entries in the Executive Control Section.

4. The SOLIN keyword will skip the automatic link and execute the OSCAR on the specified DBset.

5. The DOMAINSOLVER may be used in conjunction with Solution Sequences 101, 103, 108, and 111 to select domain decomposition solution methods.

6. The following Solution Sequences are currently available in NX Nastran:

Table 3-2. Structured Solution Sequences

SOL Number	SOL Name	Description
101	SESTATIC	Statics with Options: Linear Steady State Heat Transfer Alternate Reduction (See "Superelement Alternate Statics Solution" of the *NX Nastran User's Guide*.) Inertia Relief
103	SEMODES	Normal Modes
105	SEBUCKL	Buckling with options: Static Analysis Alternate Reduction Inertia Relief
106	NLSTATIC	Nonlinear or Linear Statics
107	SEDCEIG	Direct Complex Eigenvalues
108	SEDFREQ	Direct Frequency Response
109	SEDTRAN	Direct Transient Response
110	SEMCEIG	Modal Complex Eigenvalues

Table 3-2. Structured Solution Sequences

SOL Number	SOL Name	Description
111	SEMFREQ	Modal Frequency Response
112	SEMTRAN	Modal Transient Response
114	CYCSTATX	Cyclic Statics with Option: Alternate Reduction
115	CYCMODE	Cyclic Normal Modes
116	CYCBUCKL	Cyclic Buckling
118	CYCFREQ	Cyclic Direct Frequency Response
129	NLTRAN	Nonlinear or Linear Transient Response
144	AESTAT	Static Aeroelastic Response
145	SEFLUTTR	Aerodynamic Flutter
146	SEAERO	Aeroelastic Response
153	NLSCSH	Static Structural and/or Steady State Heat Transfer Analysis with Options: Linear or Nonlinear Analysis
159	NLTCSH	Transient Structural and/or Transient Heat Transfer Analysis with Options: Linear or Nonlinear Analysis
187	RESDDAM	DDAM analysis.
190	DBTRANS	Database Transfer.
200	DESOPT	Design Optimization with option for only Sensitivity Analysis. See Remark 7.
401	NLSTEP	Multistep, structural solution which supports a combination of static (linear or nonlinear) subcases and modal (real eigenvalue) subcases.

7. Design sensitivity is supported with SOL 200 using the DSAPRT case control command. As a result, the following legacy inputs for design sensitivity analysis with SOL 101, 103, and 105 have been undocumented:
   – Case control commands SENSITY, SET2
   – Parameters EIGD, NORM
   – Bulk entries DSCONS, DVAR, DVSET
   Since design sensitivity analysis is automatically done during design optimization, it is not necessary to use the DSAPRT case control unless specific sensitivity information is requested as part of the output, or only design sensitivity analysis is desired.

EXAMPLES:

1. In the following example, SOL 103 is executed from MSCOBJ.

```
SOL 103
```

2. In the following example, the PHASE0 subDMAP is altered, SOL 103 is relinked onto the OBJSCR DBset (which is the default for SOLOUT), and SOL 103 is executed.

```
SOL 103
COMPILE PHASE1
ALTER 'DTIIN'
TABPT SETREE,,,,// $
.
.
.
ENDALTER $
```

3. In the following example, the solution sequence called DYNAMICS is executed from the USROBJ DBset.

```
SOL DYNAMICS SOLIN = USROBJ
```

SOL 601,N or SOL 701

Executes a NX Nastran Advanced Implicit (SOL 601,N) or Explicit (SOL 701) Nonlinear Solution

FORMAT:

SOL 601,N

SOL 701

EXAMPLES:

SOL 601,106

SOL 701

REMARKS:

1. There must be no space in 601,N.

2. You must place the following statement at the beginning of each SOL 601 and 701 Nastran input file:

   ASSIGN OUTPUT2=‘advnlin’.op2,UNIT=21

   The file name ‘advnlin.op2’ is reserved. If there is an existing file with this name in the directory that the job is running, it will be overwritten.

3. Only one running job is allowed in a directory at any time. A SOL 601 (or 701) job may be terminated gracefully at any equilibrium iteration (or time step) by creating a run time option file “tmpadvnlin.rto” in the current working directory with a line “STOP=1” in the file. The following parameters relating to SOL 601 solution convergence may also be modified in the rto file: MAXITE, DTOL, ETOL, RCTOL, RTOL, STOL, RCONSM, RNORM, RMNORM, DNORM, and DMNORM. See NXSTRAT entry for description of these parameters. Only one parameter should be specified on each line in the rto file. For example,
   MAXITE=30
   ETOL=1.0E-6

4. When an analysis is completed, a restart file dbs.res is created where dbs is set by the “dbs” keyword (as for dbs.MASTER and dbs.DBALL files). This file should be preserved to run a restart analysis. To run a restart analysis, dbs.res must

exist and the parameter MODEX in NXSTRAT bulk data entry must be set to 1. The IDs of grids, elements, properties, materials, contact sets, contact pairs and contact regions should be the same in both the original and the restart model.

Note that the default for dbs is the current working directory with the prefix of the current job name.

Note that keyword scratch=no must be used when running a restart analysis.

5. In the nastran command, the keyword parallel may be used to specify the number of processors to use for the solution. The .f06 file will include a message similar to the following indicating the number of processors used by the solution:

```
NUMBER OF PROCESSORS USED.....(NPROC) = n
```

6. The SOL 601 and 701 solution and input data processing uses the **memory** keyword (≥ 256 MB) by default to specify the maximum memory that can be used. You can optionally use the environment variable NXNA_MEMORY to specify the memory used for both the solution and the input data processing. Note that if memory < 256 MB is specified, it will be ignored.

Environment Variable	Description
NXNA_MEMORY	Specifies memory in MB for the both input processing and solution.

Note

When defining NXNA_MEMORY, include the numerical value in MB, but do not include the "MB" text describer.
For example, this is a valid definition:
NXNA_MEMORY=4000
This is an invalid definition:
NXNA_MEMORY=4000MB

7. N represents a valid analysis type. The available types are:

N	SOL 601 Analysis Types
106	Static (Default)
129	Transient
153	Static with steady-state thermal coupling
159	Transient with thermal coupling; see TRANOPT parameter in TMCPARA bulk entry for options

8. You can use the following Case Control commands with SOL 601 and SOL 701. See Remark 12 below for exceptions):

ACCELERATION	EBDSET	SET
ANALYSIS	ELSUM	SHELLTHK
B2GG	FLUX	SPC
BCRESULTS	K2GG	SPCFORCES
BCSET	M2GG	STRESS
BGRESULTS	MPC	SUBCASE
BGSET	GKRESULTS	TEMPERATURE
BOLTLD	GPFORCE	THERMAL
DISPLACEMENT	IC	TITLE
DLOAD	LOAD	TSTEP
DTEMP	OLOAD	VELOCITY

9. SOL 601,153 output is in SORT1 format by default. To get SORT2 format, specify all output requests in SORT2 for both the thermal and structural subcases.

   SOL 601,159 output is in SORT2 format by default. To get SORT1 format, specify all output requests in SORT1 for both the thermal and structural subcases.

10. Only one subcase is used for SOL 601,106; SOL 601,129; and SOL 701. If multiple subcases exist, the first subcase is used. Two subcases (first two subcases) are required for SOL 601,153 and SOL 601,159.

11. Both LOAD and DLOAD can be specified for either static or transient analysis. LOAD selects constant loads while DLOAD selects time-dependent loads. See the *Advanced Nonlinear Theory and Modeling Guide* on defining time-dependent loads.

12. ANALYSIS, B2GG, BGRESULTS, BGSET, BOLTLD, GKRESULTS, GPFORCE, FLUX, K2GG, M2GG and THERMAL are not supported in SOL 701.

13. DISPLACEMENT, VELOCITY, and ACCELERATION should all reference the same set of grid points. If different sets are referenced, the sets are combined and results will be output for the combined set.

14. For shell nodes which have 5 degrees of freedom, rotation results are not output for displacements, velocities, and accelerations.

15. You can use the following Bulk Data Entries with SOL 601 (all) and SOL 701 (except those marked as “not 701”):

BCPROP	CPLSTS3 (not 701)	MATHEV (not 701)	PLOADE1 (not 701)
BCPROPS	CPLSTS4 (not 701)	MATHP	PLOADX1 (not 701)
BCRPARA	CPLSTS6 (not 701)	MATPLCY	PLPLANE (not 701)

BCTPARA
BCTADD
BCTSET
BDYOR (not 701)
BEDGE (not 701)
BFLUID (not 701)
BGSET (not 701)
BLSEG (not 701)
BOLT (not 701)
BOLTFOR (not 701)
BOLTLD (not 701)
BSURF
BSURFS
CBAR
CBEAM
CBUSH (not 701)
CBUSH1D
CDAMP1
CDAMP2
CELAS1
CELAS2
CHBDYE (not 701)
CHBDYG (not 701)
CGAP
CHEXA
CMASS1
CMASS2
CONM1
CONM2
CONROD
CONV (not 701)
CORD1C
CORD1R
CORD1S
CORD2C
CORD2R
CORD2S
CPLSTS8 (not 701)
CPYRAM
CQUAD (not 701)
CQUAD4
CQUAD8 (not 701)
CQUADR (not 701)
CQUADX4 (not 701)
CQUADX8 (not 701)
CREEP (not 701)
CROD
CTETRA
CTRAX3 (not 701)
CTRAX6 (not 701)
CTRIA3
CTRIA6 (not 701)
CTRIAR (not 701)
DLOAD
DMIG (not 701)
DTEMP
EBDSET
EBDADD
FORCE
FORCE1
FORCE2
GRAV
GRID
GROUP
LOAD
MAT1
MAT2
MAT3 (not 701)
MAT4 (not 701)
MAT5 (not 701)
MAT8
MAT9
MAT10 (not 701)
MAT11
MATS1
MATSMA (not 701)
MATSR
MATT1
MATT2
MATT3 (not 701)
MATT4 (not 701)
MATT5 (not 701)
MATT8
MATT9
MATT11
MATTC (not 701)
MATVE (not 701)
MOMENT
MOMENT1
MOMENT2
MPC
MPCADD
NXSTRAT
PARAM
PBAR
PBARL
PBCOMP
PBEAM
PBEAML
PBUSH (not 701)
PBUSH1D
PBUSHT (not 701)
PCOMP (not 701)
PCOMPG (not 701)
PCONV (not 701)
PDAMP
PELAS
PELAST
PGAP
PLCYISO
PLCYKIN
PLSOLID
PMASS
PPLANE (not 701)
PROD
PSHL3D
PSHELL
PSOLID
QHBDY (not 701)
QBDY1 (not 701)
QBDY2 (not 701)
QBDY3 (not 701)
QVOL (not 701)
RADBC (not 701)
RADM (not 701)
RADMT (not 701)
RBAR
RBE2
RBE3
RFORCE
RFORCE1
SPC
SPC1
SPCADD
SPCD
TABLED1
TABLED2
TABLEM1
TABLES1
TABLEST
TABVE (not 701)
TEMP
TEMPBC (not 701)
TEMPD
TIC
TLOAD1
TMCPARA (not 701)
TSTEP

CPENTA	MATCID	PLCYRUP
CPLSTN3 (not 701)	MATCRP (not 701)	PLOAD
CPLSTN4 (not 701)	MATG (not 701)	PLOAD1
CPLSTN6 (not 701)	MATHE	PLOAD2
CPLSTN8 (not 701)	MATHEM (not 701)	PLOAD4

16. BCTPARA, EBDSET , EBDADD , MATHE and NXSTRAT are supported only by SOLs 601 and 701. MATHEM, MATHEV, MATVE, MATSMA, MATTC , MATG, TABVE and TMCPARA are supported only by SOL 601.

17. Not all parameters in each bulk data entry are used. Restrictions are given in the description for the entry. Some restrictions are highlighted in Remark 19.

18. For PARAM entry, parameters POST, LGDISP, LGSTRN and ALPHA1 are supported by SOLs 601 and 701. In addition parameters ALPHA2, CB2, CK2, CM2, TABS and SIGMA are supported by SOL 601.

19. PARAM,POST,-2 must be specified to generate both geometry and results data blocks in the op2 file. PARAM,POST,-1 must be specified to generate only results data blocks. Otherwise, no op2 file is generated.

20. Some important restrictions in SOL 601 and 701 are highlighted below:

    - When there is element birth/death (either through EBDSET or automatic element rupture) or preload bolt, SORT2 output is not supported.

    - Structural damping is not supported for individual elements. Hence, GE field in the supported Bulk Data entries is not used. Rayleigh damping is supported and specified with PARAM ALPHA1 and ALPHA2. Only ALPHA1 is supported for SOL 701.

    - Nonstructural mass is not supported.

    - CROD and CONROD are simulated by rod elements with no torsional stiffness.

    - Several restrictions apply to beam elements, e.g. beam offsets, tapered beams, shear center. Please see remarks in the CBAR, CBEAM, PBAR, PBARL, PBEAM, PBEAML, and PBCOMP entries.

    - Only relevant fields in the MAT2 and MAT9 entries are used to define orthotropic materials instead of anisotropic materials.

    - In the PSHELL entry, only MID1 is used to define material properties of shell elements.

- Restrictions apply to the use of follower force/moment in FORCE1, FORCE2, MOMENT1, and MOMENT2 entries.
- Transverse properties for gap elements (PGAP entry) are not supported.
- Only diagonal terms in the mass matrix defined by CONM1 and CONM2 are used.
- Several restrictions apply to MATHP, CREEP, and PLOAD1 entries. Other restrictions are mentioned in the remarks for the entry.

21. For more information on SOL 601 and 701, please refer to the *Advanced Nonlinear Theory and Modeling Guide*.

TIME

Sets the maximum elapsed and I/O time.

FORMAT:

TIME[=]t1[,t2]]

DESCRIBERS:

Describer	Meaning
t1	Maximum allowable execution time in elapsed minutes. (Real or Integer; 0 < t1 < 3.15E7 minutes; Default = 3.15E7 minutes). Values entered with an exponent are interpreted as Real and must include a decimal (for example, 1.E2).
t2	Maximum allowable I/O limit in minutes. (Real or Integer > 0; Default is infinity, which is machine dependent.)

REMARKS:

1. The TIME statement is optional.
2. If t2 is specified then t1 must be specified.

EXAMPLES:

1. The following example designates a runtime of 8 hours:

```
TIME 480
```

2. The following example designates 90 seconds:

```
TIME 1.5
```

Chapter 4: Case Control Commands

4
CASE

4.1 Key to Descriptions

Brackets [ ] indicate that a choice of describers is optional.

If the describers are stacked vertically, then only one may be specified.

A brief description of the command is given.

OLOAD Applied Load Output Request

Requests the form and type of applied load vector output.

Describers in uppercase letters are keywords that must be specified as shown.

Format:

$$\text{OLOAD}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right], \text{PRINT}, \text{PUNCH}, \left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Describers in lower case are variables.

The default describers are shaded.

Examples:

OLOAD=ALL

OLOAD(SORT1,PHASE)=5

A typical example is given.

Braces { } indicate that a choice of describers is mandatory.

Describers	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be
	le will be
	ctangular fo
	REAL or I
	Requests polar format (
	Phase output is in degre
ALL	Applied loads for all po
NONE	Applied load for no points will be output.
n	Set identification of a previously appearing SET command. Only loads on points whose identification numbers appear on this SET command will be output. (Integer > 0).

Each of the describers is discussed briefly. Further details may be discussed under Remarks.

If items inside the parentheses are specified, the parentheses must also be specified in the input.

If the describer is in lower case, then it is a variable and the describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

Remarks:

1. Both PRINT and PUNCH
2. See the *NX Nastran User's* SORT2 formats and their c will be output.
3.. In a statics problem, a reque ro and nonzero) to be output.

The remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command, the command's relationship to other commands, restrictions and recommendations on its use, and further details regarding the describers.

The Case Control Section

The Case Control Section has several basic functions; specifically, it:

- Selects loads and constraints.
- Requests printing, plotting, and/or punching of input and output data (see “Plotting” in the *NX Nastran User’s Guide*.)
- Defines the subcase structure for the analysis.

The applicability tables at the end of this section indicate the applicability of each command in all solution sequences.

4.2 Format of Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other commands. Each command is described as follows:

Description

A brief sentence about the function of the command is stated.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the command line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
SET 1=5, 6, 7, 8, 9,
        10 THRU 55
```

Examples

One or more examples are given.

Describers

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The Remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command; the command's relationship to other commands, restrictions and recommendations on its use; and further descriptions of the describers.

4.3 Case Control Command Summary

This section contains a summary of all Case Control commands under the following headings:

Subcase Definition

1. Output Request Delimiters

OUTPUT	Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.
OUTPUT(PLOT)	Beginning of structure plotter output request.
OUTPUT(POST) or SETS DEFINITION	Beginning of grid point stress output requests.
OUTPUT(XYOUT) or OUTPUT(XYPLOT)	Beginning of curve plotter output request.
OUTPUT(CARDS)	Suppresses processing of Bulk Data entries (MSGMESH).

2. Subcase Delimiters

REPCASE	Delimits and identifies a repeated output request subcase.
SUBCASE	Delimits and identifies a subcase.
SUBCOM	Delimits and identifies a combination subcase.

SYM	Delimits and identifies a symmetry subcase.
SYMCOM	Delimits and identifies a symmetry combination subcase.

3. Subcase Control

MASTER	Allows the redefinition of a MASTER subcase.
MODES	Repeats a subcase.
SUBSEQ	Gives the coefficients for forming a linear combination of the previous subcases.
SYMSEQ	Gives the coefficients for combining the symmetry subcases into the total structure.

Data Selection

1. Static Load Selection

BOLTLD	Selects either a BOLTLD bulk entry or BOLTFOR bulk entries for bolt preload processing.
CLOAD	Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.
DEFORM	Selects the element deformation set.
EFLOAD	Converts surface loads from an external field solution to structural loads.
LOAD	Selects an external static loading set.

2. Dynamic Load Selection

DLOAD	Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.
LOADSET	Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.
NONLINEAR	Selects nonlinear dynamic load set for transient problems.

3. Constraint Selection

BC	Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.
BCSET	Selects the contact set for SOLs 101, 103, 111, 112, 601, and 701.
BGSET	Selects the glue contact set.
DSYM	Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.
MPC	Selects a multipoint constraint set.
RIGID	Selects the rigid element processing method for RBAR, RBE1, RBE2, RBE3, RROD, and RTRPLT elements.
SPC	Selects a single-point constraint set to be applied.
STATSUB	Selects the static solution to use in forming the differential stiffness for buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.
SUPORT1	Selects the fictitious support set (SUPORT1 entries only) to be applied to the model.

4. Fluid-Structure Interaction

FLSFSEL	Selects frequency content for controlling fluid-structure interaction.
FLSTCNT	Specifies miscellaneous control parameters for fluid-structure interaction.
MFLUID	Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

5. Thermal Field Selection

TEMPERATURE	Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

TEMPERATURE(INITIAL)	Selects initial temperature distribution for temperature-dependent material properties and heat transfer problems.
TEMPERATURE(LOAD)	Selects temperature set for static thermal load.
TEMPERATURE(MATERIAL)	Selects temperature set for temperature-dependent material properties.
TSTRU	Defines a temperature set ID for a structural run based on a heat transfer subcase.

6. Static Solution Conditions

SMETHOD	Selects iterative solver parameters.

7. Dynamic Solution Conditions

CMETHOD	Selects complex eigenvalue extraction parameters.
FREQUENCY	Selects the set of forcing frequencies to be solved in frequency response problems.
IC	Selects the initial conditions for direct transient analysis (SOLs 109, 129, and 159).
METHOD	Selects the real eigenvalue extraction parameters.
MODSEL	Selects mode numbers to include in a modal dynamic response solution.
NSM	Selects non-structural Mass (NSM) set for mass generation.
RANDOM	Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.
RESVEC	Controls the computation of residual vectors.
RMETHOD	Selects the ROTORD bulk data entry used for rotor dynamics analysis.
RSMETHOD	Selects the real eigenvalue extraction parameters in the component mode reduction of the residual structure for direct and modal solutions.

SDAMPING	Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.
SMETHOD	Selects iterative solver override options in frequency response analysis.
TSTEP	Selects integration and output time steps for linear or nonlinear transient analysis.

8. Direct Input Matrix Selection

A2GG	Selects direct input acoustic/fluid-structure coupling matrices.
B2GG	Selects direct input damping matrices.
B2PP	Selects direct input damping matrices.
K2GG	Selects direct input stiffness matrices.
K2PP	Selects direct input stiffness matrices, which are not included in normal modes.
M2GG	Selects direct input mass matrices.
M2PP	Selects direct input mass matrices, which are not included in normal modes.
TFL	Selects the transfer function set(s) to be added to the direct input matrices.
P2G	Selects direct input load matrices.

9. Nonlinear Analysis

EBDSET	Selects the element birth/death set for SOLs 601 and 701.
NLPARM	Selects the parameters used for nonlinear static analysis.
SMETHOD	Selects iterative solver override parameters in nonlinear static analysis.
TSTEP	Selects integration and output time steps for linear or nonlinear transient analysis.

TSTEPNL	Transient time step set Selection for nonlinear analysis.

10. Aerodynamic Analysis

AECONFIG	Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.
AESYMXY	Aerodynamic XY plane of symmetry flag.
AESYMXZ	Aerodynamic XZ plane of symmetry flag.
AEUXREF	Defines the Reference Aerodynamic Extra Point (Controller) Vector.
CSSCHD	Aerodynamic Control Surface Schedule.
DIVERG	Selects the divergence parameters in a static aeroelastic divergence problem.
FMETHOD	Selects the parameters to be used by the aerodynamic flutter analysis.
GUST	Selects the gust field in an aerodynamic response problem.
TRIM	Selects trim variable constraints in static aeroelastic response.

11. Design Optimization and Sensitivity (SOL 200)

ANALYSIS	Specifies the type of analysis being performed for the current subcase.
AUXCASE	Delimits Case Control commands for an auxiliary model in SOL 200.
AUXMODEL	References an auxiliary model for generation of boundary shapes in shape optimization.
BEGIN BULK	Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.
DESGLB	Selects the design constraints to be applied at the global level in a design optimization task.

DESOBJ	Selects the DRESP1, DRESP2, or DRESP3 entry to be used as the design objective.
DESSUB	Selects the design constraints to be used in a design optimization task for the current subcase.
DSAPRT	Specifies design sensitivity output parameters.
DRSPAN	Assigns a set of DRESP1 responses to a specific subcase.
MODTRAK	Selects mode tracking options in design optimization (SOL 200).

Output Selection

1. Output Control

ECHO	Controls echo (i.e., printout) of the Bulk Data.
FLSPOUT	Controls modal participation output for the fluid-structure interaction.
LABEL	Defines a character string that will appear on the third heading line of each page of the printer output.
LINE	Defines the maximum number of output lines per printed page.
MAXLINES	Sets the maximum number of output lines.
PAGE	Causes a page eject in the echo of the Case Control Section.
PLOTID	Defines a character string that will appear on the first frame of any plotter output.
SKIP	Activates or deactivates the execution of subsequent commands in the Case Control (including plot commands).
SUBTITLE	Defines a subtitle that will appear on the second heading line of each page of the printer output.
TITLE	Defines a character string that will appear on the first heading line of each page of the NX Nastran printer output.

2. Set Definition

MAXMIN	Specifies options for max/min surveys of certain output data associated with grid points.
OFREQUENCY	Selects a set of frequencies for output requests.
OMODES	Selects a set of modes for output requests.
OTIME	Selects a set of times for output requests.
PARTN	Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17). In SOLs 111 and 200, the PARTN command specifies the points at which modal participation factors are to be computed.
SET	Defines a set of element or grid point numbers to be plotted.
SETMC	Defines sets for modal contribution results.
SURFACE	Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.
VOLUME	Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

3. Physical Set Output Requests

ACCELERATION	Requests the form and type of acceleration vector output.
BCRESULTS	Requests contact result output.
BGRESULTS	Requests glue result output.
BOUTPUT	Selects contact regions for output.
CKGAP	Requests gap output for chocking elements. Valid for SOL 401 only.
CZRESULTS	Requests results output for cohesive elements. Valid for SOL 401 only.
DISPLACEMENT, VECTOR, or PRESSURE	Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.
EDE	Requests the output of the energy loss per cycle in selected elements.

EKE	Requests the output of the kinetic energy in selected elements.
ELSDCON	Requests mesh stress discontinuities based on element stresses (see STRESS).
ENTHALPY	Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).
ERP	Requests the output of equivalent radiated power for selected panels and the elements comprising the selected panels.
ESE	Requests the output of the strain energy in selected elements.
FORCE	Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis.
FLUX	Requests the form and type of gradient and flux output in heat transfer analysis.
GKRESULTS	Requests the form and type of gasket result output in SOL 601.
GPFORCE	Requests grid point force balance at selected grid points.
GPKE	Requests the output of the kinetic energy at selected grid points in normal modes analysis only.
GPRSORT	Requests that the output of composites ply results on elements referencing a PCOMPG be sorted by global ply ID, then by element ID.
GPSDCON	Requests mesh stress discontinuities based on grid point stresses (see GPSTRESS).
GPSTRAIN	Requests grid points strains for printing only.
GPSTRESS	Requests grid point stresses for printing only.
HDOT	Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).
JINTEG	Request computation of the j-integral.

MEFFMASS	Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.
MODALE	Requests the type of energy to output for SOL 111.
MODCON	Requests modal contribution results for the residual.
MPCFORCES	Requests the form and type of multipoint force of constraint vector output.
NLSTRESS	Requests the form and type of nonlinear element stress output in SOL 106.
NOUTPUT	Requests physical output in cyclic symmetry problems.
OLOAD	Requests the form and type of applied load vector output.
PANCON	Requests acoustic panel contribution results for residual.
PFRESULTS	Requests progressive failure results output for composite solid elements. Valid for SOL 401 only.
RCROSS	Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.
RMAXMIN	Defines parameters to output the minimum, maximum, absolute value maximum, average, and RMS value of stress, force, and displacement results for SOLs 101, 109, and 112.
SHELLTHK	Requests the form of shell thickness output for SOLs 601 and 701.
SPCFORCES	Requests the form and type of single-point force of constraint vector output.
STRAIN	Requests the form and type of strain output.
STRESS	Requests the form and type of element stress output.
STRFIELD	Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.
SVECTOR	Requests the form and type of solution set eigenvector output.

THERMAL	Requests the form and type of temperature output.
VELOCITY	Requests the form and type of velocity vector output.

4. Solution Set Output Requests

AEROF	Requests the aerodynamic loads on aerodynamic control points.
APRESSURE	Requests the aerodynamic pressures in static aeroelastic response.
HARMONICS	Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.
HOUTPUT	Requests harmonic output in cyclic symmetry problems.
MPRES	Requests the pressure for selected surface elements in fluid-structure interaction problems.
NLLOAD	Requests the form and type of nonlinear load output for transient problems.
SACCELERATION	Requests the form and type of solution set acceleration output.
SDISPLACEMENT	Requests the form and type of solution set displacement output.
SVELOCITY	Requests the form and type of solution set velocity output.

5. Model Checkout

ADAPTERR	Requests stress norm and strain energy norm error estimates.
ELSUM	Requests a summary of element properties for output.
GROUNDCHECK	Performs grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

WEIGHTCHECK	At each stage of the mass matrix reduction, computes rigid body mass and compares it with the rigid body mass of the g-set.

Superelement Control

EXTSEOUT	Specifies creation of an external superelement.
SEALL	Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.
SEDR	Specifies the superelement identification numbers for which data recovery will be performed.
SEDV	Specifies the superelement identification numbers for which the design variables will be processed.
SEEXCLUDE	Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.
SEFINAL	Specifies the superelement identification number of the final superelement to be assembled.
SEKREDUCE	Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.
SELGENERATE	Specifies the superelement identification numbers for which static loads will be generated.
SELREDUCE	Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.
SEMGENERATE	Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.
SEMREDUCE	Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices are assembled and reduced.
SERESP	Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.
SUPER	Assigns a subcase(s) to a superelement or set of superelements.

Multi-Body Dynamics Interface Control

ADAMSMNF	Generates Adams interface Modal Neutral File (MNF) during a SOL 103, 111, or 112 run.
ADMRECVR	Recovers stress results from an Adams flexbody analysis.
MBDEXPORT	Generates interface file for multi-body dynamics and control system software during a SOL 103, 111, or 112 run.
MBDRECVR	Recovers stress results from multi-body dynamics analysis.

Miscellaneous

$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
INCLUDE	Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.
OUTPUT	Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.
PARAM	Specifies values for parameters.

Case Control Command Descriptions

$

Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

FORMAT:

$ followed by any characters out to column 80.

EXAMPLES:

```
$ TEST FIXTURE-THIRD MODE
```

REMARKS:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

A2GG

Direct Input Acoustic/Fluid-Structure Matrix Selection

Selects direct input acoustic/fluid-structure coupling matrix or matrices.

FORMAT:

A2GG=name

EXAMPLES:

```
A2GG=ADMIG
A2GG=ADMIG1, ADMIG2, ADMIG3
A2GG=1.25*ADMIG1, 1.0*ADMIG2, 0.75*ADMIG3
SET 100=A1, A2
A2GG=100
```

DESCRIBERS:

Describer	Meaning
name	Name of direct input acoustic/fluid-structure coupling matrix [A^2_{gg}] defined using DMIG bulk entries. (Character) Scale factors may be included (see Remarks 4 and 5).

REMARKS:

1. DMIG matrices are not used unless selected using A2GG.
2. If PARAM,ASCOUP,YES, the direct input acoustic/fluid-structure coupling matrix selected using A2GG is added to the computed acoustic/fluid-structure coupling matrix. If PARAM,ASCOUP,NO, the direct input acoustic/fluid-structure coupling matrix selected using A2GG replaces the computed acoustic/fluid-structure coupling matrix.

3. The matrix must be square and field 4 of the DMIG bulk entry must contain the integer 1. When filling out the DMIG bulk entries, the GJ column index corresponds to fluid points, CJ is zero, the Gi row index corresponds to structural points, Ci corresponds to DOF, and Ai are the area values.

4. The associated DMIG matrices can be scaled using either in-line scale factors entered on A2GG (for example, A2GG=1.25*ADMIG1), or the parameter CA2 (for example, PARAM,CA2,1.25), or both. For information regarding the CA2 parameter, see “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. 1.0 should be entered for matrices in the list that are not scaled. For example, if A2GG=1.25*ADMIG1,1.0*ADMIG2,0.75*ADMIG3 is specified, the result is A2GG=1.25*ADMIG1 + ADMIG2 + 0.75*ADMIG3.

   Specifying the CA2 parameter scales all the A2GG. For example, if both PARAM,CA2,1.30 and A2GG=1.25*ADMIG1,1.0*ADMIG2,0.75*ADMIG3 are specified, the result is A2GG=1.30(1.25*ADMIG1 + ADMIG2 + 0.75*ADMIG3).

6. A2GG is supported in dynamic solutions with acoustic/fluid-structure coupling.

7. Only one A2GG case control command should be used in an input file and it should appear above any subcases.

ACCELERATION

Acceleration Output Request

Requests form and type of acceleration vector output.

FORMAT:

$$\text{ACCELERATION}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{ABS}\\ \text{REL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix},\begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
ACCELERATION=5
ACCELERATION(SORT2, PHASE)=ALL
ACCELERATION(SORT1, PRINT, PUNCH, PHASE)=17
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.

Describer	Meaning
PLOT	Computes, but does not print or punch, acceleration output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ABS	For enforced motion dynamic analysis, acceleration results will be output as absolute acceleration.
REL	For enforced motion dynamic analysis, acceleration results will be output relative to the enforced motion input.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 7.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 7.

Describer	Meaning
RPUNCH	Writes random analysis results to the punch file. See Remark 7.
ALL	Accelerations at all points will be output.
NONE	Accelerations at no points will be output.
n	Set identification of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. Acceleration output is only available for transient and frequency response problems.
3. The defaults for SORT1 and SORT2 depend on the type of analysis:
   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.
   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.
   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.
4. ACCELERATION = NONE allows overriding an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer request is present for magnitude/phase representation.

6. Acceleration results are printed and/or punched in the global coordinate system (see field CD on the GRID bulk data entry). The coordinate system for plotted acceleration output depends on the PARAM,POST setting. See the parameter POST.

7. The following applies to random solutions:

   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

   - Autocorrelation (ATOC) calculations require the RANDT1 bulk entry.

8. When doing enforced motion dynamic analysis and relative output is requested (using the REL describer), the output will be relative to the input as described by the equation:

   $$u_f = y_f - K_{ff}^{-1} K_{fs} u_s$$

   where u_f = absolute displacement

   y_f = relative displacement

   u_s = enforced motion.

REMARKS RELATED TO SOLS 601 AND 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC, RMS, and RALL are ignored.

2. Displacements, velocities and accelerations must be output for the same set of grid points if requested. Output requested for set n in this command will be combined with the sets requested in the VELOCITY and DISPLACEMENT commands, and accelerations will be output at the grid points of the combined set.

ACINTENSITY

Acoustic Intensity Output

Acoustic intensity output request at microphone points (SORT1 format only).

FORMAT:

$$ACINTENSITY\left[\left[\frac{PRINT}{NOPRINT},PUNCH\right]\left[\frac{REAL\,or\,IMAG}{PHASE}\right]\right]=\begin{bmatrix}n\\ ALL\\ NONE\end{bmatrix}$$

EXAMPLES:

```
SET 123=5,10,15,25
ACINTENSITY=123
SET 5=6,8,11
ACINTENSITY(PRINT,PUNCH,PHASE)=5
```

DESCRIBERS:

Describer	Meaning
PRINT	The print file (.f06) will be the output medium. (Character; Default = PRINT)
PUNCH	The standard punch file (.pch) will be the output medium.
NOPRINT	Generates, but does not print intensity results.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Character; Default = REAL or IMAG)

Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
n	ID of SET case control command containing the list of identification numbers of fluid grids referenced by microphone elements at which to calculate acoustic intensity results. (Integer > 0)
ALL	Requests acoustic intensity results for all microphone elements in the model.
NONE	Turns this output request off; useful for controlling output requests across subcases.

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. Acoustic intensity results will be written to the op2 file, if the value of PARAM, POST is -1 or -2.

ACPOWER

Acoustic Power Output

Acoustic power output request for the AMLREGs or 2D microphone elements (SORT2 results only).

FORMAT:

$$ACPOWER\left[\begin{array}{l} AMLREG=\left\{\begin{array}{c} ALL \\ \hline NONE \\ na \end{array}\right\}, GROUP=ng, \left[\begin{array}{c} PRINT \\ \hline NOPRINT \end{array}, PUNCH\right], \\ \left[\begin{array}{c} REAL\ or\ IMAG \\ \hline PHASE \end{array}\right] \\ \left\{\begin{array}{c} YES \\ \hline NO \end{array}\right\} \end{array}\right] = \begin{array}{c} 4 \\ \text{CASE} \end{array}$$

EXAMPLES:

```
SET 123=3,6,9
ACPOWER(AMLREG)=123
SET 5=5,10
ACPOWER(GROUP=5,PRINT,PUNCH,PHASE)
```

DESCRIBERS:

Describer	Meaning
ALL	Acoustic power for all AMLREGs. (Character; Default = ALL)
NONE	No acoustic power for any AMLREGs.

Describer	Meaning
na	ID of SET case control command containing the list of set identification numbers of AMLREG bulk entries at which to calculate results for the radiating surface. (Integer>0)
ng	ID of SET case control command containing the list of set identification numbers of GROUP bulk entries at which to calculate results for the radiating surface. (Integer > 0) See Remark 2.
PRINT	The print file (.f06) will be the output medium. (Character; Default = PRINT)
PUNCH	The standard punch file (.pch) will be the output medium.
NOPRINT	Generates, but does not print, acoustic power results.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Character; Default = REAL or IMAG)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
YES	Enables acoustic power output. (Character; Default = YES)
NO	Disables acoustic power output.

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. Only 2D microphone elements referenced in a GROUP bulk entry will be processed for acoustic power results. These elements must reference a PMIC property. Any other entity contained in a GROUP referenced by this case control command will be ignored.
3. Acoustic power results will be written to the op2 file, if the value of PARAM, POST is -1 or -2.

ACVELOCITY

Acoustic Velocity Output

Acoustic velocity output request at microphone points (SORT1 format only).

FORMAT:

$$ACVELOCITY\left[\left[\frac{PRINT}{NOPRINT},PUNCH\right]\left[\frac{REAL\ or\ IMAG}{PHASE}\right]\right]=\begin{bmatrix} n \\ ALL \\ NONE \end{bmatrix}$$

EXAMPLES:

```
SET 123=5,10,15,25
ACVELOCITY=123
SET 5=6,8,11
ACVELOCITY(PRINT,PUNCH,PHASE)=5
```

DESCRIBERS:

Describer	Meaning
PRINT	The print file (.f06) will be the output medium. (Character; Default = PRINT)
PUNCH	The standard punch file (.pch) will be the output medium.
NOPRINT	Generates, but does not print, acoustic velocity.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Character; Default = REAL or IMAG)

Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
n	ID of SET case control command containing the list of identification numbers of fluid grids referenced by microphone elements at which to calculate acoustic velocity results. (Integer > 0)
ALL	Request acoustic velocity results for all microphone elements in the model.
NONE	Turns this output request off; useful for controlling output requests across subcases.

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. Output will be in SORT1 format.
3. Acoustic velocity results will be written to the op2 file, if the value of PARAM, POST is -1 or -2.

ADAMSMNF

Generates Adams Interface Modal Neutral File

Generates Adams Interface Modal Neutral File (MNF) during SOL 103, 111, or 112.

FORMAT:

$$\text{ADAMSMNF}\left[\text{FLEXBODY}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{FLEXONLY}=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}\right],$$

$$\left[\text{OUTGSTRS}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{OUTGSTRN}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],$$

$$\left[\text{MINVAR}=\left\{\begin{matrix}\text{PARTIAL}\\ \text{CONSTANT}\\ \text{FULL}\\ \text{NONE}\end{matrix}\right\}\right],\left[\text{PSETID}=\left\{\begin{matrix}\text{NONE}\\ \text{setid}\\ \text{ALL}\\ \text{sktunit}\end{matrix}\right\}\right],$$

$$\left[\text{ADMOUT}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{CHECK}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{NONCUP}=\left\{\begin{matrix}\text{-1}\\ \text{-2}\end{matrix}\right\}\right]$$

EXAMPLES:

```
ADAMSMNF FLEXBODY=YES
```

DESCRIBERS:

Describer	Meaning
FLEXBODY	Requests the generation of MNF.
NO	Standard NX Nastran solution without MNF creation (default).
YES	MNF generation requested.

Describer	Meaning
FLEXONLY	Determines if standard DMAP solution runs or not after MNF creation is complete.
YES	Only MNF creation occurs (default).
NO	MNF file creation occurs along with standard DMAP solution.
OUTGSTRS	Determines if grid point stress is written to MNF.
NO	Do not write grid point stress to MNF (default).
YES	Write grid point stress to MNF.
OUTGSTRN	Determines if grid point strain is written to MNF.
NO	Do not write grid point strain to MNF (default).
YES	Write grid point strain to MNF.
MINVAR	Determines how mass invariants are computed.
PARTIAL	Mass invariants 5 and 9 are not computed.
CONSTANT	Mass invariants 1,2,6 and 7 are computed.
FULL	All nine mass invariants are computed.
NONE	No mass invariants are computed.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) section (including PLOTEL) or on a sketch file whose connectivity is exported to face geometry to be used in Adams. See Remark 16.
NONE	All grids, geometry and associated modal data is written to MNF (default).
setid	The connectivity of a specific element set is used to export face geometry.
ALL	The connectivity of all element sets are used to export face geometry.

Describer	Meaning
sktunit	The connectivity of element faces defined on a sketch file is used to export face geometry. Note that the value must be a negative number to distinguish it from a setid value.
ADMOUT	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of Adams/Flex results.
NO	OP2 file will not be generated (default).
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when ADMOUT=YES. See Remark 20.
NO	No debug output will be written (default).
YES	Debug output will be written.
NONCUP	Modal damping output control. See Remark 22.
–1	Output the full equivalent modal viscous damping matrix (default).
–2	Output only diagonal values of the equivalent modal viscous damping matrix.

REMARKS:

1. The creation of the Adams MNF, which is applicable in a non-restart SOL 103, 111, or 112 analysis only, is initiated by ADAMSMNF FLEXBODY=YES (other describers are optional) along with the inclusion of the bulk data entry DTI,UNITS. The MNF file naming convention is as follows: ‘jid_seid.mnf’, where seid is the integer number of the superelement (0 for residual-only run). The location of these files is the same directory as the jid.f06 file.

2. ADAMSMNF must appear above the subcase level.

3. Since Adams is not a unitless code, the Data Table Input bulk entry DTI,UNITS is required for an MBDEXPORT ADAMS FLEXBODY=YES run. The DTI,UNITS entry specifies the system of units of the original NX Nastran input file, and is then included with the data written to the MNF file. NX Nastran does not do a units

conversion of the nastran data when writing the MNF file. Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

All entries are required. Acceptable character strings are listed below.

Mass:

KG - kilogram

LBM – pound-mass (0.45359237 kg)

SLUG – slug (14.5939029372 kg)

GRAM – gram (1E-3 kg)

OZM – ounce-mass (0.02834952 kg)

KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

MGG – megagram (1E3 kg)

MG – milligram (1E-6 kg)

MCG – microgram (1E-9 kg)

NG – nanogram (1E-12 kg)

UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

4. Since DTI,UNITS determines all units for the MNF, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the MNF. For example, if the model mass is kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1 ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the MNF would be: “DTI,UNITS,1,KG,N,M,S”.

5. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points; or for a residual-only type model, you can use standard NX Nastran ASET Bulk Data entries to define the attachment points.

6. The nine mass variants are:

$$\underset{1 \times 1}{{}^{1}I} = \sum_{p=1}^{N} m_p$$

$$\underset{3 \times 1}{{}^{2}I} = \sum_{p=1}^{N} m_p s_p$$

$$\underset{3 \times M}{{}^{3}I_j} = \sum_{p=1}^{N} m_p \Phi_p \quad j = 1, \ldots, M$$

$$\underset{3 \times M}{{}^{4}I} = \sum_{p=1}^{N} m_p \tilde{s}_p \bar{\Phi}_p + I_p \Phi_p^*$$

$$\underset{3 \times M}{{}^{5}I_j} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1, \ldots, M$$

$$\underset{M \times M}{{}^{6}I} = \sum_{p=1}^{N} m_p \Phi_p^T \Phi_p + \Phi_p^{*T} I_p \Phi_p^*$$

$$\underset{3 \times 3}{{}^{7}I} = \sum_{p=1}^{N} m_p \tilde{s}_p^T \tilde{s}_p + I_p$$

$$\underset{3 \times 3}{{}^{8}I_j} = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj} \quad j = 1, \ldots, M$$

$$\underset{3 \times 3}{{}^{9}I_{jk}} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \phi_{pk} \quad j, k = 1, \ldots, M$$

$s_p = [xyz]^T$ are the coordinates of grid point p in the basic coordinate system.

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix} = \text{skew symmetric vector cross product operator.}$$

φ_p = partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid *p*.

I_p = inertia tensor *p*.

φ_p^* = partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid *p*.

$\tilde{\phi}_{pf}$ = skew-symmetric matrix formed for each grid translational degree of freedom for each mode.

M=number of modes.

N=number of grids.

7. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTS (ns) should be at least ns = n+(6+p), assuming that residual flexibility is on. In the above equation for ns, the number of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL Bulk Data entries; the number of load cases is p. In general, you can't have too many SPOINTs, as excess ones will simply be truncated with no performance penalty.

8. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in a more accurate representation of the component shapes at little additional cost.

9. OMIT or OMIT1 Bulk Data entries are not supported.

10. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.

11. CBEND elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

12. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an MNF. It must have consistent units with regard to the DTI,UNITS Bulk Data entry. Before generating the MNF, NX Nastran will

appropriately scale the WTMASS from the physical mass matrix and mode shapes.

13. There is a distinction between how an ADAMSMNF FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

14. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran, the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with ADAMSMNF FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This caution only applies to a superelement run and not to a residual-only or a part superelement run.

15. OUTGSTRS and OUTGSTRN entries require the use of standard NX Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN= commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain print to the NX Nastran .f06 file.

16. To reduce the FE mesh detail for dynamic simulations, PSETID (on the ADAMSMNF Case Control command) defined with a SET entry (i.e. setid) is used to define a set of PLOTELs or other elements used to select grids to display the components in Adams. This option can significantly reduce the size of the MNF without compromising accuracy in the Adams simulation providing that the mass invariant computation is requested. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids attached only to a-set or exterior grids), a SEELT Bulk Data entry must be specified to keep that display element with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

    If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements used to select grids to display the

component in Adams. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements' sets, the union of which will be used to define a set of PLOTELs or other elements used to select grids to display the component in Adams. If the user wishes to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

The following element types are not supported for writing to an MNF, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CMASS3, CMASS4, CRAC2D, CRAC3D, CTWIST, CWEDGE, CWELD, and GENEL.

PSETID can also point to a sketch file using PSETID=-sktunit, where sktunit references an ASSIGN statement of the form

```
ASSIGN SKT='sketch_file.dat',UNIT=sktunit.
```


The grids defined for the elements' faces in the sketch file, along with *all* external (i.e. boundary) grids for the superelements, will be the only grids (and their associated data) written to the MNF.

The format of the sketch file, which describes the mesh as a collection of faces, must be as follows:

```
face_count
face_1_node_count face_1_nodeid_1 face_1_nodeid_2 ...
face_2_node_count face_2_nodeid_1 face_2_nodeid_2 ...

<etc>
```

Faces must have a node count of at least two. For example, a mesh comprised of a single brick element might be described as follows:

```
6
4 1000 1001 1002 1003
4 1007 1006 1005 1004
4 1000 1004 1005 1001
4 1001 1005 1006 1002
4 1002 1006 1007 1003
4 1003 1007 1004 1000
```

Alternatively, the mesh might be described as a stick figure using a collection of lines (two node faces), as shown below:

```
8
2 101 102
2 102 103
2 103 104
2 104 105
2 105 106
```

```
2 106 107
2 107 108
2 108 109
```

17. Typical NX Nastran data entry requirements are described below.

    Typical Parameters:

    - PARAM,RESVEC,character_value – controls calculation of residual flexibility (including inertia relief) modes. In SOL 103, residual flexibility is on by default for only component modes (o-set).
    - PARAM,GRDPNT, value - mass invariants ${}^{1}I$, ${}^{2}I$, and ${}^{7}I$ will be computed using results of NX Nastran grid point weight generator execution in the basic coordinate system.

    Typical Case Control:

    - ADAMSMNF FLEXBODY=YES is required for MNF generation.
    - METHOD=n is required before or in the first subcase for modal solutions.
    - SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.
    - OUTPUT(PLOT) is necessary to define elements used to select grids to display the component in Adams when PSETID=ALL or setid.

      SET n=list of elements (including PLOTELs) is used to select grids to display the component.
    - OUTPUT(POST) is necessary to define volume and surface for grid stress or strain shapes.

      SET n=list is a list of elements for surface definition for grid stress or strain shapes.

      Stress and strain data in the MNF is limited to the six components (i.e. 3 normal and 3 shear) for a grid point for a given mode.

      SURFACE n SET n NORMAL z3 is used to define a surface for writing stress and strain data. Only one FIBER selection is allowed for each SURFACE, thus the use of the FIBRE ALL keyword on the SURFACE case control command will write stresses to the MNF at the Z1 fiber location only.

      Since the FIBRE keyword only applies to stresses, strain data will always be written to the MNF at the MID location.

      Stress and strain data at grid points can only be written to the MNF for surface and volume type elements (e.g. CQUAD and CHEXA).

VOLUME n SET n is a volume definition.

The default SYSTEM BASIC is required with SURFACE or VOLUME.

- STRESS(PLOT) is necessary for stress shapes.
- STRAIN(PLOT) is necessary for strain shapes.
- GPSTRESS(PLOT) is necessary for grid point stress shapes to be included in the MNF.
- GPSTRAIN(PLOT) is necessary for grid point strain shapes to be included in the MNF.

Typical Bulk Data:

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for MNF generation. For input files containing superelements, this command must reside in the main bulk data section.
- SPOINT,id_list defines and displays modal amplitude.SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.
- SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.
- RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.
- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).
- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main Bulk Data Section.
- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).
- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).
- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in Adams.
- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

18. ADAMSMNF and MBDEXPORT case control entries cannot be used in the same analysis run. In other words, an Adams MNF file or a RecurDyn RFI file can be

generated during a particular NX Nastran execution, but not both files at the same time. Attempting to generate both files in the same analysis will cause an error to be issued and the execution to be terminated.

19. The ADMOUT=YES option is used when you would like results recovery (using the ADMRECVR case control entry) from an Adams/Flex analysis. This option requires the following assignment command:

    ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

    inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be used as input into an NX Nastran SOL 103 run using the ADMRECVR case control capability to perform results recovery from an Adams/Flex analysis. FLEXBODY=YES is required with its use.

    The data blocks output are:

    MGGEW - physical mass external sort with weight mass removed
    MAAEW - modal mass
    KAAE - modal stiffness
    CMODEXT - component modes.

    This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

    If differential stiffness is included, the static portion of the results will not be included in the recovered results when using ADMRECVR or MBDRECVR.

20. Setting CHECK=YES (which is only available when ADMOUT=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06.

21. The ADAMSMNF data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using ADAMSMNF. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

22. If any damping is defined in the model, an equivalent modal viscous damping will be determined for each mode and written to the MNF. This equivalent modal viscous damping is defined as:

    $D = \psi^T B_e \psi$

    where D is the equivalent modal viscous damping matrix, ψ is the eigenvector matrix, and B_e is the equivalent viscous damping matrix.

    The equivalent viscous damping matrix is given by:

$$B_e = B^1_{AA} + B^2_{AA} + \frac{G}{W3} K^1_{AA} + \frac{1}{W4} K^4_{AA}$$

where *G*, *W3*, and *W4* are structural damping-related parameters described in the “Parameter Descriptions” section of this guide.

By default, the full equivalent modal viscous damping matrix is written to the MNF. To write only the diagonal values of the equivalent modal viscous damping matrix to the MNF, specify NONCUP=–2 or specify PARAM,NONCUP,-2.

If both the NONCUP describer and the NONCUP parameter are specified, the NONCUP describer specification takes precedence.

23. Preload conditions are not supported.

ADAPTERR

Error estimator output request for a SOL 401 statics subcase.

Controls the computation and output of error estimates from a statics subcase.

FORMAT:

$$\text{ADAPTERR}\left[\left(\text{STRESS}, \text{STNERGY}, \text{STEP}, \begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
ADAPTERR(STRESS)=ALL
ADAPTERR(STRESS,STNERGY,PUNCH)=ALL
ADAPTERR(STEP)=ALL
```

DESCRIBERS:

Describer	Meaning
STRESS	Request the stress norm and the stress error norm. See Remarks 3 and 4.
STNERGY	Request the strain energy norm and strain energy error norm. See Remarks 3 and 4. (Default)
STEP	Request the output at the output increment steps defined with the TSTEP1 entries. See Remark 5.
PRINT	Compute and write output to the print (.f06) file. (Default)
PLOT	Compute output only.
ALL	Compute error estimates.
NONE	Do not compute error estimates.

REMARKS:

1. The ADAPTERR case control command is only supported in SOL 401. The error estimates are only computed in the statics subcase type (ANALYSIS=STATICS). The ADAPTERR command must be defined above the subcases (globally).

2. The calculation of error estimates is supported for the following element types:

Solid elements	CHEXA, CPENTA, CPYRAM, CTETRA (excludes CHEXA and CPENTA elements referencing PCOMPS bulk entries)
Axisymmetric elements	CQUADX4, CQUADX8, CTRAX3, CTRAX6
Plane strain elements	CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8 (includes elements referencing PGPLSN bulk entries)
Plane stress elements	CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8

3. The stress norm is calculated from:

$$\|u\|_{L_2} = \sqrt{\frac{1}{\Omega}\int_{\Omega} \sigma_{unaveraged}^{T}\sigma_{unaveraged}d\Omega}$$

The stress error norm is calculated from:

$$\|e_{\sigma}\| = \sqrt{\frac{1}{\Omega}\int_{\Omega}\left(\sigma_{unaveraged} - \sigma_{averaged}\right)^{T}\left(\sigma_{unaveraged} - \sigma_{averaged}\right)d\Omega}$$

The strain energy norm is calculated from:

$$\|u\| = \sqrt{\int_{\Omega} \sigma_{unaveraged}^{T} D^{-1}\sigma_{unaveraged}d\Omega}$$

The strain energy error norm is calculated from:

$$\|e_{U}\| = \sqrt{\int_{\Omega}\left(\sigma_{unaveraged} - \sigma_{averaged}\right)^{T} D^{-1}\left(\sigma_{unaveraged} - \sigma_{averaged}\right)d\Omega}$$

where

Ω is the element volume,

$\sigma_{unaveraged}$ is the unaveraged stress vector,

$\sigma_{averaged}$ is an averaged stress vector computed at a grid point using the stress vectors from elements connected to the grid point,

D matrix is the constitutive relation.

When computing $\sigma_{averaged}$, stress values are not averaged across different element families, material properties, material coordinate systems, orientation angles in 2D solid elements, and thicknesses in plane stress elements.

4. You can specify both the STRESS and STNERGY describers to request stress norm, stress error normal, strain energy norm and strain energy error norm output.

5. The software always outputs the maximum value on each element, for each output type requested, by comparing the values from all solution steps. In addition, if you specify the STEP describer, the software will output what you have requested at the output increment steps defined with the TSTEP1 entries.

ADMRECVR

Adams stress recovery.

Recovers stress results from an Adams/Flex analysis.

FORMAT:

$$\text{ADMRECVR}\left[\text{ADMFORM} = \begin{Bmatrix}\text{BINARY}\\ \text{ASCII}\end{Bmatrix}\right],$$

$$\left[\text{MSRMODE} = \begin{Bmatrix}0\\ 1\\ 2\end{Bmatrix}\right],\left[\text{RGBODY} = \begin{Bmatrix}\text{NO}\\ \text{YES}\\ \text{BOTH}\end{Bmatrix}\right],$$

$$\left[\text{MSGLVL} = \begin{Bmatrix}0\\ 1\text{ - }4\end{Bmatrix}\right],\left[\text{ADMCHK} = \begin{Bmatrix}\text{NO}\\ \text{YES}\end{Bmatrix}\right]$$

EXAMPLES:

```
ASSIGN INPUTT2=`adams_results.mdf' UNIT=13
...
CEND

STRESS(PLOT)=100
ADMRECVR
```

DESCRIBERS:

Describer	Meaning
ADMFORM	Specifies the format of the Adams/Flex modal deformations file (see remarks 1 and 2)
BINARY	OUTPUT2 file (default)

Describer	Meaning
ASCII	PUNCH file.
MSRMODE	Specifies stress recovery type (see remarks 6 and 7).
0	Component modal definitions are stored in an OUTPUT2 file (specifically, a *.out file was created by using ADMOUT=YES on the ADAMSMNF case control in a pre-Adams/Flex NX Nastran run). The OUTPUT2 files used in this case do not contain data blocks used for MNF creation (default).
1	Same as option 0, except that the OUTPUT2 file will contain 10 additional data blocks used for MNF creation by an Adams pre-processor (specifically, a *.out file created through use of the mnfx.alt DMAP alter capability).
2	No file reference (specifically, component modal definitions will be recomputed)
RGBODY	Requests the addition of rigid body motion with modal deformations (see remark 5).
NO	Do not include rigid body motion (default).
YES	Include rigid body motion.
BOTH	Generate two output data blocks; one containing rigid body motion and the other without.
MSGLVL	Level of diagnostic output from Lanczos eigensolver when component modal definitions are determined (applies only when MSRMODE=2).
0	No output (default).
1	Warning and fatal messages.
2	Summary output.
3	Detailed output on cost and convergence.
4	Detailed output on orthogonalization.
ADMCHK	Requests debug output be written to the f06 file (See Remark 9).

Describer	Meaning
NO	No debug output will be written (default).
YES	Debug output will be written.

REMARKS:

1. When modal deformations to be read are in binary (OUTPUT2) format (specifically, ADMFORM=BINARY), the following statement needs to be specified near the top of the NX Nastran input file in the file management section:

   ASSIGN INPUTT2=`<MDFilename>' UNIT=13

   where <MDFilename> is the name of the modal deformations file from Adams.

2. To input the modal deformations file from Adams in ASCII (Punch) format (specifically, ADMFORM=ASCII), the following statement needs to be included in the bulk data section:

   INCLUDE `<MDFilename>' where `<MDFilename>' is the name of the modal deformations file.

3. Dynamic stress/strain output can either be in .f06, PUNCH, and/or OUTPUT2 according to standard NX Nastran functionality. However, stress recovery in NX Nastran from Adams/Flex results does not support XYPLOT output.

4. If displacements, stresses, and/or strains are to be available for post processing, one or more of the following statements must appear in the case control section of the NX Nastran input file:

   DISP(PLOT) = <set id>

   STRAIN(FIBER,PLOT) = <set id>

   STRESS(PLOT) = <set id>

   Even though the recovery process is performed in SOL 103, the derived results are transient response results. The default output format for transient results is SORT2. If no SORT2 results are desired, then SORT1 must be requested for all output requests.

5. Rigid body motions from an Adams simulation are included in the modal deformation file, but they are not applied unless the RGBODY keyword is set to YES or BOTH. Including rigid body motion affects the display and animation of the flexible component, but it has no effect on dynamic stresses. For RGBODY=YES, the displacements output data block OUGV1 or OUGV2 will contain a ‘0’ in word

3 of each header record. For RGBODY=NO, OUGV1 or OUGV2 will contain a ‘1’ in word 3 of each header record. For RGBODY=BOTH, two OUGV1 or OUGV2 data blocks will be generated; one for RGBODY=YES and one for RGBODY=NO.

6. For MSRMODE=0 or 1, stress recovery references the OUTPUT2 file obtained from the initial CMS analysis (specifically, ADMOUT=YES on the ADAMSMNF case control entry or use of the mnfx.alt DMAP alter capability). No other files are required. The geometric data needs to be included in the bulk data of the NX Nastran input file because geometry is missing from the OUTPUT2 file. This mode of stress recovery is faster than the MSRMODE=2 mode. To reference this OUTPUT2 file the following line needs to be included in the file specification section of the NX Nastran input file:

   ASSIGN INPUTT2=`<OUTPUT2_filename>' UNIT=20

7. For MSRMODE=2, no files are referenced for stress recovery. Instead, a full CMS reanalysis is performed to build the reference data for the stress recovery analysis. Obviously, the analysis time is significantly far greater compared to the MSRMODE=0 or 1 method, but this method frees up disk space. There is also risk in using this method. If the reanalysis generates slightly different component eigenvalues or eigenvectors than were generated during the creation of the Adams MNF in the initial NX Nastran run, then the Adams results in the Adams MDF (modal deformation file) will be inconsistent and incorrect results will be recovered. Something as simple as a sign change for one eigenvector will cause incorrect results to be recovered. It is, therefore, *highly recommended* that MSRMODE=0 or 1 *always* be used.

8. This capability must be performed in SOL 103 and is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

9. Setting ADMCHK=YES is *not* recommended for models of realistic size due to the amount of data that will be written to the f06.

AECONFIG

Aeroelastic Configuration Name

Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases. The configuration name is the Aerodynamic Supergroup identified as part of the aeroelastic model in the Aeroelasticity module prior to analysis submission. The standard location of the AECONFIG entry will be above the SUBCASE entries. An AECONFIG located within a SUBCASE entry will override the global AECONFIG; this requires attachment to an existing aerodynamic database. If the AECONFIG is not found, new qualified datablocks will be created from the data in the Bulk Data Section.

FORMAT:

Assigns a DBset member name

```
AECONFIG        =config-name.
```

EXAMPLES:

Assigns a MASTER file for the aerodynamic and aeroelastic DBsets.

```
AECONFIG        =CONFIG_A
```

DESCRIBERS:

Describer	Meaning
config-name	The configuration name. This is the Aerodynamic SuperGroup name identified as part of the aeroelastic model.

AEROF

Aerodynamic Force Output Request

Requests the aerodynamic loads on aerodynamic control points.

FORMAT:

AEROF=n

EXAMPLES:

```
AEROF=ALL
AEROF=5
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a previously appearing SET command. (Integer>0)
ALL	Output forces at all points.

REMARKS:

1. The SET command references box or body element identification numbers.
2. Output is in the units of force or moment.
3. Only aerodynamic forces on points specified on the SET command will be output.

AESYMXY

Aerodynamic Flow Symmetry About XY Plane

Aerodynamic XY plane of symmetry flag. This is used to indicate whether the aerodynamic model has symmetry with respect to the ground. This selection is typically done within each subcase, but a case control default can be defined by placing an entry above the subcase.

FORMAT:

$$\text{AESYMXY}=\left\{\begin{array}{c}\text{SYMMETRIC}\\ \text{ANTISYMMETRIC}\\ \text{ASYMMETRIC}\end{array}\right\}$$

EXAMPLE 1:

AESYMXY=ASYMMETRIC

DESCRIBERS:

Describer	Meaning
SYMMETRIC	Indicates that the aerodynamic model is moving in a symmetric manner with respect to the XY plane.
ANTISYMMETRIC	Indicates that the aerodynamic model is moving in an antisymmetric manner with respect to the XY plane.
ASYMMTRIC	Indicates that the aerodynamic model has no reflection about the XY plane.

REMARKS:

1. If the AESYMXY and AESYMXZ commands are not present for a particular subcase, aerodynamic XY symmetry will be determined from the SYMXY field of the AEROS Bulk Data entry for static aeroelastic analysis and from the SYMXY field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.

2. Symmetric implies ground effect while asymmetric implies free air analysis.

AESYMXZ

Aerodynamic Flow XZ Symmetry About XZ Plane

Aerodynamic XZ plane of symmetry flag. This is used to support symmetric models about the centerline. Default is ASYMMETRIC (no symmetry). (Subcase level inheritance from AESYMXY applies.)

FORMAT:

$$\text{AESYMXY}=\left\{\begin{array}{c}\text{SYMMETRIC}\\ \text{ANTISYMMETRIC}\\ \text{ASYMMETRIC}\end{array}\right\}$$

EXAMPLE 1:

AESYMXZ=SYMMETRIC

DESCRIBERS:

Describer	Meaning
SYMMETRIC	Indicates that a half span aerodynamic model is moving in a symmetric manner with respect to the XZ plane.
ANTISYMMETRIC	Indicates that a half span aerodynamic model is moving in an antisymmetric manner with respect to the XZ plane.
ASYMMETRIC	Indicates that a full aerodynamic model is provided. (Default.)

REMARKS:

If the AESYMXZ and AESYMXY commands are not present, aerodynamic XZ symmetry is determined from the SYMXZ field of the AEROS Bulk Data entry for

static aeroelastic analysis and from the SYMXZ field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.

ANALYSIS

Analysis Discipline Subcase Assignment

Specifies the type of analysis being performed for the current subcase.

FORMAT:

ANALYSIS=type

EXAMPLES:

```
ANALYSIS=STATICS
ANALYSIS=MODES
```

DESCRIBERS:

Table 4-1. SOL 200 Analysis Types

Describer	Meaning		
type	Analysis type. Allowable values and applicable solution sequences (Character):		
	STATICS	Statics	SOL 200
	MODES	Normal Modes	
	BUCK	Buckling	
	DFREQ	Direct Frequency	
	MFREQ	Modal Frequency	
	MTRAN	Modal Transient	
	DCEIG	Direct Complex Eigenvalue Analysis	

Table 4-1. SOL 200 Analysis Types

Describer	Meaning		
	MCEIG	Modal Complex Eigenvalue Analysis	
	SAERO	Static Aeroelasticity	
	DIVERGE	Static Aeroelastic Divergence	
	FLUTTER	Flutter	

Table 4-2. SOL 401 Analysis Types

Describer	Meaning		
type	Analysis type. Allowable values and applicable solution sequences (Character):		
	STATICS	Statics	SOL 401
	MODES	Normal Modes	
	PRELOAD	Bolt Preload	
	CYCMODES	Cyclic Modes	
	FOURIER	Fourier Modes	

Table 4-3. SOLs 108, 110, 111, and 112 Analysis Types

Describer	Meaning
type	Analysis type. Allowable values and applicable solution sequences (Character):

Table 4-3. SOLs 108, 110, 111, and 112 Analysis Types

Describer	Meaning		
	MODES	Normal Modes	SOLs 110, 111, and 112 only.
	RANDOM	Random Analysis	SOLs 108 and 111 only.

Table 4-4. SOLs 153, 159, 601,153 and 601,159 Analysis Types

Describer	Meaning		
type	Analysis type. Allowable values and applicable solution sequences (Character):		
	HEAT	Heat Transfer Analysis	SOLs 153, 159, 601,153 and 601,159 only.
	STRUCTURE	Structural Analysis	

REMARKS:

1. ANALYSIS = STRUC is the default in SOLs 153 and 159.

2. In SOL 200, all subcases, including superelement subcases, must be assigned by an ANALYSIS command either in the subcase or above all subcases. Also, all subcases assigned by ANALYSIS=MODES must contain a DESSUB request. If a SOL 200 job contains both ANALYSIS=STATICS and ANALYSIS=BUCK subcases, the STATICS subcases should come before ANALYSIS=BUCK. A SOL 200 job may not contain both DFREQ and MFREQ subcases at the same time.

3. ANALYSIS=DIVERG is only available for analysis in SOL 200. Sensitivity and optimization are not supported for this analysis type.

4. In order to obtain normal modes data recovery in SOLs 110, 111, and 112, ANALYSIS = MODES must be specified under one or more separate subcase(s) which contains requests for data recovery intended for normal modes only. For example, in SOL 111:

```
METH=40
SPC=1
SUBCASE 1 $ Normal Modes
  ANALYSIS=MODES
  DISP=ALL
SUBCASE 2 $ Frequency response
  STRESS=ALL
  DLOAD=12
  FREQ=4
```

All commands which control the boundary conditions (SPC, MPC, and SUPORT) and METHOD selection should be copied inside the ANALYSIS=MODES subcase or specified above the subcase level.

5. ANALYSIS = RANDOM is only available for analysis in SOLs 108 and 111. It is used to simplify the process of performing multiple random analyses via the RANDOM and optionally RCROSS case control commands over frequency response subcases of the same frequencies. For example, in SOL 111:

```
SDAMPING=101
SPC=2
ACCELERATION(RPRINT)=ALL
STRESS(SORT2,PSDF)=ALL
RCROSS(PRINT)=1
METHOD=1
$
SUBCASE 1 $ Normal Modes
  ANALYSIS=MODES
  DISP=ALL
SUBCASE 2 $ Frequency Response using FREQ set 13
  FREQUENCY=13
  DLOAD=111
SUBCASE 3 $ Frequency Response using FREQ set 13
  FREQUENCY=13
  DLOAD=211
SUBCASE 4 $ Random Response referencing FRF from subcases 2 and 3
  ANALYSIS=RANDOM
  RANDOM=100
SUBCASE 5 $ Random Response referencing FRF from subcases 2 and 3
  ANALYSIS=RANDOM
  RANDOM=200
$ Change frequencies
SUBCASE 11 $ Frequency Response using FREQ set 23
  FREQUENCY=23
  DLOAD=111
SUBCASE 12 $ Frequency Response using FREQ set 23
  FREQUENCY=23
  DLOAD=211
SUBCASE 14 $ Random Response referencing FRF from subcases 11 and 12
  ANALYSIS=RANDOM
```

```
  RANDOM=1100
SUBCASE 15 $ Random Response referencing FRF from subcases 11 and 12
  ANALYSIS=RANDOM
  RANDOM=1200
```

REMARKS RELATED TO SOL 601:

1. ANALYSIS=STRUC and ANALYSIS=HEAT are supported for SOL 601,153 and SOL 601,159.

2. For SOL 601,153 and SOL 601,159, two subcases are required. The first two subcases must be one with ANALYSIS=STRUC (default) and one with ANALYSIS=HEAT. The parameter COUP in TMCPARA bulk entry is used to specify the type of coupling between the structural and heat transfer analysis.

APRESSURE

Aerodynamic Pressure Output Request

Requests the aerodynamic pressures in static aeroelastic response.

FORMAT:

$$\text{APRES} = \left\{ \begin{array}{c} \text{n} \\ \text{ALL} \end{array} \right\}$$

EXAMPLES:

APRES=ALL

APRES=6

DESCRIBERS:

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only aerodynamic pressures on the referenced aerodynamic boxes will be output. (Integer>0)
ALL	Pressures at all points will be output.

AUXCASE

Auxiliary Model Case Control Delimiter

Delimits Case Control commands for an auxiliary model in SOL 200.

FORMAT:

AUXCASE

EXAMPLES:

AUXCAS

AUXC

REMARKS:

1. AUXCASE indicates the beginning of Case Control commands for an auxiliary model. AUXCASE commands must follow the primary model Case Control commands.

2. All Case Control commands following this entry are applicable until the next AUXCASE or BEGIN BULK command. Commands from preceding Case Control Sections are ignored.

3. Each auxiliary model Case Control must be delimited with the AUXCASE command.

4. The AUXMODEL command is used to associate the auxiliary model Case Control with a particular auxiliary model.

AUXMODEL

Auxiliary Model Identification Number

References an auxiliary model for generation of boundary shapes in shape optimization.

FORMAT:

AUXMODEL=n

EXAMPLES:

```
AUXMODEL=4
AUXM=4
```

DESCRIBERS:

Describer	Meaning
n	Auxiliary model identification number. (Integer>0)

REMARKS:

1. AUXMODEL references a particular auxiliary model for analysis and may only be specified in the auxiliary model Case Control Section.
2. See the BEGIN BULK command for the Bulk Data definition of an auxiliary model.

B2GG

Direct Input Damping Matrix Selection

Selects direct input damping matrices.

FORMAT:

B2GG=name

EXAMPLES:

```
B2GG=BDMIG
B2GG=BDMIG1, BDMIG2, BDMIG3
B2GG=1.25*BDMIG1, 1.0*BDMIG2, 0.75*BDMIG3
SET 100=B1, B2
B2GG=100
```

DESCRIBERS:

Describer	Meaning
name	Name of $[B^2_{gg}]$ matrix that is input on the DMIG Bulk Data entry. Scale factors may be included (see Remarks 4 and 5). See “Matrix Assembly Operations in SubDMAP SEMG” in the *NX Nastran User’s Guide*. (Character)

REMARKS:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the damping matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG,name entry must contain the integer 6.

4. The associated DMIG matrices can be scaled using either in-line scale factors on B2GG (for example, B2GG=1.25*BDMIG1), using the parameter CB2 (for example, PARAM,CB2,1.25), or both. See “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

   ```
   B2GG=1.25*BDMIG1,1.0*BDMIG2,0.75*BDMIG3
   ```

   The parameter CB2 when defined will scale all B2GG. For example, if PARAM,CB2,1.30 is defined with the B2GG example above, the result would be B2GG=1.30(1.25*BDMIG1 + 1.0*BDMIG2 + 0.75*BDMIG3).

6. The B2GG command is limited to use in the first subcase only.

B2PP

Direct Input Damping Matrix Selection

Selects direct input damping matrices.

FORMAT:

B2PP=name

EXAMPLES:

```
B2PP=BDMIG
B2PP=BDMIG1, BDMIG2, BDMIG3
B2PP=1.25*BDMIG1, 1.0*BDMIG2, 0.75*BDMIG3
B2PP=(1.25,0.5)*BDMIG1, (1.0,0.0)*BDMIG2, (0.75,-2.2)*BDMIG3
SET 100=B1, B2
B2PP=100
```

DESCRIBERS:

Describer	Meaning
name	Name of [B^2_{pp}] matrix that is input on the DMIG Bulk Data entry. Scale factors may be included (see remarks 7 and 8). See “Formulation of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide*. (Character)

REMARKS:

1. DMIG entries will not be used unless selected.
2. B2PP is used only in dynamics problems.
3. The matrix must be square or symmetric and field 4 on the DMIG,name entry must contain a 1 or 6.

4. It is recommended that PARAM,AUTOSPC,NO be specified. See “Constraint and Mechanism Problem Identification in SubDMAP SEKR” in the *NX Nastran User’s Guide*.

5. The B2PP command is supported across subcases. A B2PP command selecting a different DMIG matrix can be defined for each subcase.

6. The associated DMIG matrices can be scaled using in-line scale factors on B2PP (for example, B2PP=1.25*BDMIG1).

7. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

```
B2PP=1.25*BDMIG1,1.0*BDMIG2,0.75*BDMIG3
```

BC

Boundary Condition Identification

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

FORMAT:

BC=n

EXAMPLES:

BC=23

DESCRIBERS:

Describer	Meaning
n	Identification number. (Integer>0)

REMARKS:

1. In SOLs 103, 105, 145, and 200 BC is required in each subcase if multiple boundary conditions are specified for normal modes, buckling, and flutter analysis.
2. If only one boundary condition is specified, then BC does not have to be specified and n defaults to zero.

BCRESULTS

Contact Result Output Request (SOLs 101, 103, 111, 112, 401, 601, and 701)

FORMAT:

$$\text{BCRESULTS}\left[\left(\text{TRACTION}, \text{FORCE}, \text{SEPDIS}, \text{STATUS}, \begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
BCRESULTS=ALL
BCRESULTS(FORCE,PLOT)=ALL
BCRESULTS(TRACTION,FORCE,PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
TRACTION	Contact pressure (scalar) and in-plane contact tractions (vector in basic coordinate system with units of force per area) are output for each contact grid point.
FORCE	Contact force vector in basic coordinates is output for each contact grid point.
SEPDIS	For SOLs 101, 103, 111, and 112, requests the initial and final separation distance for grids on the source region. For SOL 401, requests the final separation distance, and the total and incremental slide distance for grids on the source and target regions. See Remark 1.
STATUS	Requests the status of contact elements in SOL 401. See Remark 3.

Describer	Meaning
PRINT	The printer will be the output medium.
PLOT	Computes and puts contact results in OP2 file only.
PUNCH	The punch file is the output media.
ALL	Contact results at all contact grid points will be output.
NONE	Contact results will not be output.
n	Set identification of a previously appearing SET command. Only contact grid points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS :

1. For SOLs 101, 103, 111, and 112, the SEPDIS describer requests both the initial and final separation distance for grids on the source region. The separation distance is a scalar quantity representing the source side normal distance to the target. During the solution, the separation distance is known at the element integration points, but is written to the grids when output. The result at each source grid is the value of separation distance at the closest contact element. If there are two or more contact elements equidistant from the grid, then the minimum value of separation distance is used at the grids rather than the average, since the average gives unexpected results for coarse meshes.

   For SOL 401, the SEPDIS describer requests the final separation distance for grids on both the source and target regions. It is computed based on the current deformed configuation. For grids on the source region, the separation distance is a scalar quantity representing the source side normal distance to the target. For grids on the target region, the separation distance is a scalar quantity representing the target side normal distance to the source. During the solution, the separation distance is known at the element integration points, but is written to the grids when output. The result at each source and target grid is the value of separation distance at the closest contact element. If there are two or more contact elements equidistant from the grid, then the minimum value of separation distance is used at the grids rather than the average, since the average gives unexpected results for coarse meshes.

   In addition for SOL 401, the SEPDIS describer requests the total and incremental slide or slip distance for grids on both the source and target regions. By default, it requests slide output. If you set the system cell 642 to 0, the SEPDIS describer requests the total and incremental slip.

Slide output in SOL 401

SOL 401 computes slide as:

$$u_{slide} = (u_s - u_t) - \hat{\eta} \cdot (u_s - u_t)$$

The slide distance is a relative displacement in the tangential direction between the source and target faces. It is computed without regard to the status of the contact condition. For example, a source and target may not be in contact but their locations are changing relative to one another. These changes are included in the slide distance output. The tangential slide distances are output in the basic coordinate system. The incremental slide distance is the sliding which occurred since the last output step. The total slide distance is computed in the current deformed configuation and relative to the initial, undeformed configuration. For example, if a model is loaded then unloaded in several subcases causing a source and target to slide a distance and then return to their initial relative positions, the total slide distance in this example is zero.

Slip output in SOL 401

The slip distance computed by SOL 401 is a summation of tangential slipping which occurs when the contact condition has the status of "slipping". As a result, slip distance is only computed when a source and target are actually in contact. Slip output is not computed when contact elements have a status of "sticking" or "inactive". The status of "slipping" occurs when the contact frictional force (contact normal force x coefficient of friction) is smaller than the tangential loads. The tangential slip distances are output in the basic coordinate system. The incremental slip distance is the slipping which occurred since the last output step. The total slip distance is the summation of the incremental slipping which occurred from all previous solution steps from all static subcases. Although, if a contact element becomes inactive as a result of the source and target separating, the accumulated total slip is reset back to zero.

2. For edge-to-edge contact, force, traction, and separation distance output can be requested. By default, the contact force for axisymmetric elements is based on a 2π section. The system cell 587 can optionally be set to 1 to select the per radian section basis. The contact force unit is consistent with the applied and reaction force unit for each element.

3. When the STATUS describer is included on the BCRESULTS command in SOL 401, an integer value indicating the contact status is output on each grid point included in a contact source or target region. The status values are:

   0: No contact exists.

   1: A sticking contact condition exist.

   2: A sliding contact condition exist.

REMARKS RELATED TO SOLS 601 AND 701:

1. The axisymmetric elements CTRAX3, CQUADX4, CTRAX6, and CQUADX8 are always on a per radian basis in solutions 601 and 701. As a result, the edge-to-edge contact force unit for axisymmetric elements is “force/radian”.

BCSET

Contact Set Selection (SOLs 101, 103, 105, 111, 112, 401, 601 and 701)

Selects the contact set for SOLs 101, 103, 105, 111, 112, 401, 601 and 701.

FORMAT:

BCSET=n

EXAMPLES:

```
BCSET=5
```

DESCRIBERS:

Describer	Meaning
n	Contact set identification of a BCTSET, BCTADD or BCTPARM Bulk Data entry. (Integer>0)

REMARKS:

1. BCSET references a BCTSET Bulk Data entry if there is only one contact set. When multiple contact sets are used, BCSET must reference a BCTADD bulk data entry.

2. The following table summarizes all supported contact conditions.

Table 4-5. Contact Summary

Type	Description	Solution Support

Table 4-5. Contact Summary		
Edge-to-Edge	Contact between the edges of axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8).	Solution 101 and consecutive solutions 103, 105, 111, and 112 (elements in XZ or XY plane). Solution 401 (elements in XZ or XY plane). Solution 601 (elements in XZ plane only).
Surface-to-Surface	Contact between shell element faces (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR) and solid element faces (CHEXA, CPYRAM, CPENTA, CTETRA).	Solution 101 and consecutive solutions 103, 105, 111, and 112. Solution 401 (solid element faces only: CHEXA, CPYRAM, CPENTA, CTETRA). Solutions 601 and 701.

3. A converged SOL 101 contact condition can be included in a consecutive normal mode solution (SOL 103). The normal mode solution results, which included the SOL 101 contact conditions, can then be used in an optional dynamic response calculation (SOLs 111 and 112). See the chapter on contact in the *NX Nastran User's Guide* for more information.

4. When a BCSET case control command exists, CGAP elements are treated as linear contact elements if the system cell (412) OLDGAPS is set to 0 (default). You can use CGAP elements this way with surface-to-surface contact defined (BCTSET bulk entries exist), or without (no BCTSET bulk entries exist). In the case where no surface-to-surface contact is defined, “n” can point to a BCTPARM bulk entry which optionally defines PENN, PENT or PENTYP for the CGAP/linear contact elements, or to nothing if a BCTPARM bulk entry does not exist (an integer value for “n” is still required in this case). See the chapter on surface contact in the NX Nastran User’s Guide for more information.

5. If the contact solution is having problems converging, the fields on the BCTPARM bulk data entry can be modified from their defaults. See the BCTPARM bulk data entry, and the chapter on surface contact in the NX Nastran User’s Guide for more information.

6. Contact conditions can be included in SMP and RDMODES solutions. See “Recursive Domain Normal Modes Analysis (RDMODES)” in the *Parallel*

Processing Guide. Contact conditions cannot be included in a GDMODES, FDMODES, or HDMODES DMP solution.

7. When defining contact regions and pairs on geometry which is not tangent continuous, creating single contact regions which cross corner transitions can result in non-uniform stress results around the corners. It is recommended to break these areas into multiple regions and pairs. See the chapter on contact in the *NX Nastran User's Guide* for more information.

8. Contact definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the contact definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BCRESULTS case control command.

9. If multiple eigenvalue subcases (solutions 103 or 105) include STATSUB commands to select linear static subcases, and the linear static subcases include glue and/or linear contact definitions, the BCSET and/or BGSET case control commands must be specified in the global case. Failure to do this can result in erroneous results.

10. Contact conditions can be included in a model with main bulk data type superelements, but the contact regions must be in the residual. Contact conditions are not supported in a model with partitioned type superelements. Main bulk data and partitioned type superelements are both described in Chapter 2 of the *NX Nastran Superelement User's Guide*.

11. You cannot restart from a solution which included contact conditions. You cannot add contact conditions when restarting.

12. The combination of a bolt preload in which the bolt is meshed with solid elements, the element iterative solver, and contact conditions is not supported.

BEGIN BULK

Case Control and Bulk Data Delimiter

Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.

FORMAT:

$$\text{BEGIN}\,[\,\text{BULK}\,]\begin{bmatrix}\text{AUXMODEL} = \text{auxmid}\\ \text{SUPER} = \text{seid}\end{bmatrix}$$

EXAMPLES:

```
BEGIN BULK
BEGIN AUXMODEL=22
```

DESCRIBERS:

Describer	Meaning
AUXMODEL	Indicates the beginning of an auxiliary model Bulk Data Section.
auxmid	Auxiliary model identification number. (Integer>0)
SUPER	Indicates the beginning of partitioned superelement Bulk Data Section.
seid	Superelement identification number. (Integer ≥ 0)

REMARKS:

1. BEGIN BULK is not required if there are no Bulk Data entries and there is no ENDDATA entry. The BEGIN SUPER or BEGIN AUXMODEL Bulk Data must lie between BEGIN BULK and ENDDATA entries.

2. For an auxiliary model, AUXMID is referenced by the AUXMODEL Case Control command.

3. Partitioned Bulk Data Sections defined by BEGIN SUPER are used to define only one superelement each. Bulk Data commands which define superelements are ignored in partitioned Bulk Data Sections.

   Superelements specified by a BEGIN SUPER entry can be automatically attached to other superelements based on relative location of grid points. For connection to the downstream superelement, the global coordinate directions of the attachment grid points of the upstream superelement will be internally transformed to the global coordinate directions of the grid points of the downstream superelement. For displacement data recovery, the output will be in the original global coordinate directions.

BGRESULTS

Glue Result Output Request (SOLs 101, 103, 105, 401, 601)

FORMAT:

$$\text{BGRESULTS}\left[\left(\text{TRACTION}, \text{FORCE}, \text{SEPDIS}, \begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix}, \begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
BGRESULTS=ALL
BGRESULTS(FORCE,PLOT)=ALL
BGRESULTS(TRACTION,FORCE,PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
TRACTION	Glue normal traction (scalar) and in-plane glue tractions (vector in basic coordinate system) are output for each glue grid point. Traction units are force per area.
FORCE	Glue force vector is output for each glue grid point.
SEPDIS	For SOL 401 only. Requests the final separation distance, and the total and incremental slide distance for grids on the source and target regions.
PRINT	The printer will be the output medium.
PLOT	Computes and puts glue results in OP2 file only.
PUNCH	The punch file is the output media.

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue or time, depending of the solution sequence.
SORT2	Output will be presented as a tabular listing of load,frequency or time for each grid point (not supported for SOL 401).
ALL	Glue results at all contact grid points will be output.
NONE	Glue results will not be output.
n	Set identification of a previously appearing SET command. Only glue grid points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. The glue traction request is supported in solutions 101, 103 (but only for a static preload subcase if present), 105, and 401 (static subcase only). The glue results from SOL 105 are a result of the applied static loads which are not necessarily the loads at which buckling occurs.

2. Sliding glue can be requested in SOL 401 with the parameter setting SLIDE=1 on the BGPARM entry. Sliding glue includes a normal stiffness, but no tangential stiffness. When sliding glue is requested, the SEPDIS describer can be used to request slide distance output. The slide distance represents the tangential distance traveled by the source and target grids relative to each other. The total and incremental slide distance output is written in the basic coordinate system on grid points for both the source and target regions. The total slide distance is the vector sum of the incremental slide distances from all time steps. If a subcase is sequentially dependent, the total slide distance will also include the total slide distance from a previous static subcase. Any gaps between the glue edges and surfaces are preserved as slidiing occurs.

REMARKS RELATED TO SOLS 601 AND 701:

1. Glue traction request is not support in SOL 601.

BGSET

Glue Contact Set Selection

Selects the glue contact set.

FORMAT:

BGSET=n

EXAMPLES:

```
BGSET=5
```

DESCRIBERS:

Describer	Meaning
n	Glue contact set identification of a BGSET Bulk Data entry. (Integer>0)

REMARKS:

1. The default search distance defined on the BGSET bulk entry will glue the closest sections of the source and target regions. This value can be adjusted to control the parts of a region to be glued. The software searches in both the positive and negative normal directions from the source region.

2. The following table summarizes all supported glue conditions.

Table 4-6. Glue Summary

Type	Description	Solution Support

Table 4-6. Glue Summary		
Edge-to-Edge	Glue between the edges of shell, axisymmetric, plane stress, and plane strain elements. Elements being glued must be in the same plane.	All solutions except 144 -146 and 701. SOLs 401 and 601 do not support gluing of shell edges. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.
Edge-to-Surface	Glue between shell element edges and shell or solid element faces.	All solutions except 144 -146, 401, 601, and 701. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.
Surface-to-Surface	Glue between shell or solid element faces and shell or solid element faces.	All solutions except for SOL 144–146 and 701. SOLs 401 and 601 do not support gluing of shell faces. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They can be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.

3. When defining glue regions and pairs on geometry which is not tangent continuous, creating single glue regions which cross corner transitions can result in non-uniform stress results around the corners. It is recommended to break these areas into multiple regions and pairs. See the chapter on constraints in the *NX Nastran User's Guide* for more information on glue definitions.

4. Glue definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the glue definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the

ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BGRESULTS case control command.

5. Inertia relief (PARAM,INREL) and glue conditions can be included together when using the sparse solver (default), but not the element iterative solver.

6. Shell offsets are not accounted for in edge-to-surface gluing. The gluing occurs at the grid locations and not at the offset location.

7. If multiple eigenvalue subcases (solutions 103 or 105) include STATSUB commands to select linear static subcases, and the linear static subcases include glue and/or linear contact definitions, the BCSET and/or BGSET case control commands must be specified in the global case. Failure to do this can result in erroneous results.

8. Glue conditions can be included in a model with main bulk data type superelements. Each glue pair must be contained in a single superelement or in the residual. That is, the glue regions cannot cross superelement boundaries. Glue conditions are not supported in a model with partitioned type superelements. Main bulk data and partitioned type superelements are both described in Chapter 2 of the *NX Nastran Superelement User's Guide*.

9. When gluing solid element faces, a glue condition is not created if the dot product of the element face normals is greater or equal to 0. Including SYSTEM(516)=1 will override this and allow the glue condition to be created if the search distance is satisfied.

10. You cannot add glue conditions when restarting. You can restart from a solution which included glue conditions, but only to request addition output.

11. Glue definitions model linear glue behavior in solution 106. The glue condition does not update in regard to geometry updates, nonlinear material changes, or orientation changes. A glue definition should only be defined in portions of the structure that exhibit linear behavior.

BOLTLD (case control)

Bolt Preload Set Selection

Selects a bolt preload set ID for bolt preload processing.

FORMAT:

BOLTLD=n

EXAMPLES:

```
BOLTLD=5
```

DESCRIBERS:

Describer	Meaning
n	In SOLs 101, 103, 105, 107 through 112, and 601, set identification number (SID) of a BOLTFOR or BOLTLD bulk entry. (Integer>0) In SOL 401, set identification number (SID) of a BOLTFOR, BOLTFRC, BOLTSEQ, or BOLTLD bulk entry. (Integer>0)

REMARKS:

1. Bolt preload is supported in SOLs 101, 103, 105, 107 through 112, 401, and 601.
2. Bolt preload is not supported in a solution RESTART.
3. The following remarks appy to SOLs 101, 103, 105, 107 through 112.
   - Subcases with BOLTLD case control commands should be either the first or the last.
   - In SOLs 101, 103, 105, 107 through 112, for the special case where the sparse solver is used, the model does not contain contact, and the keyword scratch = yes, BOLTLD case control commands in the subcases must be

ordered. For example, if a model contains three subcases with BOLTLD = 1 used twice and BOLTLD = 2 used once, the BOLTLD case control commands should be ordered with:

- The first and second subcases containing BOLTLD = 1 and the third subcase containing BOLTLD = 2.

- The first subcase containing BOLTLD = 2 and the second and third subcases containing BOLTLD = 1.

- Superelements with preloaded bolts are allowed. However, the elements used to define the bolts along with the bolt preload forces must be in the residual.
- Inertia relief (PARAM,INREL) is not supported with bolt preload.
- For solid bolts, the case control section must include an SPC set reference. The SPC set may be empty (no DOF assigned).
- The combination of a bolt preload in which the bolt is meshed with solid elements, the element iterative solver, and contact conditions is not supported.
- Enforced displacements defined with non-zero SPC or SPCD bulk entries can be included with a bolt preload. When an SPCD is defined with a bolt preload, system cell 581 must be set to 1 to override an error trap that exists in the software. The software behaviour changes during the bolt preload solution steps for various input scenarios (with and without contact defined; enforced displacements defined with SPCD entries or with SPC entries). See “Enforced Displacements and Bolt Preloads” in the *User’s Guide* for a description of the software behaviour.
- For the dynamic solutions 103, and 107 through 112, a static subcase containing the BOLTLD case control command must be referenced by a STATSUB case control command. This is the only method for a bolt preload to be part of a dynamics solution.

4. For solution 401, the software iterates until the resulting bolt force matches, within a tolerance, the bolt preload you requested. The final bolt strain required to achieve the desired bolt force includes the effects of any other defined loads (thermal loads, contact forces, and conditions from a previous subcase). For example, if you compare the results from a solution with only bolt preloads defined with the results from the same model with the addition of thermal loads and contact, the bolt force will be the same, but the final bolt strain needed to achieve the bolt force will be different.

5. For solution 401, a bolt preload subcase requires the ANALYSIS=PRELOAD and BOLTLD=n commands. A subcase with ANALYSIS=STATICS and the BOLTLD=n will cause a fatal error. Multiple bolt preload subcases can be defined,

they can be defined as sequentially dependent or nonsequentially dependent, and can optionally include the new bolt preload sequence described below. As a result, you can define bolt preload subcases to apply or remove bolt forces in any sequence.

6. The following rules apply in SOL 401 when BOLTFOR, BOLTFRC, BOLTLD, or BOLTSEQ bulk entries are defined using the same SID.

   - Multiple BOLTFOR and BOLTFRC entries with the same SID can be referenced by the BOLTLD case control command directly, and by the BOLTSEQ and BOLTLD bulk entries. The software will combine and apply them together.

   - If a BOLTSEQ entry is referenced by the BOLTLD case control command, BOLTFOR, BOLTFRC, and BOLTLD entries with the same SID as the BOLTSEQ are only used if they are referenced in a sequence defined on the BOLTSEQ entry.

   - If a BOLTLD bulk entry is referenced by the BOLTLD case control command or by the BOLTSEQ bulk entry, BOLTFOR and BOLTFRC entries with the same SID as the BOLTLD entry are used if they are referenced on the BOLTLD entry.

   - Only one BOLTSEQ entry can be referenced. The software will fatal if multiple BOLTSEQ entries with the same SID are referenced.

   - Only one BOLTLD entry can be referenced. The software will fatal if multiple BOLTLD entries with the same SID are referenced.

REMARK RELATED TO SOL 601:

1. When applying the bolt preload, the software shortens the length of the elements representing the bolts. The warning message "BOLT SHORTENING IS TOO LARGE, THE PROGRAM MIGHT NOT CONVERGE" is produced if the axial shortening of any CBEAM or CBAR element included on the BOLT bulk entry is greater than 0.8 of its initial length.

BOLTRESULTS

Requests bolt results output in SOL 401.

Requests bolt results output in SOL 401.

FORMAT:

$$\text{BOLTRESULTS}[(\text{PRINT}, \text{PUNCH}, \text{PLOT})] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
BOLTRESULTS=ALL
BOLTRESULTS(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. The referenced SET lists the bolt IDs in which output is requested. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. The BOLTRESULTS case control is only supported in SOL 401.

2. The output requested with the BOLTRESULTS case control command includes the axial force, bending moment, the shear forces, and the axial strain for each bolt ID listed on the SET command. The output is computed relative to the coordinate system used to define the bolt axis. The computed results for a bolt are written to each element defining that bolt. For example, if you display the bending moment in a post processor, the bolt will display as a constant.

BOUTPUT

Contact Output Requests

Selects contact regions for output.

FORMAT:

$$\text{BOUTPUT}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
BOUTPUT=ALL
BOUTPUT=5
```

DESCRIBERS:

Describer	Meaning
SORT1	Output is presented as a tabular listing of slave nodes for each load or time depending on the solution sequence.
SORT2	Output is presented as a tabular listing of load or time for each slave node.
PRINT	The print file is the output media.
PUNCH	The punch file is the output media.
PLOT	Generate output histories for slave nodes, but do not print.
ALL	Histories of all the slave nodes listed in all the BOUTPUT Bulk Data entries are output. If no BOUTPUT Bulk Data entries are specified, histories of all the slave nodes in all the contact regions are output.

Describer	Meaning
n	Set identification of previously appearing SET command. Only contact regions with identification numbers that appear on the SET command are selected for output. If there is a BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for slave nodes listed in the Bulk Data entry are output. If there is no BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for all the slave nodes in that contact region are output.
NONE	Result histories for slave nodes are not calculated or output.

REMARKS:

1. BOUTPUT is processed in SOLs 106, 129, 153, and 159 only.
2. SORT1 is the default in SOLs 106 and 153. SORT2 is the default in SOLs 129 and 159.

CKGAP

Gap Result Output Request

Requests gap result output for chocking elements in SOL 401.

FORMAT:

$$\text{CKGAP}\left[\left(\begin{bmatrix}\text{GRID}\\ \text{GAUSS}\\ \text{BOTH}\end{bmatrix},\begin{bmatrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
CKGAP(GAUSS)=ALL
```

DESCRIBERS:

Describer	Meaning
GRID	Requests gap results for corner grids. (Default)
GAUSS	Requests gap results for Gauss points.
BOTH	Requests gap results for corner grids and Gauss points.
PRINT	The printer is the output medium. (Default)
PUNCH	The punch file is the output medium.
PLOT	Generates results for requested set but no printer output.
ALL	Results are output for all chocking elements.
n	Identification number of a SET case control command that lists the chocking elements for which results are output. (Integer>0)
NONE	No results are output for any chocking elements.

REMARKS:

1. The software uses the gap result at the gauss points to determine if the element is chocked or unchocked.

2. Gap results at the grid points is computed using a linear extrapolation of the gap results at the gauss points. Since gap results can vary across the elements, the linear extrapolation can introduce inaccurate gap results at the grid points. This becomes more significant when there is a large variation of gap across an element, which is possible with a coarse mesh.

CLOAD

Static Load Request for Upstream Superelement Loads

Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.

FORMAT:

CLOAD=n

EXAMPLES:

```
CLOAD=15
```

DESCRIBERS:

Describer	Meaning
n	Identification number of a unique CLOAD Bulk Data entry. (Integer>0)

REMARKS:

1. This command may only appear in the residual structure subcases (see “SUPER”) and if used it must be specified in all of them.
2. The CLOAD Bulk Data entry must reference previously processed LSEQ (load sequence) Bulk Data that was requested by LOADSET Case Control commands on the upstream (SUPER ≠ 0) subcases.
3. The resulting load is added to those produced by LOAD and TEMP(LOAD) Case Control commands in the residual structure subcases.

CMETHOD

Complex Eigenvalue Extraction Method Selection

Selects complex eigenvalue extraction parameters.

FORMAT:

CMETHOD=n

EXAMPLES:

```
CMETHOD=77
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of EIGC (or EIGP) bulk entry. (Integer>0)

REMARKS:

1. The CMETHOD command must be specified in order to compute complex eigenvalues.
2. See the description for the parameter, UNSYMF, to perform complex eigenvalue analysis in Solution 106.

CRSTRN

Requests creep strain output at grid points for SOL 401.

Requests creep strain at grid points.

FORMAT:

$$\text{CRSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix}\right\}$$

EXAMPLES:

```
CRSTRN=ALL
CRSTRN(PRINT,PLOT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

CSSCHD

Aerodynamic Control Surface Schedule

Selects control system schedule information.

FORMAT:

CSSCHD=n

EXAMPLES:

```
CSSCHD=10
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a control system schedule that appears on a CSSCHD Bulk Data entry.

REMARKS:

One or more CSSCHD entries can be invoked by this entry.

CYCFORCES

Requests MPC force output at the grid points selected for the automatic coupling in a SOL 401 cyclic symmetry analysis.

Requests MPC force output at the grid points selected for the automatic coupling in a SOL 401 cyclic symmetry analysis.

FORMAT:

$$\text{CYCFORCES}[(\text{PRINT}, \text{PUNCH}, \text{PLOT})] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
CYCFORCES=ALL
CYCFORCES(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. The SET command is a list of grid point IDs. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.

2. The CYCFORCES output only occurs on the grid points included in a coupling source and target region. The grid points listed on the referenced SET command should be included in a coupling region. Grid points listed in a referenced SET which are not in a source or target region will be ignored.

CYCSET

Selects a cyclic symmetric boundary coupling in SOL 401.

Selects a cyclic symmetric boundary coupling in SOL 401.

FORMAT:

CYCSET=n

EXAMPLES:

```
CYCSET=5
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a CYCSET or CYCADD bulk entry. (Integer>0)

REMARKS:

1. n can reference a single CYCSET bulk entry, or it can reference the CYCADD bulk entry which combines multiple CYCSET entries.
2. The CYCSET case control selects the CYCSET or CYCADD bulk entries. The CYCSET case control must be defined above the subcase level. As a result, the MPC's generated by the automatic coupling are used in every subcase (cyclic modes, static, and normal modes subcases).

CZRESULTS

Cohesive Element Output Request

Requests results output for cohesive elements in SOL 401.

FORMAT:

$$\text{CZRESULTS}\left[\left(\text{TRACTION}, \text{RLTDIS}, \text{DAMAGE}, \begin{bmatrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
CZRESULTS(DAMAGE,PUNCH)=100
```

DESCRIBERS:

Describer	Meaning
TRACTION	Requests normal and in-plane traction results. See Remarks 1 and 2. (Default)
RLTDIS	Requests normal and in-plane relative displacement results. See Remarks 1 and 2.
DAMAGE	Requests damage results. See Remark 1.
PRINT	The printer is the output medium. (Default)
PUNCH	The punch file is the output medium.
PLOT	Generates results for requested set but no printer output.
ALL	Results are output for all cohesive elements.
n	Identification number of a SET case control command that lists the cohesive elements for which results are output. (Integer>0)
NONE	No results are output for any cohesive elements.

REMARKS:

1. All results are reported at the corner grid points.

2. Four values are reported for tractions and relative displacements. One value is the magnitude of the traction or relative displacement in the direction normal to the element. The other three values are the components relative to the basic coordinate system of the traction or relative displacement in the plane of the element. Tractions have units of force per unit area. Relative deformation has units of length.

DEFORM

Element Deformation Static Load

Selects the element deformation set.

FORMAT:

DEFORM=n

EXAMPLES:

```
DEFORM=27
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of DEFORM Bulk Data entries. (Integer>0)

REMARKS:

1. DEFORM Bulk Data entries will not be used unless selected by the DEFORM command in the Case Control Section.
2. DEFORM is only applicable in linear statics, inertia relief, differential stiffness, and buckling problems (SOLs 101, 105, 114, and 200), and will produce a fatal message in other solution sequences.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM) and constrained displacement loads (SPC, SPCD).
4. Static, thermal, and element deformation loads should have unique identification numbers.

5. In the superelement solution sequences, if this command is used in a cold start, it must also be specified in the restart.

DESGLB

Request Design Constraints at the Global Level

Selects the design constraints to be applied at the global level in a design optimization task.

FORMAT:

DESGLB=n

EXAMPLES:

```
DESGLB=10
DESG=25
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a set of DCONSTR or a DCONADD Bulk Data entry identification number. (Integer>0)

REMARKS:

1. If used, this command must occur before the first SUBCASE.
2. A DESGLB command is optional. It should be used to invoke either constraints on responses that are independent of a particular subcase (such as WEIGHT or VOLUME), or constraints that involve a combination of responses from various subcases (see also DRSPAN Case Control command).
3. DESGLB constraints on DRESP2 or DRESP3 responses that are independent of DRESP1 responses are also acceptable.
4. When DESGLB constraints refer to second level responses (DRESP2 and/or DRESP3) that are based on combinations of DRESP1 responses from more than one subcase, each such DRESP1 must have been assigned to a particular

subcase by way of the DRSPAN Case Control command. Such DESGLB constraints are output for the last analyzed subcase associated with the second level response in question.

DESOBJ

Design Objective

Selects the DRESP1, DRESP2, or DRESP3 entry to be used as the design objective.

FORMAT:

$$\text{DESOBJ}\left[\left(\begin{array}{c}\text{MAX}\\ \text{MIN}\end{array}\right)\right] = n$$

EXAMPLES:

```
DESOBJ=10
DESO=25
```

DESCRIBERS:

Describer	Meaning
MIN	Specifies that the objective is to be minimized.
MAX	Specifies that the objective is to be maximized.
n	Set identification of a DRESP1, DRESP2, or DRESP3 Bulk Data entry. (Integer>0)

REMARKS:

1. A DESOBJ command is required for a design optimization task and is optional for a sensitivity task. No more than one DESOBJ may appear in Case Control.

2. If the DESOBJ command is specified within a SUBCASE, the identified DRESPi Bulk Data entry uses a response only from that subcase. If DESOBJ appears above all SUBCASE commands and there are multiple subcases, it uses a global response (See also Remarks 5 and 6).

3. A DRESPi (i=1, 2, 3) entry referenced as an objective cannot, in general, represent multiple responses. For example, DRESP1 responses which include multiple grids in the ATTi field or multiple frequencies in the ATTB field cannot be referenced. If DESOBJ references such a DRESPi, either by mistake or intentionally, the very first response in the represented set of responses will be taken as the objective function.

4. The minimization or maximization is on the algebraic value of the objective function. Thus, care must be exercised if negative values are expected or if there is a possibility for the objective function to change sign during optimization. For example, to minimize the absolute value of a displacement component, the absolute value or the square of the relevant response may need to be specified with a DRESP2 response that references the DRESP1 for that particular displacement component. If it is possible for the objective function to change sign, as in the case of a displacement component, the change of sign can cause the objective function to move in an undesirable direction. When such a possibility exists, it is best to place appropriate constraints on the value of the response that is used for the objective function.

5. A global response, as mentioned in Remark 2, may be either a subcase independent response or one which includes a combination of DRESP1 responses from more than one subcase (see also DRSPAN Case Control command, and the Remarks for DESGLB).

6. When the global response includes a combination of DRESP1 responses from more than one subcase, each such DRESP1 response must have been assigned to a particular subcase by way of the DRSPAN Case Control command. A DESOBJ referencing such a global response is output for the last analyzed subcase associated with the second level response in question.

7. When the design is infeasible due to violated constraints, the objective function may move in the direction opposite to that desired in an attempt to reach the feasible region.

8. The best design found is not necessarily the one in the last design cycle or the one with the lowest value of the objective function. If there are one or more feasible designs (very small positive within specified or default tolerance, or negative maximum normalized constraint values), then the best design is the feasible design with the lowest value of the objective function. If there are no feasible designs (all positive non-trivial values of the maximum normalized constraint), then the best infeasible design found is that with the smallest value of the maximum normalized constraint, without regard for the value of the objective function.

DESSUB

Design Constraints Request at the Subcase Level

Selects the design constraints to be used in a design optimization task for the current subcase.

FORMAT:

DESSUB=n

EXAMPLES:

```
DESSUB=10
DESS=25
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a set of DCONSTR and/or a DCONADD Bulk Data entry identification number. (Integer ≥ 0)

REMARKS:

1. A DESSUB command is required for every subcase for which constraints are to be applied. An exception to this is ‘global constraints’, which are selected by the DESGLB command.
2. All DCONSTR and DCONADD Bulk Data entries with the selected set ID will be used.
3. A DESSUB command appearing before all subcases is applicable for all subcases except for those subcases where a different DESSUB command appears.

DISPLACEMENT

Displacement Output Request

Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.

FORMAT:

$$\text{DISPLACEMENT}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{ABS}\\ \text{REL}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
DISPLACEMENT=5
DISPLACEMENT(REAL)=ALL
DISPLACEMENT(SORT2, PUNCH, REAL)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of load, frequency or time for each grid point.
PRINT	The printer will be the output medium.

Describer	Meaning
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, displacement data.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ABS	For enforced motion dynamic analysis, displacement results will be output as absolute displacement.
REL	For enforced motion dynamic analysis, displacement results will be output relative to the enforced motion input.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 8.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 8.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 8.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 8.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 8.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 8.

Describer	Meaning
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 8.
RPUNCH	Writes random analysis results to the punch file. See Remark 8.
ALL	Displacements for all points will be output.
NONE	Displacement for no points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. VECTOR and PRESSURE are alternate forms and are entirely equivalent to DISPLACEMENT.

4. DISPLACEMENT = NONE overrides an overall output request.

5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer output request is present for magnitude/phase representation.

6. The units of translation are the same as the units of length of the model. Rotations are in units of radians.

7. Displacement results are printed and/or punched in the global coordinate system (see field CD on the GRID bulk data entry). The coordinate system for plotted displacement output depends on the PARAM,POST setting. See the parameter POST.

8. The following applies to random solutions:

   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

   - Autocorrelation (ATOC) calculations require the RANDT1 bulk entry.

9. When doing enforced motion dynamic analysis and relative output is requested (using the REL describer), the output will be relative to the input as described by the equation:

$$u_f = y_f - K_{ff}^{-1} K_{fs} u_s$$

where u_f = absolute displacement

y_f = relative displacement

u_s = enforced motion.

REMARKS RELATED TO SOLS 601 AND 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC, RMS, and RALL are ignored.

2. Displacements, velocities and accelerations must be output for the same set of grid points if requested. Output requested for set n in this command will be combined with the sets requested in the VELOCITY and ACCELERATION commands, and displacements will be output at the grid points of the combined set.

DIVERG

Static Aeroelastic Divergence Request

Selects the divergence parameters in a static aeroelastic divergence problem.

FORMAT:

DIVERG=n

EXAMPLES:

```
DIVERG=70
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a DIVERG Bulk Data entry. (Integer>0)

REMARKS:

Static aeroelastic divergence analysis can be performed in SOLs 144 and 200.

DLOAD

Dynamic Load Set Selection

Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.

For SOLs 401, 601 and 701, DLOAD selects a time-dependent load for nonlinear analysis.

FORMAT:

DLOAD=n

EXAMPLES:

```
DLOAD=73
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE Bulk Data entry. (Integer>0)
n (SOLs 401, 601, or 701)	Set identification of a DLOAD or TLOAD1 Bulk Data entry. (Integer>0)

REMARKS RELATED TO TRANSIENT AND FREQUENCY RESPONSE:

1. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.
2. TLOAD1 and TLOAD2 may be selected in a transient or frequency response problem.
3. Either a RLOADi or TLOADi entry (but not both) must be selected in an aeroelastic response problem. If RLOADi is selected, a frequency response is calculated. If TLOADi is selected, then transient response is computed by Fourier transform.

When there are only gust loads (GUST entry), the DLOAD selects a TLOADi or RLOADi entry with zero load along with field 3 of the GUST command.

4. The DLOAD command will be ignored if specified for upstream superelements in dynamic analysis. To apply loads to upstream superelements, please see the “LOADSET” .

REMARKS RELATED TO SOLS 601 AND 701:

1. In SOLs 601 and 701, DLOAD may be used in a static or transient analysis.

2. Both the DLOAD and LOAD case control commands can be specified in a SOLs 601 and 701 analysis. DLOAD is used to select time-dependent loads and LOAD is used to select constant loads.

3. See Section 5.1 in the *Advanced Nonlinear Theory and Modeling Guide* on defining time-dependent loads.

DRSPAN

Assigns DRESP1 Responses to a Specific Subcase

Assigns a set of DRESP1 responses to a specific subcase, by referencing an existing SET. This set should then contain the list of the DRESP1 to be assigned to the subcase for which the DRSPAN data references the set.

FORMAT:

DRSPAN = SETID

EXAMPLE:

```
DRSPAN=375
```

DESCRIBERS:

Describer	Meaning
SETID	The ID number of a previously defined SET containing the list of DRESP1 responses that should be assigned to the subcase within which this DRSPAN data resides.

REMARKS:

1. A DRESP1 response assigned to a particular subcase may not be assigned to another subcase.
2. A DRSPAN referenced SET may not be referenced by another DRSPAN command.
3. A DRSPAN referenced SET may not refer to ALL.
4. A DRESP2 or DRESP3 response that references DRSPAN based DRESP1 responses, may not contain non-DRSPAN based DRESP1.

5. A DRESP2 or DRESP3 response that references DRSPAN based DRESP1 responses may not be nested within another DRESP2 or DRESP3, or have nested DRESP2. If nesting is done with DRSPAN, the job may produce unreliable results if it does not fail. See Remark 6 below for an exception.

6. One exception to Remark 5 above is the case of referenced (nested) DRESP2, which are actually DRESP1 that are mathematical functions (integrated responses) defined in Remark 20 for DRESP1. In accordance with Remark 20 of DRESP1, these responses should be referred to as DRESP2 from a referencing DRESP2 (should not be referenced from a DRESP3). Such responses actually being DRESP1, however, may therefore be specified in DRSPAN sets.

7. An objective function referencing a DRSPAN related second level response (DRESP2 or DRESP3) should be above the subcase level.

8. A constraint referencing a DRSPAN related second level response (DRESP2 or DRESP3) should be referenced by a DESGLB command, and therefore should be above the subcase level.

9. The DRSPAN command can be used with superelements, as long as all DRESP1 responses in a DRSPAN related DRESP2 or DRESP3 are from the same superelement.

DSAPRT

Design Sensitivity Output Parameters

Specifies design sensitivity output parameters.

FORMAT:

$$\text{DSAPRT}\left[\left(\left[\begin{array}{c}\text{FORMATTED}\\ \text{UNFORMATTED}\\ \text{NOPRINT}\end{array}\right],\left[\begin{array}{c}\text{NOEXPORT}\\ \textit{EXPORT}\end{array}\right],\right.\right.$$
$$\left.\left.[\text{START} = \text{i}], [\text{BY} = \text{y}], [\text{END} = \text{k}]\right)\right] = \left[\begin{array}{c}\text{ALL}\\ \text{n}\\ \text{NONE}\end{array}\right]$$

EXAMPLES:

```
DSAPRT(FORMATTED,EXPORT)
DSAPRT(FORMATTED,START=FIRST,BY=3,END=LAST)=101
DSAPRT(UNFORMATTED,START=FIRST)
DSAPRT(UNFORMATTED,EXPORT)
DSAPRT(FORMATTED,END=4)=ALL
DSAPRT(UNFORMATTED,END=SENS)=ALL
DSAPRT(NOPRINT, EXPORT)
```

DESCRIBERS:

Describer	Meaning
FORMATTED	Output will be presented with headings and labels.
UNFORMATTED	Output will be printed as a matrix print (see the description for the MATPRN module in the *NX Nastran DMAP Programmer's Guide.*).
NOPRINT	No output will be printed.
EXPORT	Output will be exported to an external binary file specified by PARAM,IUNIT.

Describer	Meaning
NOEXPORT	Output will not be exported to an external binary file.
START=i	Specifies the first design cycle for output. (Integer>0 or Character: “FIRST” or “LAST”; Default = 1 or “FIRST”)
BY=j	Specifies the design cycle interval for output. (Integer ≥ 1, or >0; Default = 0) See Remark 2 .
END=k	Specifies the last design cycle for output. (Integer>0 or Character: “FIRST”, “LAST”, or “SENS”; Default = “LAST”)
ALL	All design responses (defined in DRESP1, DRESP2, and DRESP3 entries) will be output.
n	Set identification of a previously appearing SET command. Only sensitivities of responses with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Only one DSAPRT may appear in the Case Control Section and should appear above all SUBCASE commands.

2. Sensitivity data will be output at design cycles i, i+j, i+2j, ..., k. Note that the BY=0 default implies no sensitivity analysis at the intermediate design cycles.

3. END = SENS requests design sensitivity analysis, and no optimization will be performed.

4. If both DSAPRT and PARAM,OPTEXIT, 4, -4, or 7 are specified then DSAPRT overrides PARAM,OPTEXIT, 4, -4, or 7. PARAM,OPTEXIT values and the equivalent DSAPRT commands are as follows:

OPTEXIT	Equivalent DSAPRT Command
4	DSAPRT(UNFORMATTED, END=SENS)
-4	DSAPRT(NOPRINT, EXPORT, END=SENS)
7	DSAPRT(UNFORMATTED, START=LAST)

5. Formatted sensitivity output is not supported for aeroelastic response types (DRESP1 RTYPE=TRIM, STABDER or FLUTTER). Unformatted output must be requested.

DSYM

Dihedral Symmetry Option in Cyclic Symmetry

Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.

FORMAT:

$$\text{DSYM} = \left\{ \begin{array}{c} \text{S} \\ \text{A} \\ \text{SS} \\ \text{SA} \\ \text{AS} \\ \text{AA} \end{array} \right\}$$

EXAMPLES:

```
DSYM=AS
```

DESCRIBERS:

Describer	Meaning
S, A	The problem is assumed to be symmetrical (S) or antisymmetrical (A) with respect to the plane containing Side 1 of segment 1.
SS, SA, AS, AA	The problem is assumed to be symmetrical (or antisymmetrical) with respect to the plane containing Side 1 of segment 1 (denoted by the first symbol), and also with respect to a plane perpendicular to Side 1 (denoted by the second symbol).

DTEMP

Time-assigned temperature set selection for SOL 401, 601 and 701.

Selects a time-assigned temperature set to be used for temperature dependent material properties and thermal loading.

FORMAT:

DTEMP =n

EXAMPLES:

```
DTEMP=5
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of DTEMP or DTEMPEX bulk entries. (Integer > 0)

REMARKS:

1. Only supported in static subcases for SOL 401.
2. If both DTEMP and TEMP(LOAD) are specified, the last one defined takes precedence. For example, if you define the TEMP(LOAD) command globally, and you define the DTEMP command in a subcase, the DTEMP command is used for that subcase.

REMARKS RELATED TO SOLS 601 AND 701:

1. The set identification number must be of a DTEMP bulk entry. The DTEMPEX bulk entry is not supported.

2. DTEMP and TEMP(LOAD) cannot be specified in the same model.

EBDSET

Element Birth/Death Set Selection

Selects the element birth/death set (SOLs 601 and 701)

FORMAT:

EBDSET=n

EXAMPLES:

```
EBDSET=5
```

DESCRIBERS:

Describer	Meaning
n	Element birth/death set identification of an EBDSET or EBDADD Bulk Data entry. (Integer>0)

ECHO

Bulk Data Echo Request

Controls echo (i.e., printout) of the Bulk Data.

FORMAT:

$$\text{ECHO}=\left\{\begin{array}{l}\left[\text{UNSORT}\left[\begin{array}{c}\text{SORT}[(\,[\text{EXCEPT}]\text{cdni},.)]\\ \text{NOSORT}\end{array}\right]\right.,\text{PUNCH}\left[\left(\begin{array}{c}\text{BSTBULK}\\ \text{NEWBULK}\end{array}\right)\right]\\ \text{BOTH}\\ \text{NONE}\end{array}\right\}$$

EXAMPLES:

```
ECHO=NOSORT
ECHO=BOTH
ECHO=PUNCH,SORT(MAT1,PARAM)
ECHO=SORT(EXCEPT DMI,DMIG)
ECHO=PUNCH(BSTBULK)
ECHO=PUNCH(NEWBULK)
ECHO=SORT,PUNCH(BSTBULK)
```

DESCRIBERS:

Describer	Meaning
UNSORT	The unsorted Bulk Data will be printed. If SORT is also not specified then the sorted Bulk Data will not be printed.
SORT	The sorted (arranged in alphanumeric order) Bulk Data will be printed.
NOSORT	The sorted Bulk Data will not be printed. If UNSORT is also not specified then the unsorted Bulk Data will not be printed.

Describer	Meaning
cdni,...	Bulk Data entry names to be included or excluded by EXCEPT, in the sorted echo printout. The PUNCH file is not affected by cdni.
EXCEPT	Exclude cdni Bulk Data entries from sorted echo printout. See Remark 6.
BOTH	Both sorted and unsorted Bulk Data will be printed. This is equivalent to ECHO=SORT, UNSORT.
NONE	Neither sorted nor unsorted Bulk Data will be printed.
PUNCH	The entire Bulk Data will be written to the punch file in sorted form.
BSTBULK	For SOL 200 design optimization runs, an unsorted updated bulk data file will be written to the punch file, using the updated design variables and properties as well as any mode-tracked eigenvalue DRESP1 (i.e. the updated design data) for the BEST design cycle obtained and indicated in the SOL 200 run. See Remark 7.
NEWBULK	For SOL 200 design optimization runs, an unsorted updated bulk data file will be written to the punch file, using the updated design variables and properties as well as any mode-tracked eigenvalue DRESP1 (i.e. the updated design data) for the LAST design cycle obtained and indicated in the SOL 200 run. See Remark 7.

REMARKS:

1. If no ECHO command appears, a sorted Bulk Data will be printed.
2. Comments will appear at the front of the sorted file if ECHO=PUNCH.
3. Portions of the unsorted Bulk Data can be selectively echoed by including the commands ECHOON and ECHOOFF at various places within the Bulk Data. ECHOOFF stops the unsorted echo until an ECHOON command is encountered. Many such pairs of commands may be used. The ECHOON and ECHOOFF commands may be used in the Executive and Case Control Sections; however, ECHOOF should not be the first entry as continuation entries will not be handled correctly.
4. If the SORT (cdni,...) is specified in a restart in SOLs 101 through 200, then the continuation entries will not be printed.

5. If the SORT (cdni,...) describer is used, then it must appear as the last describer, as in the example above.

6. If EXCEPT is specified then it must be specified before all cdni. All Bulk Data entry types will be listed except those given for cdn1, cdn2, etc. If EXCEPT is not specified, then only those Bulk Data entry types listed under cdn1, cdn2, etc. will be listed.

7. For SOL 200, often the last design cycle and the best design cycle coincide. However, BSTBULK is useful for cases where the best design cycle is actually an earlier cycle. In contrast, NEWBULK may be useful when the user knows that the job will be continued later starting from the last design cycle.

EDE

Element Energy Loss Per Cycle Output Request

Requests the output of the energy loss per cycle in selected elements.

FORMAT:

$$\text{EDE}\left[\left(\left[\begin{matrix}(\text{PRINT}, \text{PUNCH})\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right]\left[\begin{matrix}\text{RMAG}\\ \text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right][\text{THRESH} = \text{p}]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\end{matrix}\right\}$$

EXAMPLES:

```
EDE=ALL
EDE(PUNCH, THRESH=.0001)=19
```

DESCRIBERS:

Describer	Meaning
PRINT	Write energies to the print file (default).
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy in frequency response analysis only.
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.

Describer	Meaning
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
RMAG	Outputs the energy magnitude in real data format.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
THRESH	Suppresses energies for elements having an energy value of less than p% in all output files. THRESH overrides the value of TINY described in Remark 1. (Default=0.001)
ALL	Computes energy for all elements.
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EDE command or above all subcases. (Integer>0)
NONE	Element energy will not be output.

REMARKS:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.

4. The equations used to calculate elemental damping energy components are given below.

   Average Damping Energy:

$$E_{avg} = \Pi\omega\left[\{u_r\}^T[B_e]\{u_r\} + \{u_i\}^T[B_e]\{u_i\}\right]$$

   where:

E	=	elemental energy
$\{u_r\}$	=	displacement - real part
$\{u_i\}$	=	displacement - imaginary part
$[B_e]$	=	elemental damping

   Real Part of Damping Energy Amplitude:

$$E_{amp_real} = \Pi\omega\left[\{u\}^T[B_e]\{u_r\} - \{u_i\}^T[B_e]\{u_i\}\right]$$

   Imaginary Part of Damping Energy Amplitude:

$$E_{amp_imag} = -2\Pi\omega\{u_r\}^T[B_e]\{u_i\}$$

   Magnitude of Damping Energy Amplitude:

$$E_{amp_mag} = \sqrt{E_{amp_real}^2 + E_{amp_imag}^2}$$

   Phase of Damping Energy Amplitude:

$$\theta_{amp} = \arctan\left(\frac{E_{amp_imag}}{E_{amp_real}}\right)$$

   Peak Damping Energy:

$$E_{peak} = E_{avg} + E_{amp_mag}$$

5. In SOL 111, EDE can only be requested if PARAM,DDRMM,-1 is used.

6. Only damping from the viscous dampers (e.g., CVISC, CDAMPi, etc.) are included. Structural damping is not included in the calculation.

EFLOAD

External Field Surface Loading

Converts surface loads from an external field solution to structural loads.

FORMAT:

$$\text{EFLOAD}\left[\text{NDIR}=n,\text{SCID}=m\right]=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}$$

EXAMPLES:

```
EFLOAD=YES
EFLOAD(NDIR=2)=YES
EFLOAD(NDIR=3,SCID=14)=Yes
EFLOAD(SCID=7)=YES
```

DESCRIBERS:

Describer	Meaning
NDIR=*n*	*n* (integer>0) specifies the normal direction of the surface patch defined in the same subcase. Allowable values for *n* are 1, 2, and 3. See Remarks 2 and 3.
SCID=*m*	*m* (integer>0) specifies the identification number of a local coordinate system. See Remarks 4 and 5.
YES	Convert external field surface loads to structural loads.
NO	Do not convert external field surface loads to structural loads.

REMARKS:

1. A subcase that includes an EFLOAD case control command is required for each surface patch that external field surface loads are converted to structural loads. The last subcase cannot contain an EFLOAD case control command, but only output requests and other structural loads (if any).

2. The table lists the allowable values for *n* and the corresponding normal directions.

n	Rectangular CSYS	Cylindrical CSYS
1 (default)	z	z
2	y	θ
3	x	r

3. If NDIR=*n* is not specified, NDIR=1 is used by default.

4. If SCID=*m* is not specified, the normal direction to the surface patch and the coordinates in the corresponding external force points matrix are defined relative to the basic coordinate system.

5. If SCID=*m* is specified, the normal direction to the surface patch and the coordinates in the corresponding external force points matrix are defined relative to a cylindrical coordinate system whose:

   - Origin coincides with the basic coordinate system.

   - Cylindrical axis coincides with the z-axis of the basic coordinate system.

   - Tangential coordinate, θ, is measured from the x-axis of the basic coordinate system. ($0 \le \theta \le 360$ degrees)

   The local coordinate system with *m* as the identification number does not need to be cylindrical. By specifying any local coordinate system, the cylindrical coordinate system described above is used as the coordinate basis.

6. For a direct frequency response (SOL 108) solution, the structural loads converted from the external field surface loads are used as scaling factor components of the dynamic load.

7. For a statically prestressed normal modes analysis, two runs are required. The first run is a linear static (SOL 101) solution that converts the external field surface loads to structural loads. During this run, a punch (.pch) file is automatically written with the converted external field surface loads. The second run is a normal modes (SOL 103) solution. For this run, the first subcase, which is the static subcase, must include EFLOAD=YES. The second subcase, which is the

normal modes subcase, must include STATSUB=1. The bulk data must include an INCLUDE bulk entry that references the .pch file written during the first run.

8. For additional information on how to use the EFLOAD case control command to convert surface loads from an external field solution to structural loads, see External Force Fields in the *NX Nastran User's Guide*.

9. An external force field can be included in a rotor dynamic analysis with the direct complex eigenvalue solution sequence (SOL 107). Three solution steps are required, and are described in External Force Fields in the *NX Nastran User's Guide*.

EKE

Element Kinetic Energy Output Request

Requests the output of the kinetic energy in selected elements.

FORMAT:

$$\text{EKE}\left[\left(\left[\begin{matrix}(\text{PRINT},\text{PUNCH})\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right]\left[\begin{matrix}\text{RMAG}\\ \text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right][\text{THRESH} = \text{p}]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\end{matrix}\right\}$$

EXAMPLES:

```
EKE=ALL
EKE(PUNCH, THRESH=.0001)=19
```

DESCRIBERS:

Describer	Meaning
PRINT	Write energies to the print file (default).
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy in frequency response analysis only.
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.

Describer	Meaning
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
RMAG	Outputs the energy magnitude in real data format.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
THRESH	Suppresses energies for elements having an energy value of less than p% in all output files. THRESH overrides the value of TINY as described in the remarks. (Default=0.001)
ALL	Computes energy for all elements.
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EKE command or above all subcases. (Integer>0)
NONE	Element energy will not be output.

REMARKS:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.

4. The equations used to calculate elemental kinetic energy components are given below.

   Average Kinetic Energy:

$$E_{avg} = \frac{1}{4}\left[\{v_r\}^T[M_e]\{v_r\} + \{v_i\}^T[M_e]\{v_i\}\right]$$

   where:

E	=	elemental energy
$\{v_r\}$	=	velocity - real part
$\{v_i\}$	=	velocity - imaginary part
$[M_e]$	=	elemental mass

   Real Part of Kinetic Energy Amplitude:

$$E_{amp_real} = \frac{1}{4}\left[\{v_r\}^T[M_e]\{v_r\} - \{v_i\}^T[M_e]\{v_i\}\right]$$

   Imaginary Part of Kinetic Energy Amplitude:

$$E_{amp_imag} = -\frac{1}{2}\{v_r\}^T[M_e]\{v_i\}$$

   Magnitude of Kinetic Energy Amplitude:

$$E_{amp_mag} = \sqrt{E_{amp_real}^2 + E_{amp_imag}^2}$$

   Phase of Kinetic Energy Amplitude:

$$\theta_{amp} = \arctan\left(\frac{E_{amp_imag}}{E_{amp_real}}\right)$$

   Peak Kinetic Energy:

$$E_{peak} = E_{avg} + E_{amp_mag}$$

5. In SOL 111, EKE can only be requested if PARAM,DDRMM,-1 is used.

ELSDCON

Element Stress Discontinuity Output Request

Requests mesh stress discontinuities based on element stresses.

FORMAT:

$$\text{ELSDCON}\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix} = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
ELSDCON=ALL
ELSDCON=19
```

DESCRIBERS:

Describer	Meaning
ALL	Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request. (Integer>0)
NONE	No element stress discontinuity output.

REMARKS:

1. This output is available in linear static analysis in SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.

2. Only elements used to define the surface or volume are output. See the descriptions of “SURFACE” and “VOLUME” .

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS command must be present for printed output and the STRFIELD command for postprocessing output using the .xdb file (PARAM,POST,0) for the same surfaces and volumes.

ELSTRN

Requests elastic strain output at grid points on elements for SOL 401. Also requests elastic strain output at grid points on composite shell elements for SOLs 101 and 106.

Requests elastic strain at grid points on elements.

FORMAT:

$$\text{ELSTRN}\left[\left(\left[\begin{matrix}\text{STRCUR}\\ \text{FIBER}\end{matrix}\right],\left[\begin{matrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{matrix}\right],\ \text{PRINT, PUNCH, PLOT}\right)\right]=\left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
ELSTRN=ALL
ELSTRN(PRINT,PUNCH)=17
The following example is only supported in SOLs 101 and 106:
ELSTRN(STRCUR,CORNER,PRINT) = ALL
```

DESCRIBERS FOR SOL 401:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

DESCRIBERS FOR SOLS 101 AND 106:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.
STRCUR	Elastic strain at the reference plane and curvatures is output for plate elements.
FIBER	Elastic strain at locations Z1, Z2 is computed for plate elements.
CENTER	Outputs elastic strains at the center of the element only. See Remark 2 for SOLs 101 and 106.
CORNER or BILIN	Outputs elastic strains at the center and corner grid points only using extrapolation. See Remark 3 for SOLs 101 and 106.
SGAGE	Outputs elastic strains at the center and corner grid points using strain gage approach. See Remark 4 for SOLs 101 and 106.
CUBIC	Outputs elastic strains at the center and corner grid points using cubic bending correction. See Remark 5 for SOLs 101 and 106.

REMARKS FOR SOL 401:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

REMARKS FOR SOLS 101 AND 106:

1. The STRCUR option is default for shell elements that reference a PCOMP or PCOMPG entry. Any FIBER option on STRAIN or ELSTRN or THSTRN will override STRCUR option for all STAIN and ELSTRN and THSTRN.
2. For the CENTER option, elastic strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element.
3. For the CORNER (or BILIN) option, elastic strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry.
4. For the SGAGE option, elastic strains are output at the center and grid points for CQUAD4 elements that reference a PCOMP or PCOMPG entry.
5. For the CUBIC option, elastic strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PCOMP or PCOMPG entry.
6. The CENTER/CORNER(BILIN)/SGAGE/CUBIC option on STRAIN card overrides the corresponding option on ELSTRN and THSTRN card.

ELSUM

Element Summary Output Request

Requests output of an element property summary.

FORMAT:

$$\text{ELSUM}[(\text{EID},\text{PID},\text{BOTH},\text{EIDSUM},\text{PIDSUM},\text{SUMMARY},\text{NSMCONT})] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
ELSUM = ALL
ELSUM(PID) = 9
ELSUM(EIDSUM) = ALL
```

DESCRIBERS:

Describer	Meaning
EID	Group element summary output by element type. (Default)
PID	Group element summary output by element property type.
BOTH	Output element summaries for both the EID and PID groupings.
EIDSUM	Output only a summary of mass totals for the EID grouping.
PIDSUM	Output only a summary of mass totals for the PID grouping.
SUMMARY	Output only a summary of mass totals for both the EID and PID groupings.
NSMCONT	Add non-structural mass from NSM, NSM1, NSML, and NMSL1 bulk entries to the EID and/or PID element summaries.

Describer	Meaning
ALL	Element summary output for all elements.
n	Set identification of a previously appearing SET command. Only produces output for elements whose identification numbers appear on this SET command. (Integer>0)
NONE	No element summary output.

REMARKS:

1. The ELSUM command is ignored in heat transfer solutions.
2. ELSUM output is only available in the F06 file. PUNCH and PLOT options are not supported.
3. The ELSUM command produces a summary of properties for elements. The properties include element-id, material-id, length or thickness, area, volume, structural mass, non-structural mass, total mass, and the product of the total mass and the WTMASS parameter. Total mass is the sum of structural and non-structural mass.
4. For elements referencing temperature dependent materials defined with MATTi entries, the ELSUM output is approximated using temperature=0.0.
5. Certain element types produce only partial data. For these element types, output will contain element-id, length or thickness, volume, and area without mass data. Totals will thus not include those elements for which mass data was not generated.
6. Mass data will be computed for the following elements: CBAR, CBEAM, CBEND, CBUSH1D, CHEXA, CMASSi, CONMi, CONROD, CPENTA, CPLSTNi, CPLSTSi, CPYRAM, CQUAD4, CQUAD8, CQUADR, CQUADX4, CQUADX8, CROD, CSHEAR, CTETRA, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, CTUBE.
7. EIDSUM takes precedence over EID if both describers are used.
8. PIDSUM takes precedence over PID if both describers are used.
9. If the NSMCONT describer is specified in combination with the EID describer, a table is produced that identifies the contribution of each non-structural mass bulk entry to the total element non-structural mass. If the NSMCONT describer is specified in combination with the PID describer, a table is produced that identifies non-structural mass from all NSM type bulk entries for all element property types by property identification number.

REMARKS RELATED TO SOLS 601 AND 701:

1. NSM will be zero in the summary tables.

ENTHALPY

Heat Transfer Enthalpy Output Request

Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).

FORMAT:

$$\text{ENTHALPY}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
ENTHALPY=5
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates but does not print enthalpies.
ALL	Enthalpy for all points will be output.
NONE	Enthalpy for no points will be output.

ENTHALPY

Describer	Meaning
n	Set identification of previously appearing SET command. Only enthalpies of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

ENTHALPY=NONE is used to override a previous ENTHALPY=n or ENTHALPY=ALL command.

ERP

Equivalent Radiated Power Output Request

Requests the output of equivalent radiated power for selected panels and the shell elements comprising the selected panels in frequency response solutions 108 and 111, and in the equivalent ANALYSIS=DFREQ (or MFREQ) options with solution 200.

FORMAT:

$$\text{ERP}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right],\ \text{PRINT},\ \text{PUNCH},\ \text{PLOT},\ \text{SOLUTION}=\left\{\begin{matrix}\text{ALL}\\ \text{setf}\end{matrix}\right\},\ \text{ERPCOEFF}=c,\ \text{ELEMOUT}=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}\right)\right]=\left\{\begin{matrix}\text{ALL}\\ \text{setp}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
ERP(PRINT,SOLUTION=53,ERPCOEFF=0.75)=55
```

DESCRIBERS:

Describer	Meaning
SORT1	Present equivalent radiated power output as a tabular listing at individual frequencies sorted from highest to lowest absolute equivalent radiated power. (Default) See Remark 3.
SORT2	Present equivalent radiated power output as a tabular listing sorted according to cumulative total equivalent radiated power over all frequencies. See Remark 3.
PRINT	Write output to the print file. (Default)

Describer	Meaning
PUNCH	Write output to the punch file.
PLOT	Do not write ERP data to either the print or punch file.
SOLUTION = ALL	Calculate equivalent radiated power at all frequencies specified by FREQUENCY case control commands.
SOLUTION = setf	Calculate equivalent radiated power at all frequencies specified by a SET case control command having the identification number setf. (Integer > 0)
ERPCOEFF = c	Coefficient used to scale the equivalent radiated power. See Remark 3. (Real > 0.0; Default = 1.0)
ELEMOUT = YES	Output equivalent radiated power by element for elements that comprise the selected panels. See Remark 4.
ELEMOUT = NO	Do not output equivalent radiated power by element for elements that comprise the selected panels.
ALL	Output equivalent radiated power for all panels.
setp	Output equivalent radiated power for selected panels. *setp* selects the set identification of a previously appearing SET case control command. The SET command lists the names (character entry) of panels defined on a PANEL bulk entry. The SET command must be specified in the same subcase as the ERP command or above all subcases. For ERP output requests, the panels defined on the PANEL bulk entry must reference SET3 bulk entries only. (Integer > 0)
NONE	Do not output equivalent radiated power.

REMARKS:

1. Equivalent radiated power calculation is currently supported for shell elements only. These elements include CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, and CTRIAR.

2. ERP output is supported in the frequency response solutions 108 and 111. You can request ERP output for models that contain only structural elements, or models that include both structural and fluid elements.

3. The equivalent radiated power attributable to the ith element at frequency ω is given by:

$$ERPE_i(\omega) = \frac{1}{2} c \int_{S_i} v_n(\mathbf{x},\omega)\, v_n^*(\mathbf{x},\omega)\, ds$$

where:

c is the scaling coefficient specified using the ERPCOEFF describer. Usually taken to be the product of density and speed of sound in the fluid medium.

$v_n(x,\omega)$ is the normal velocity as a function of position at frequency ω.

$v_n^*(x,\omega)$ is the complex conjugate of the normal velocity as a function of position at frequency ω.

S_i is the surface area of the ith element.

The absolute equivalent radiated power for the jth panel at frequency ω is the sum of the equivalent radiated power for each element comprising the jth panel at frequency ω and is given by:

$$ERPP_j(\omega) = \sum_i ERPE_i(\omega)$$

The total equivalent radiated power from all panels at frequency ω is the sum of all the panel contributions at frequency ω and is given by:

$$ERPT(\omega) = \sum_j ERPP_j(\omega)$$

If the SORT1 (default) describer is specified, the normalized equivalent radiated power for the jth panel at frequency ω is computed as follows:

$$\overline{ERPP}_j(\omega) = \frac{ERPP_j(\omega)}{ERPT(\omega)}$$

The output is then grouped according to frequency and sorted by panel in descending order of equivalent radiated power.

If the SORT2 describer is specified, the normalized equivalent radiated power for the jth panel at frequency ω is computed as follows:

$$\overline{ERPP}_j(\omega) = \frac{ERPP_j(\omega)}{\sqrt{\frac{1}{N}\sum_{k=1}^{N}\left(ERPT(\omega_k)\right)^2}}$$

where the total equivalent radiated power is summed over all (*N*) frequencies specified by the SOLUTION describer. The output is sorted from highest to lowest according to:

$$\overline{\overline{ERPP}}_j = \frac{\sqrt{\frac{1}{N}\sum_{k=1}^{N}\left(ERPP_j(\omega_k)\right)^2}}{\sqrt{\frac{1}{N}\sum_{k=1}^{N}\left(ERPT(\omega_k)\right)^2}}$$

4. Only absolute equivalent radiated power values are available for element output.

ESE

Element Strain Energy Output Request

Requests the output of the strain energy in selected elements.

FORMAT:

$$\text{ESE}\left[\left(\left[\begin{matrix}(\text{PRINT}, \text{PUNCH})\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right]\left[\begin{matrix}\text{RMAG}\\ \text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right][\text{THRESH} = p]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ n\end{matrix}\right\}$$

EXAMPLES:

```
ESE=ALL
ESE (PUNCH, THRESH=.0001)=19
```

DESCRIBERS:

Describer	Meaning
PRINT	Write energies to the print file (default).
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy in frequency response analysis only.
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.

Describer	Meaning
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
RMAG	Outputs the energy magnitude in real data format.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
THRESH	Suppresses energies for elements having an energy value of less than p% in all output files. THRESH overrides the value of TINY described in Remark 1. (Default=0.001)
ALL	Computes energy for all elements.
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the ESE command or above all subcases. (Integer>0)
NONE	Element strain energy will not be output.

REMARKS:

1. If THRESH = p is not specified then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element strain energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.

4. The equations used to calculate elemental strain energy components are given below.

   Average Strain Energy:

$$E_{avg} = \frac{1}{4}\left[\{u_r\}^T[K_e]\{u_r\} + \{u_i\}^T[K_e]\{u_i\}\right]$$

   where:

E	=	elemental energy
$\{u_r\}$	=	displacement - real part
$\{u_i\}$	=	displacement - imaginary part
$[K_e]$	=	elemental stiffness

   Real Part of Strain Energy Amplitude:

$$E_{amp_real} = \frac{1}{4}\left[\{u_r\}^T[K_e]\{u_r\} - \{u_i\}^T[K_e]\{u_i\}\right]$$

   Imaginary Part of Strain Energy Amplitude:

$$E_{amp_imag} = -\frac{1}{2}\{u_r\}^T[K_e]\{u_i\}$$

   Magnitude of Strain Energy Amplitude:

$$E_{amp_mag} = \sqrt{E^2_{amp_real} + E^2_{amp_imag}}$$

   Phase of Strain Energy Amplitude:

$$\theta_{amp} = \arctan\left(\frac{E_{amp_imag}}{E_{amp_real}}\right)$$

   Peak Strain Energy:

$$E_{peak} = E_{avg} + E_{amp_mag}$$

5. In SOL 111, ESE can only be requested if PARAM,DDRMM,-1 is used.

6. Element data recovery for thermal loads is not currently implemented in dynamics.

7. Element strain energy is available for nonlinear static analysis (SOL 106), but the INTOUT entry on the NLPARM bulk entry must be set to “YES” since the

strain energy at the intermediate steps is required to correctly integrate the strain energy over all load steps. All other nonlinear solution sequences do not support element strain energy output.

8. The strain energy for nonlinear elements is calculated by integrating the specific energy rate, which is the inner product of strain rate and stress, over element volume and time.

$$E = \int_0^t \int_V \dot{\varepsilon}^T \sigma \, dV \, d\tau$$

Equation 4-1.

where:

σ = stress tensor

$\dot{\varepsilon}$ = tensor of the strain rate

V = element volume

t = actual time in the load history

Loads from temperature changes are included in Equation 4-1. If we assume a linear variation of temperatures from subcase to subcase, then the strain energy in Equation 4-1, for the special case of linear material and geometry becomes

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et}$$

Equation 4-2.

where P_{et} is the element load vector for temperature differences.

For linear elements, the default calculation of strain energy is also Equation 4-2. You can optionally set the parameter XFLAG to 0 to change the strain energy calculation to

$$E = \frac{1}{2} u^T K_e u - u^T P_{et}$$

Equation 4-3.

In Equation 4-3, the temperatures are assumed to be constant within a subcase. See the XFLAG parameter in the *NX Nastran Quick Reference Guide*.

EXTSEOUT

External Superelement Creation Specification

Specify the various requirements for the creation of an external superelement.

FORMAT:

$$\text{EXTSEOUT}\left[\left(\left[\text{STIFFNESS, MASS, DAMPING, K4DAMP, LOADS, GEOM, ASMBULK,}\right.\right.\right.$$

$$\left.\left.\left.\text{EXTBULK, EXTID = seid, DMIGSFIX = }\begin{Bmatrix} cccccc \\ EXTID \end{Bmatrix}\right]\left[\begin{matrix}\text{MATDB(orMATRIXDB)} \\ \text{DMIGDB} \\ \text{DMIGOP2=unit} \\ \text{DMIGPCH} \\ \text{MATOP4(orMATRIXOP4)=unit}\end{matrix}\right]\right)\right]$$

EXAMPLES:

```
EXTSEOUT
EXTSEOUT(ASMBULK,EXTID=100)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=200)
EXTSEOUT(EXTBULK,EXTID=300)
EXTSEOUT(DMIGDB)
EXTSEOUT(ASMBULK,EXTID=400,DMIGOP2=21)
EXTSEOUT(EXTID=500,DMIGPCH)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=500,DMIGSFIX=XSE500,DMIGPCH)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=500,DMIGSFIX=EXTID,DMIGPCH)
EXTSEOUT(STIF,MASS,DAMP,EXTID=600,ASMBULK,EXTBULK,MATDB)
EXTSEOUT(STIF,MASS,DAMP,GEOM,EXTID=600)
```

DESCRIBERS:

Describer	Meaning
STIFFNESS	Store the boundary stiffness matrix. See Remarks 1 and 2.
MASS	Store the boundary mass matrix. See Remark 1.

Describer	Meaning
DAMPING	Store the boundary viscous damping matrix. See Remarks 1 and 2.
K4DAMP	Store the boundary structural damping matrix. See Remarks 1 and 2.
LOADS	Store the loads matrix and associated DTI,SELOAD entries. See Remarks 1, 2, and 21.
GEOM	Store geometry. See Remark 20.
ASMBULK	Generate bulk entries related to the subsequent superelement assembly process and store them on the assembly punch file (.asm). This data is to be included in the main bulk portion of the subsequent assembly solution. See Remarks 4 and 13.
EXTBULK	Generate and store bulk entries for the external superelement on the standard punch file (.pch) when used in combination with one of either MATDB, DMIGDB, or DMIGOP2. This data is used in the BEGIN SUPER portion of the bulk section of the subsequent assembly solution. EXTBULK is ignored if either DMIGPCH or MATOP4 is specified. If EXTBULK is not specified, the subsequent assembly solution retrieves the required data for the external superelement from the medium on which the boundary matrices are stored. See Remarks 5 and 6.
EXTID = *seid*	*seid* (integer>0) is the superelement ID to be used in the SEBULK and SECONCT bulk entries stored on the assembly punch file (.asm) if ASMBULK is specified and in the BEGIN SUPER bulk entry stored on the standard punch file (.pch) if DMIGPCH or MATOP4 is specified. See Remarks 3, 4, 5, and 7.
DMIGSFIX = *cccccc*	*cccccc* is the suffix (up to six characters and must not = any EXTSEOUT keyword) that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH keyword is specified. See Remarks 8 through 11.

Describer	Meaning
DMIGSFIX = EXTID	The *seid* defined by the EXTID keyword is the suffix that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH keyword is specified. See Remarks 8 through 11.
MATDB (or MATRIXDB)	Store the boundary matrices and other information on the database (default).
DMIGDB	Similar to MATDB (or MATRIXDB) except that the boundary matrices are stored as DMIG bulk entries on the database.
DMIGOP2 = *unit*	Store the boundary matrices as DMIG bulk entries on an OUTPUT2 file whose Fortran unit number if given by *unit* (integer>0). See Remark 14.
DMIGPCH	Store the boundary matrices as DMIG bulk entries on the standard punch file (.pch). See Remarks 6 through 13.
MATOP4 = *unit* (or MATRIXOP4 = *unit*)	Store the boundary matrices on an OP4 file whose Fortran unit number is given by *unit*. (Integer ≠ 0) If *unit* > 0, matrices are written to the OP4 file in sparse format. If *unit* < 0, matrices are written to the OP4 file in full matrix format. See Remarks 3, 5, 6, 13, 14, and 23.

REMARKS:

1. If none of the describers STIFFNESS, MASS, DAMPING, K4DAMP, and LOADS are specified, then all of the boundary matrices are stored by default. If any subset of the describers STIFFNESS, MASS, DAMPING, K4DAMP, and LOADS are specified, then only the boundary matrices specified are stored.

2. STIFFNESS, DAMPING, K4DAMP, and LOADS may be abbreviated to STIF, DAMP, K4DA, and LOAD, respectively.

3. EXTID and an *seid* value must be specified if one or more of ASMBULK, EXTBULK, DMIGPCH, or MATOP4 are specified. If the DMIGSFIX=EXTID form is employed along with the DMIGPCH keyword, the value *seid* may not exceed

999999, since this value becomes part of the names given to the DMIG matrices generated on the standard punch file (.pch). See Remark 11.

4. If ASMBULK is specified, the following bulk entries are generated and stored on the assembly punch file (.asm):

   SEBULK *seid* …

   SECONCT *seid* …

   GRID entries for the boundary points

   CORD2x entries associated with the above GRID entries

5. If DMIGPCH is not specified, but MATOP4 or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) is specified, the following bulk entries are generated and stored on the standard punch file (.pch):

   BEGIN SUPER *seid*

   GRID entries for the boundary points

   GRID entries for the interior points referenced by PLOTEL entries

   CORD2x entries associated with the above GRID entries

   EXTRN

   ASET/ASET1

   QSET/QSET1

   SPOINT

   PLOTEL

6. If DMIGPCH or MATOP4 is specified, then EXTBULK is ignored even if it is specified.

7. If DMIGPCH is specified, the following bulk entries are generated and stored on the standard punch file (.pch):

   BEGIN SUPER seid

   GRID entries for the boundary points

   CORD2x entries associated with the above GRID entries

   ASET/ASET1

   SPOINT

DMIG entries for the requested boundary matrices

8. The DMIGSFIX keyword is ignored if DMIGPCH is not specified.

9. If DMIGPCH is specified without the DMIGSFIX keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

   KAAX (boundary stiffness matrix)

   MAAX (boundary mass matrix)

   BAAX (boundary viscous damping matrix)

   K4AAX (boundary structural damping matrix)

   PAX (boundary load matrix)

10. If the DMIGSFIX = *cccccc* form is employed along with the DMIGPCH keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

    K*cccccc* (boundary stiffness matrix)

    M*cccccc* (boundary mass matrix)

    B*cccccc* (boundary viscous damping matrix)

    K4*cccccc* (boundary structural damping matrix)

    P*cccccc* (boundary load matrix)

11. If the DMIGSFIX = EXTID form is employed along with the DMIGPCH keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

    K*seid* (boundary stiffness matrix)

    M*seid* (boundary mass matrix)

    B*seid* (boundary viscous damping matrix)

    K4*seid* (boundary structural damping matrix)

    P*seid* (boundary load matrix)

12. If the DMIGPCH option is specified, the boundary DMIG matrices generated and stored on the standard punch file (.pch) may not be as accurate as the boundary matrices resulting from other options (MATDB/MATRIXDB or DMIGOP2

or MATOP4/MATRIXOP4). Accordingly, this may result in decreased accuracy from the subsequent assembly job utilizing these DMIG matrices.

13. The punch output resulting from EXTSEOUT usage is determined by ASMBULK, EXTBULK, DMIGPCH, and MATOP4 as follows:

    - No ASMBULK, EXTBULK, DMIGPCH, or MATOP4 results in no punch output.
    - ASMBULK, but no DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output being generated and stored on the assembly punch file (.asm). See Remark 4.
    - No ASMBULK, but DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output being generated and stored on the standard punch file (.pch). See Remarks 5 or 7, as appropriate.
    - ASMBULK and DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output consisting of two distinct and separate parts. One part is generated and stored on the assembly punch file (.asm) as indicated in Remark 4. The other part is generated and stored on the standard punch file (.pch) as indicated in Remark 5 or 7, as appropriate.

14. If DMIGOP2=*unit* or MATOP4=*unit* is specified, an appropriate ASSIGN OUTPUT2 or ASSIGN OUTPUT4 statement must be present in the File Management Section (FMS) for the absolute value of *unit*. An OUTPUT2 file created during a LP-64 Nastran executable superelement creation run cannot be used in a ILP-64 Nastran executable system run and vice versa.

15. The creation of an external superelement using EXTSEOUT involves running a NX Nastran job with the following additional data:

    - The data for the creation of the external superelement is specified by the EXTSEOUT case control command, which must appear above the subcase level.
    - The boundary points of the external superelement are specified by ASET/ASET1 bulk entries.
    - If the creation involves component mode reduction, the required generalized coordinates are specified using QSET/QSET1 bulk entries. The boundary data for the component mode reduction may be specified using the BNDFIX/BNDFIX1 and BNDFREE/BNDFREE1 bulk entries (or their equivalent BSET/BSET1 and CSET/CSET1 bulk entries). (The default scenario assumes that all boundary points are fixed for the component mode reduction.)
    - The output for the external superelement is generated in the assembly job. This output consists of displacements, velocities, accelerations, SPC forces,

MPC forces, grid point force balances, stresses, strains, and element forces. However, in order for this output to be generated in the assembly job, the output requests must be specified in the external superelement creation run. Normally, the only output requests for the external superelement that are honored in the assembly job are those that are specified in the creation run. There is, however, one important exception to this: the displacement, velocity, acceleration, SPC forces, and MPC forces output for the boundary grid points as well as for all grid points associated with PLOTEL bulk entries can be obtained in the assembly job *even if there is no output request specified for these points in the creation run*.

- If the assembly job involves the use of PARAM bulk entries, then the following points should be noted:
  - o PARAM entries specified in the main bulk portion of the input data apply *only to the residual and not to the external superelement*.
  - o PARAM entries specified in the BEGIN SUPER portion of the bulk section for an external superelement apply *only to the superelement*.
  - o The most convenient way of ensuring that PARAM entries apply not only to the residual but also to all external superelements is to specify such PARAM entries in Case Control, not in the main bulk section. This is particularly relevant for such PARAMs as POST.

16. Output transformation matrices (OTMs) are generated for the following outputs requested in the in external superelement run with EXTSEOUT:
    - DISPLACEMENT
    - VELOCITY
    - ACCELERATION
    - SPCFORCE
    - MPCFORCES
    - GPFORCE
    - STRESS
    - STRAIN
    - FORCE

    Only these external superelement results can be output in the system analysis run. PARAM,OMID,YES is not applicable to the OTMs.

17. If a PARAM,EXTOUT or PARAM,EXTUNIT also exist, they will be ignored. The existence of the EXTSEOUT case control command takes precedence over PARAM,EXTOUT and PARAM,EXTUNIT.

18. This capability is enabled in SOLs 101, 103, 107-112, 129, 144-146, 159, and 187. In general, it is not supported for thermal analyses, however, in SOL 101 with Guyan reduction, thermal loads can be transferred to a system model via the o-set output transformation matrices. Fluid and acoustic elements such as absorbors are not supported in an external superelement, although they can be included as part of the residual in a system run. For SOL 103, this capability will create the external superelement and also create the component results and data blocks to be written to an .op2 file if requested. For the other solution sequences, this capability will only create the external superelement and no other analyses will be performed. Superelement results can be recovered in the second step (i.e. superelement assembly, analysis, and data recovery) for SOLs 101, 103, 105, 107-112, and 144-146.

19. The run creating the external superelement using this capability is not a superelement run. No superelement designations are allowed (i.e. SUPER, SEALL, SESET, BEGIN SUPER, etc.).

20. The GEOM describer will output geometry data blocks GEOM1EXA, GEOM2EXA, and GEOM4EXA containing all of the external superelement geometry to support post-processing. This describer only works for the MATDB (or MATRIXDB), DMIGDB, and DMIGOP2 storage options. By default, the full geometry will not be exported; the GEOM describer must be explicitly defined to have these geometry data blocks written.

21. The LOADS describer will output load information in the $[P_a]$ matrix along with associated DTI,SELOAD bulk entries for each load represented in the $[P_a]$ matrix. DTI,SELOAD bulk entries will not be output for the DMIGPCH option; the use of the DMIGPCH option requires the use of the P2G case control command in the system analysis in order to access the load information defined in the $[P_a]$ matrix that is stored in DMIG format. Thermal loads and enforced motion loads using the SPCD bulk data definition method are not supported. The load information and DTI,SELOAD bulk entries that are output depend on the method in which loads are defined and referenced.

    Static solutions. For example, SOL 101:

    - In a creation run, the LOAD = *n* case control command will create a single load. The $[P_a]$ matrix will have a single column that contains the load values referenced by *n*. The corresponding LIDSE and EXCSE values on the DTI,SELOAD bulk entry will be the value *n*.

      To select this load in a system run, the LIDSE field on the SELOAD bulk entry should equal the value of *n* from the creation run.

- In a creation run, the LOADSET = *n* case control command is generally used to create multiple loads. Each definition of an LSEQ bulk entry will create a column in the $[P_a]$ matrix regardless of whether or not the LSEQ is referenced by the LOADSET = *n* case control command. However, an unreferenced LSEQ will generate a null column in the $[P_a]$ matrix. The corresponding LIDSE and EXCSE on an DTI,SELOAD bulk entry will be the value of LID and EXCITEID, respectively, on an LSEQ bulk entry.

  To select these loads in a system run, the LIDSE field on the SELOAD bulk entry should equal the value of an EXCITEID from the creation run.

- If both LOADSET and LOAD appear in the case control, LOADSET takes precedence.

Dynamic solutions. For example, SOL 103:

- In a creation run, a column in the $[P_a]$ matrix is created for each load (not enforced motion) defined on RLOADi and TLOADi bulk entries whether or not they are referenced in the case control. The corresponding LIDSE and EXCSE on the DTI,SELOAD bulk entries will both be the value of EXCITEID on the RLOADi or TLOADi bulk entry.

  To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

- In a creation run, the LOADSET = n case control command can be used to create multiple loads. Each definition of an LSEQ bulk entry will create a column in the [Pa] matrix. A LSEQ bulk entry must be referenced by a LOADSET case control command in order to generate a column in the $[P_a]$ matrix. The corresponding LIDSE and EXCSE on an DTI,SELOAD bulk entry will be the value of LID and EXCITEID, respectively, on an LSEQ bulk entry.

  To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

22. To include differential stiffness in the definition of an external superelement, two subcases are required in the creation run. The first subcase is a static subcase. The second subcase performs the analysis and generation of the external superelement. To obtain the displacement field used to generate the differential stiffness effects, include a STATSUB case control command in the second subcase that references the first subcase. Always include the EXTSEOUT case control command above the subcase level. An example of the required setup is as follows:

```
…
SOL 112
CEND
TITLE = …
EXTSEOUT(…)
$
```

```
SUBCASE 1
     $ STATIC SUBCASE
     LOAD = 11
$
SUBCASE 2
     $ DYNAMIC SUBCASE
     TSTEP = 100
     STATSUB = 1
     METHOD = 10
     DLOAD = 20
BEGIN BULK
…
```

For accuracy and consistency, the loads used to generate differential stiffness for the external superelement during the creation run should be the same loads used in the system run without any scaling. If the loads are scaled by a non-unity scaling factor from a case control command like P2G or a bulk entry like LOAD, the differential stiffness portion of the external superelement stiffness matrix will no longer be consistent with the applied loads.

The ability to generate an external superelement including differential stiffness effects is available for SOLs 103, 107-112, and 187. For SOL 112, the IC(STATSUB,DIFFK) or IC(TZERO,DIFFK) case control commands can be used to generate differential stiffness effects instead of the STATSUB case control command.

23. For the MATOP4 (or MATRIXOP4) option, the number of digits of precision for matrix data is controlled by the DIGITS parameter.

24. See Parameters and Superelements for considerations regarding parameter definitions in a system solution which includes external superelements.

FLSFSEL

Fluid-Structure Frequency Selection

Frequency selection for controlling the fluid-structure interaction.

FORMAT:

$$\text{FLSFSEL}\left[\text{LFREQFL}=\begin{Bmatrix}0.0\\ f_1\end{Bmatrix}\right],\left[\text{HFREQFL}=\begin{Bmatrix}1.0\text{E}+30\\ f_1\end{Bmatrix}\right],$$
$$\left[\text{LFREQF}=\begin{Bmatrix}0.0\\ f_1\end{Bmatrix}\right],\left[\text{HFREQF}=\begin{Bmatrix}1.0\text{E}+30\\ f_1\end{Bmatrix}\right],$$
$$\left[\text{LMODESFL}=\begin{Bmatrix}0\\ m_1\end{Bmatrix}\right],\left[\text{LMODES}=\begin{Bmatrix}0\\ m_1\end{Bmatrix}\right],$$
$$\left[\text{FLUIDSE}=\begin{Bmatrix}0\\ seid\end{Bmatrix}\right]$$

EXAMPLES:

```
FLSFSEL LFREQL=1.0 HFREQFL=2.0
```

DESCRIBERS:

Describer	Meaning
LFREQFL	The lower bound modal frequency for fluid modes in hertz. Default is 0.0 unless f_1 is specified.
HFREQFL	The upper bound modal frequency for fluid modes in hertz. Default is 1.0E+30 unless f_1 is specified.
LFREQ	The lower bound modal frequency for structural modes in hertz. Default is 0.0 unless f_1 is specified.

Describer	Meaning
HFREQF	The upper bound modal frequency for structural modes in hertz. Default is 1.0E+30 unless f_1 specified.
LMODESFL	The number of lowest fluid modes to be used. If 0, then LFREQFL and HFREQFL will be used to determined number of modes. Default is 0 unless m_1 is specified.
LMODES	The number of lowest structural modes to be used. If 0, then LFREQFL and HFREQFL will be used to determined number of modes. Default is 0 unless m_1 is specified.
FLUIDSE	Defines a superelement for fluids only. Default is 0, unless *seid* is specified.

REMARKS:

All the entries specified in this case control statements are available as PARAMETER statements.

FLSPOUT

Fluid-Structure Modal Participation Output

Modal participation output control for the fluid-structure interaction.

FORMAT:

$$\text{FLSPOUT}\left[\text{FLUIDMP}=\left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}\right],\left[\text{STRUCTMP}=\left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}\right],$$

$$\left[\text{GRIDFMP}=\left\{\begin{matrix}\text{ALL}\\ setf\end{matrix}\right\}\right],\left[\text{GRIDMP}=\left\{\begin{matrix}\text{ALL}\\ setg\\ \text{NONE}\end{matrix}\right\}\right],$$

$$\left[\text{OUTFMP}=\left\{\begin{matrix}\text{ALL}\\ p\\ \text{NONE}\end{matrix}\right\}\right],\left[\text{OUTSMP}=\left\{\begin{matrix}\text{ALL}\\ p\\ \text{NONE}\end{matrix}\right\}\right],$$

$$\left[\text{PANELMP}=\left\{\begin{matrix}\text{ALL}\\ setp\\ \text{NONE}\end{matrix}\right\}\right],\left[\text{FEPS}=\left\{\begin{matrix}\text{1.0E -11}\\ eps\end{matrix}\right\}\right],$$

$$\left[\text{SEPS}=\left\{\begin{matrix}\text{1.0E -11}\\ eps\end{matrix}\right\}\right],\left[\text{ARF}=\left\{\begin{matrix}0.95\\ a\end{matrix}\right\}\right],\left[\text{ARS}=\left\{\begin{matrix}0.95\\ a\end{matrix}\right\}\right],$$

$$\left[\text{PSORT}=\left\{\begin{matrix}\text{ABSOLUTE}\\ \text{REAL}\\ \text{IMAGINARY}\end{matrix}\right\},\left\{\begin{matrix}\text{DESCENDING}\\ \text{ASCENDING}\end{matrix}\right\}\right]$$

EXAMPLES:

```
FLSPOUT FLUIDMP=ALL PANELMP=ALL STRUCTMP=ALL
```

DESCRIBERS:

Describer	Meaning
FLUIDMP	First “n” fluid modes to be used for modal participation requests. See FLUIDMP parameter. Default=NONE.
STRUCTMP	First “n” structural modes to be used for modal participation requests. See FLUIDMP parameter. Default=NONE.
GRIDFMP	Fluid grid set for modal participation output. Default is ALL unless setf is specified in which case the modal participation factors will be output for all fluid grids in setf.
GRIDMP	Structure grid set for modal participation output. Default is NONE unless setg is specified in which case the modal participation factors will be output for all structural grids.
OUTFMP	Request “p” highest fluid mode participation factors be output. Default=NONE.
OUTSMP	Request “p” highest structural mode participation factors be output. Default=NONE.
PANELMP	Include panel id's which are included in a “setp” case control set for modal participation calculations. Default=NONE.
FEPS	Threshold for filtering fluid modal participation factors from being printed. Default=1.0E-11.
SEPS	Threshold for filtering structure modal participation factors from being printed. Default=1.0E-11.
ARF	Determines which fluid participation factors are printed. The default is ARF=0.95, which means only values above 95% of the highest value for each mode are printed.
ARS	Determines which structural participation factors are printed. The default is ARS=0.95, which means only values above 95% of the highest value for each mode are printed.
PSORT	Requests how the modal participation output is sorted. The request must occur in pairs. For example, PSORT= (REAL,ASCENDING).

REMARKS:

All the entries specified in this case control statements are available as PARAMETER statements.

FLSTCNT

Control Parameters for Fluid-Structure Interaction

Miscellaneous control parameters for fluid-structure interaction.

FORMAT:

$$\text{FLSTCNT}\left[\text{ACSYM}=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}\right],\left[\text{ACOUT}=\left\{\begin{matrix}\text{PEAK}\\ \text{RMS}\end{matrix}\right\}\right],$$

$$\left[\text{ASCOUP}=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}\right],\left[\text{PREFDB}=\left\{\begin{matrix}1.0\\ P\end{matrix}\right\}\right],$$

$$\left[\text{SKINOUT}=\left\{\begin{matrix}\text{NONE}\\ \text{PUNCH}\\ \text{FREEFACE}\\ \text{STOP}\end{matrix}\right\}\right],\left[\text{AGGPCH}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],$$

$$\left[\text{SFEF70}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],$$

EXAMPLES:

```
FLSTCNT ACOUT=RMS PREFDB=1.0E-06
```

DESCRIBERS:

Describer	Meaning
ACSYM	Requests the symmetric solution for fluid structure analysis, when YES. When NO, non-symmetric solution is requested. (Character; Default = YES)

Describer	Meaning
ACOUT	PEAK or RMS output with the FORCE Case Control request. (Character; Default = PEAK)
ASCOUP	Requests a fluid-structure coupled solution when YES. When NO, the fluid and the structure are decoupled (Character; Default = YES)
PREFDB	Specifies the reference pressure. (Real; Default = 1.0)
SKINOUT	Specifies if coupling data with the pairing information is output. SKINOUT=FREEFACE writes coupling data to a .dat and punch files, and writes the coupling datablock to the .op2 file. The solution proceeds after the data is written. Both coupled and uncoupled faces are written. SKINOUT=STOP works like SKINOUT=FREEFACE, but the solution stops immediately after the debug files are created. SKINOUT=PUNCH writes coupling data to .dat and punch files. The solution proceeds after the data is written. It will not write the coupling datablock to the .op2 file. Only coupled free faces are written. SKINOUT=NONE does not output coupling data. (Character; Default = NONE)
AGGPCH	Requests the output of the fluid-structure coupling matrix AGG to the punch file when YES. When NO, the coupling matrix is not written. (Character; Default = NO)
SFEF70	Requests the import of a fluid-structure coupling matrix created by SFE AKUSMOD when YES. When NO, the coupling matrix is not imported. (Character; Default = NO)
NONE	No debug deck or pairing information is generated.
PUNCH	Debug pairing information file is created. The name of the files will be the base name of the deck appended with “_acdbg.dat” and “_acdbg.pch”.
FREEFACE	All free faces are written into a debug file.

Describer	Meaning
STOP	The run is terminated as soon as the debug file with pairing information is created.

REMARKS:

1. All the entries specified in this case control statement are available as PARAMETER statements.

2. The coupling data files output by the SKINOUT describer include dummy shell elements. The PSHELL IDs referenced be these elements have the following meaning.

   - Shell elements representing the *coupled* structural free faces reference PSHELL ID = 1.

   - Shell elements representing the *coupled* fluid free faces reference PSHELL ID = 2.

   - Shell elements representing the *uncoupled* structural free faces reference PSHELL ID = 3.

   - Shell elements representing the *uncoupled* fluid free faces reference PSHELL ID = 4.

3. If SFEF70 = YES, NX Nastran does not compute the coupling, and instead uses the SFE AKUSMOD coupling definition from the external file. NX Nastran expects the AKUSMOD file in the same directory where the job is being run, and expects the file name to be fort.70. An ASSIGN statement that uses UNIT=70 must be defined in the file management section of your input file if the coupling file is not named fort.70 or if it is in a location other than where the job is run. For example,

   ASSIGN OUTPUT2='/directory_path/user_file_name.70' UNIT=70

FLUX

Flux, Gradient, or Particle Velocity Output Request

Requests the form and type of gradient and flux output in heat transfer solutions 153, 159, and 601,n. Also requests particle velocity output in a coupled fluid-structural analysis with solutions 103, 107, 108, 109, 110, 111, 112 and 200.

FORMAT:

$$\text{FLUX}[(\text{PRINT}, \text{PLOT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
FLUX=ALL
FLUX(PUNCH,PRINT)=17
FLUX=25
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	The output will be sent to the plot file.
ALL	Flux for all elements will be output.
NONE	Flux for no elements will be output.
n	Set identification of a previously appearing SET command. Only fluxes of elements with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. FLUX=ALL in SOL 159 may produce excessive output.
2. FLUX=NONE overrides an overall request.

REMARKS RELATED TO SOL 601:

1. FLUX output request is supported for heat transfer analysis in SOL 601,153 and SOL 601,159.
2. Output for CHBDY type elements is not supported.

FMETHOD

Flutter Analysis Method Parameter Selection

Selects the parameters to be used by the aerodynamic flutter analysis.

FORMAT:

FMETHOD=n

EXAMPLES:

```
FMETHOD=72
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a FLUTTER Bulk Data entry. (Integer>0)

REMARKS:

1. An FMETHOD command is required for flutter analysis.
2. A CMETHOD command in also required for the K-method of flutter analysis.
3. If this entry is being used in SOL 200 in conjunction with flutter design conditions, the METHOD selected on the FLUTTER Bulk Data entry must be “PK” or “PKNL”.

FORCE

Element Force Output or Particle Velocity Request

Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis.

FORMAT:

$$\text{FORCE}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix},\begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
FORCE=ALL
FORCE(REAL, PUNCH, PRINT)=17
FORCE=25
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.

Describer	Meaning
PLOT	Generates force output for requested set but no printed output.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 6.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 6.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 6.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 6.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 6.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 6.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 6.
RPUNCH	Writes random analysis results to the punch file. See Remark 6.

Describer	Meaning
CENTER	Output CQUAD4, CQUADR, CTRIAR element forces at the center only.
CORNER or BILIN	Output CQUAD4, CQUADR, CTRIAR element forces at the center and grid points using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element forces at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element forces at center and grid points using cubic bending correction.
ALL	Forces for all elements will be output.
n	Set identification of a previously appearing SET command. Only forces of elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	Forces for no elements will be output.

REMARKS:

1. ALL should be used with caution in transient solutions as it can produce a large amount of output.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

- XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. FORCE=NONE overrides an overall request.

4. In nonlinear transient analysis, this request is ignored for nonlinear elements.

5. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. Consequently, options specified in subcases other than the first subcase will be ignored.

   - If the STRESS command is specified in the first subcase then the option on the STRESS command is used in all subcases with STRESS, STRAIN, and FORCE commands.

   - If the STRAIN command and no STRESS command is specified in the first subcase, then the option on the STRAIN command is used in all subcases containing STRESS, STRAIN, and FORCE commands.

   - If the FORCE command and no STRESS or STRAIN command is specified in the first subcase, then the option on the FORCE command is used in all subcases containing STRESS, STRAIN, and FORCE commands.

   - If STRESS, STRAIN, and FORCE commands are not specified in the first subcase, then the CENTER option is used in all subcases containing STRESS, STRAIN, and FORCE commands.

6. The following applies to random solutions:

   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

FREQUENCY

Frequency Set Selection

Selects the set of forcing frequencies to be solved in frequency response problems.

FORMAT:

FREQUENCY=n

EXAMPLES:

```
FREQUENCY=17
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5 Bulk Data entries. (Integer>0)

REMARKS:

1. A frequency set selection is required for a frequency response problem.
2. A frequency set selection is required for transient response by Fourier methods (SOL 146).
3. All FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5 entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if
   - $|f_N - f_{N-1}|<10^{-5}|f_{MAX} - f_{MIN}|$

GCRSTRN

Requests creep strain output at Gauss points for SOL 401.

Requests creep strain at Gauss points.

FORMAT:

$$\text{GCRSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
GCRSTRN=ALL
GCRSTRN(PRINT,PLOT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GELSTRN

Requests elastic strain output at Gauss points for SOL 401.

Requests elastic strain at Gauss points.

FORMAT:

$$\text{GELSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
GELSTRN=ALL
GELSTRN(PRINT,PLOT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GKRESULTS

Gasket Result Output Request (SOL 601 only)

Requests the form and type of gasket result output in SOL 601.

FORMAT:

$$\text{GKRESULTS}\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix}\right) = \text{ALL or NONE}$$

EXAMPLES:

```
GKRESULTS=ALL
GKRESULTS(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PLOT	Computes and puts gasket results in OP2 file only.
ALL	Gasket results will be output for all gasket elements.
NONE	Gasket results will not be output.

REMARKS:

1. Gasket results consist of gasket pressure, gasket closure strain, gasket yield pressure, plastic gasket closure strain, and gasket status. Figure 4-1 depicts a gasket load/unload characteristic defined using a MATG bulk entry where TABLD, TABLUi, and YPRESS refer to fields on the MATG bulk entry.

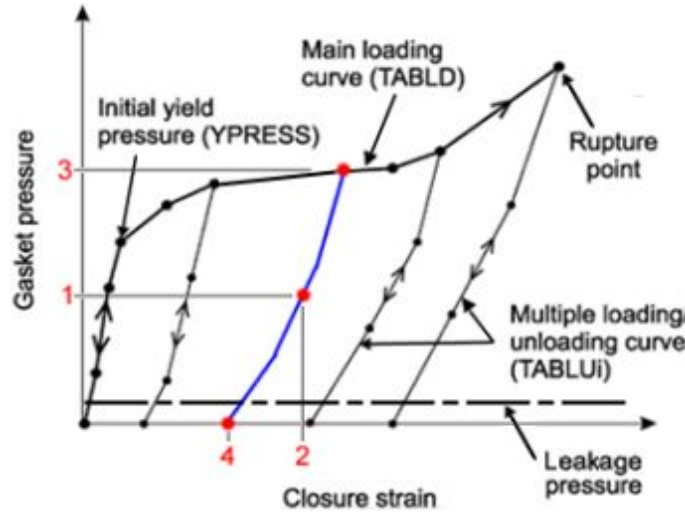


Figure 4-1. Gasket load/unload characteristic. Gasket pressure (1), gasket closure strain (2), gasket yield pressure (3), plastic gasket closure strain (4).

2. The leakage pressure is automatically set to 1% of the initial yield pressure.

3. Gasket status is defined as follows:

Status	**Definition**	**Integer in .op2 (OGK datablock)**
Open	The gasket pressure is less than the leakage pressure.	1
Closed	The gasket pressure is higher than the leakage pressure but has not yet caused plasticity.	2
Sealed	There has been plastic gasket deformation and the current pressure is above the gasket leakage pressure.	3
Leaked	After plastic deformation, the gasket pressure has dropped below gasket leakage pressure.	4
Crushed	Gasket closure strain has exceeded the rupture value.	5

GPFORCE

Grid Point Force Output Request

Requests grid point force balance at selected grid points.

FORMAT:

$$\text{GPFORCE}\ \left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix},\text{PUNCH}\right],\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
GPFORCE=ALL
GPFORCE=17
GPFORCE(SORT2,PRINT,PUNCH,PHASE) = 123
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time depending on solution sequence. (Default)
SORT2	Output will be presented as a tabular listing of load, frequency, eigenvalue, or time for each grid point.
PRINT	The printer will be the output medium. (Default)
PUNCH	The punch file will be the output medium.
NOPRINT	Generates, but does not print, grid point force balance results.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
ALL	Grid point force balance for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the grid point force balance output. (Integer>0)
NONE	Grid point force balance is not calculated or output.

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. PARAM,DDRMM,-1 is required in the modal solution sequences 111, 112, 146, and 200.

4. The printing of the grid point forces will be suppressed if PARAM,NOGPF,-1 appears in the Bulk Data.

5. The Bulk Data entry PARAM,NOELOF,+1 will cause the output of the grid point forces to be aligned with the edges of the two-dimensional elements. The default value of -1 will suppress this output.

6. The Bulk Data entry PARAM,NOELOP,+1 will cause the output of the sum of the forces parallel to the edges of adjacent elements. The default value of -1 will suppress this output.

7. The output of grid point forces aligned with the edges of elements is available for the following elements:

```
CBAR
CROD
CBEAM
CSHEAR
CONROD
CTRIA3
```

   The positive direction for grid point forces aligned with the edges of elements is from the reference point to the load point as indicated on the printed output.

8. Grid point force balance is computed from linear and nonlinear elements, the sum of applied and thermal loads, and MPC and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), general elements, DMIG entries, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. The following table summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system:

Contributions Included	Contributions Ignored
Applied Loads	GENEL Forces
SPC Forces	DMIG and DMI Forces
Element Elastic Forces	Boundary Loads from Upstream Superelements
Thermal Loads	
MPC and Element Forces	

9. Only the element elastic forces are included when the grid point forces are aligned with the edges of elements.

10. In inertia relief analysis, the GPFORCE output related to SPCFORCES and applied loads is interpreted differently for SOLs 101 and 200:

    - In SOLs 101 and 200, the SPCFORCE and applied load output includes both the effect of inertial loads and applied loads.

11. GPFORCE is supported in a nonlinear static analysis (SOLs 106 and 601). It is not supported in SOLs 129 or 701. PARAM,NOELOF and PARAM,NOELOP are not supported in nonlinear static analysis; therefore, Remarks 3,4,5 and 7 do not apply to SOL 106.

GPKE

Grid Point Kinetic Energy Output Request

Requests the output of the kinetic energy at selected grid points in normal modes analysis only.

FORMAT:

$$\text{GPKE}\left[\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right], (\text{PUNCH},\ \text{THRESH} = \text{e})\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
GPKE=ALL
GPKE (PRINT, PUNCH)=19
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generates, but does not print, grid point kinetic energy output.
PUNCH	The punch file will be the output medium.
e	Minimum energy threshold. Only energies above this value will be printed and/or punched.
ALL	Grid point kinetic energy for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Grid point kinetic energy for no points will be output.

REMARKS:

1. Grid point kinetic energy is only available for normal modes analysis.

2. Both PRINT and PUNCH may be requested.

3. GPKE=NONE overrides an overall output request.

4. For models using the lumped mass formulation, the grid point kinetic energy can be used to examine the distribution of kinetic energy among the grid points. It is computed as:

$$E_{k_g} = \Phi_g^{mass} \otimes [M_{gg}\Phi_g^{mass}]$$

   where φ_g^{mass} is the mass-normalized eigenvector so that the total grid point kinetic energy is scaled to be unity. Note that the operator shown in the previous equation indicates term-wise matrix multiplication.

5. The grid point kinetic energy output has limited meaning for a coupled mass formulation. Since this mass formulation produces a coupling of mass across grid points, the sharing of kinetic energy among grid points can occur. In general, this obscures the meaning of the computation as a means of identifying important model parameters to control modal behavior.

GPLSTRN

Requests plastic strain output at Gauss points for SOL 401.

Requests plastic strain at Gauss points.

FORMAT:

$$\text{GPLSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix}\right\}$$

EXAMPLES:

```
GPLSTRN=ALL
GPLSTRN(PRINT,PLOT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GPRSORT

Composites Ply Results Sorted Output

Requests that the output of composites ply results (stress, strain and failure indices) on elements referencing a PCOMPG be sorted by global ply ID, then by element ID, versus by element ID, then by global ply ID.

FORMATS:

$$\text{GPRSORT} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

EXAMPLES:

```
GPRSORT = ALL
GPRSORT = 16
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a previously appearing SET command.
ALL	All composite elements referencing a PCOMPG property entry type. See Remarks.

REMARKS:

1. Composite element output will be sorted by global ply ID, and by element ID. This sorted output is only available for composite elements referencing a PCOMPG property entry, which includes global ply IDs.
2. Composite elements referencing the PCOMP property entry will be excluded from the sorted output.

GPSDCON

Grid Point Stress Discontinuity Output Request

Requests mesh stress discontinuities based on grid point stresses.

FORMAT:

$$\text{GPSDCON}\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix} = \begin{Bmatrix}\text{ALL}\\ \text{n}\end{Bmatrix}$$

EXAMPLES:

```
GPSDCON=ALL
GPSDCON=19
```

DESCRIBERS:

Describer	Meaning
ALL	Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request. (Integer>0)
NONE	No grid point stress discontinuity output.

REMARKS:

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.

2. Only elements used to define the surface or volume are output. See the description for the SURFACE or VOLUME commands.

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS command must be present for printed output and the STRFIELD command for postprocessing output using the .xdb file (PARAM,POST,0) for the same surfaces and volumes.

GPSTRAIN

Grid Point Strain Output Request for Printing Only

Requests grid point strains for printing only.

FORMAT:

$$\text{GPSTRAIN}\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix} = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
GPSTRAIN=ALL
GPSTRAIN=19
```

DESCRIBERS:

Describer	Meaning
ALL	Grid point strain requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point strain output request. (Integer>0)
NONE	No grid point strain output.

REMARKS:

1. For statics, normal modes and transient analysis output will be presented for each surface or volume as a tabular listing of grid point strains for each load,

eigenvalue, and time step. (See “DISPLACEMENT” for a discussion of SORT1 and SORT2.)

2. Only grid points connected to elements used to define the surface or volume are output. See the description for “SURFACE” and “VOLUME” .

3. Element strain output (STRAIN) must be requested for elements referenced on SURFACE and VOLUME commands.

4. In nonlinear transient analysis, grid point strains are computed only if parameter LGDISP is -1, which is the default, and for elements with linear material properties only.

5. For the graphical display of grid point strains in the computation of mesh strain discontinuities, the STRFIELD command must be specified.

GPSTRESS

Grid Point Stress Output Request

Requests grid point stresses.

FORMAT:

$$\text{GPSTRESS}\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix} = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
GPSTRESS=ALL
GPSTRESS=19
```

DESCRIBERS:

Describer	Meaning
ALL	Grid point stress requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point stress output request. (Integer>0)
NONE	No grid point stress output.

REMARKS:

1. For statics, normal modes and transient analysis output will be presented for each surface or volume as a tabular listing of grid point stresses for each load,

eigenvalue, and time step. (See “DISPLACEMENT” for a discussion of SORT1 and SORT2.)

2. Only grid points connected to elements used to define the surface or volume are output. See the description for “SURFACE” and “VOLUME” .

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands.

4. In nonlinear transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default, and for elements with linear material properties only. Grid point stresses are not computed for the hyperelastic elements.

5. For the graphical display of grid point stresses in the computation of mesh stress discontinuities, the STRFIELD command must be specified.

6. Grid point stress is not output for midside nodes.

GROUNDCHECK

Rigid Body Motion Grounding Check

Perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

FORMAT:

$$\text{GROUNDCHECK}\left[\begin{array}{l}\left(\left[\begin{array}{c}\text{PRINT}\\ \text{NOPRINT}\end{array}\right], \text{PUNCH}, \text{SET}=\left(\left\{\begin{array}{c}\text{G, N, N + AUTOSPC, F, A}\\ \text{ALL}\end{array}\right\}\right)\right.\\ \left.\text{GRID=gid, THRESH = e,DATAREC} = \left[\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right], (\text{RTHRESH = r})\right)\end{array}\right]=\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

EXAMPLES:

```
GROUNDCHECK=YES
GROUNDCHECK(GRID=12,SET=(G,N,A),THRESH=1.E-5,DATAREC=YES)=YES
```

DESCRIBERS:

Describer	Meaning
PRINT	Writes output to the print file. (Default)
NOPRINT	Does not write output to the print file.
PUNCH	Writes output to the punch file.
SET	Selects degree-of-freedom set(s). (Default: SET=G).
gid	Reference grid point for the calculation of the rigid body motion.
e	Maximum strain energy which passes the check. The default value is computed by dividing the largest term in the stiffness matrix by 1.E10.

Describer	Meaning
DATAREC	Requests data recovery of grounding forces. (Default: DATAREC=NO)
r	Grounding forces which are larger than the product of (r x largest grounding force) will be printed if DATAREC=YES. (Default=0.1) (0. < r < 1.)

REMARKS:

1. GROUNDCHECK must be specified above the subcase level.
2. SET=N+AUTOSPC uses the stiffness matrix for the n-set with the rows corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.

GSTRAIN

Requests strain output at Gauss points for SOL 401.

Requests strain at Gauss points.

FORMAT:

$$\text{GSTRAIN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
GSTRAIN=ALL
GSTRAIN(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GSTRESS

Requests stress output at Gauss points for SOL 401.

Requests stress at Gauss points.

FORMAT:

$$\text{GSTRESS}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix}\right\}$$

EXAMPLES:

```
GSTRESS=ALL
GSTRESS(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GTHSTRN

Requests thermal strain output at Gauss points for SOL 401.

Requests thermal strain at Gauss points.

FORMAT:

$$\text{GTHSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
GTHSTRN=ALL
GTHSTRN(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcases for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

GUST

Aerodynamic Gust Load Requests

Selects the gust field in an aeroelastic response problem.

FORMAT:

GUST=n

EXAMPLES:

```
GUST=73
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a GUST Bulk Data entry. (Integer>0)

REMARKS:

The choice of transient or frequency response GUST depends upon the type of TLOAD or RLOAD referenced on the selected GUST entry.

HARMONICS

Requests Solution Harmonics

Requests the solution harmonics for cyclic symmetry and axisymmetric models in solutions 114, 115, 116, 118, and 401.

FORMAT:

$$\text{HARMONICS} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

EXAMPLES:

```
HARMONICS=ALL
HARMONICS=32
```

DESCRIBERS:

Describer	Meaning
ALL	All harmonics will be used.
n	Set identification number. The referenced SET lists the harmonic numbers to use. (Integer>0)

REMARKS:

1. Either HARMONICS=ALL or HARMONICS=n is required in cyclic symmetry solutions 114, 115, 116, 118, and in a ANALYSIS=CYCMODES subcase in SOL 401.

   HARMONICS=n is required in a ANALYSIS=FOURIER subcase in SOL 401. HARMONICS=ALL is not supported in a ANALYSIS=FOURIER subcase.

2. *n* is the ID of a SET command which lists the harmonic numbers. For example,

   . . .

```
HARMONICS = 101
SET 101 = 3,5,8,12
...
```

3. The HOUTPUT case control command can be used to request the harmonic numbers for results output.

HDOT

Heat Transfer Rate of Change of Enthalpy Output Request

Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).

FORMAT:

$$\text{HDOT}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
HDOT=5
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates but does not print rate of change of enthalpy.
ALL	Rate of change of enthalpy for all points will be output.
NONE	Rate of change of enthalpy for no points will be output.

Describer	Meaning
n	Set identification of previously appearing SET command. Only rates of change of enthalpy for points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

HDOT=NONE is used to override a previous HDOT=n or HDOT=ALL command.

HOUTPUT

Harmonic Output Request

Requests harmonic output for cyclic symmetry and axisymmetric models in solutions 114, 115, 116, 118, and 401.

FORMAT FOR SOLS 114, 115, 116, 118:

$$\text{HOUTPUT}[(\text{C}, \text{S}, \text{C*}, \text{S*})] = \left\{ \begin{array}{c} \text{ALL} \\ K \end{array} \right\}$$

FORMAT FOR SOL 401:

$$\text{HOUTPUT} = \left\{ \begin{array}{c} \text{ALL} \\ K \end{array} \right\}$$

EXAMPLES:

```
HOUTPUT=ALL
HOUTPUT=5
```

DESCRIBERS:

Describer	Meaning
ALL	All harmonics will be output.
k	Set identification number. The referenced SET lists the harmonics for output. (Integer>0)
C, S, C*, S*	Harmonic coefficients for SOLs 114, 115, 116, 118. Not supported by SOL 401.See Remark 4.

REMARKS:

1. *k* is the ID of a SET command which lists the harmonic numbers for output. For example,

```
...
HOUTPUT = 101
SET 101 = 3,5,8,12
...
```

2. HOUTPUT=ALL requests output for all harmonics specified on the HARMONICS command.

3. In SOLs 114, 115, 116, 118, either the HOUTPUT or NOUTPUT command is required to obtain data recovery in cyclic symmetry analysis.

   In SOL 401, the HOUTPUT command is required to obtain harmonic data recovery in a ANALYSIS=CYCMODES or ANALYSIS=FOURIER subcase.

4. In SOLs 114, 115, 116, 118, C and S correspond to the cosine and sine coefficients when the STYPE field is ROT or AXI on the CYSYM Bulk Data entry. C, S, C*, and S* correspond to the cosine symmetric, sine symmetric, cosine antisymmetric, and sine antisymmetric coefficients respectively, when the STYPE field is DIH on the CYSYM Bulk Data entry.

IC

Transient Initial Condition Set Selection

Selects the initial conditions for transient analyses (SOLs 109, 112, 129, 159, 601, and 701).

FORMAT:

$$\text{IC}\left[\begin{array}{c}\text{PHYSICAL}\\ \text{MODAL}\\ \text{STATSUB[,DIFFK]}\\ \text{TZERO[,DIFFK]}\end{array}\right] = n$$

EXAMPLES:

```
IC = 17
IC(PHYSICAL) = 10
IC(MODAL) = 20
IC(STATSUB) = 30
IC(STATSUB,DIFFK) = 1030
IC(TZERO)= 0
IC(TZERO,DIFFK)= 0
```

DESCRIBERS:

Describer	Meaning
PHYSICAL	Use the TIC bulk entries selected by set *n* as the initial conditions for coordinates involving grid and scalar points (default). See Remarks 4 and 5.
MODAL	Use the TIC bulk entries selected by set *n* as the initial conditions for modal coordinates. See Remarks 4 and 5.
STATSUB	Use the solution of the static analysis subcase *n* as the initial conditions. See Remark 4.

Describer	Meaning
TZERO	Use the static deflection resulting from the loading at time = 0 as the initial condition for a direct transient analysis (SOL 109) or a modal transient analysis (SOL 112). For this option, *n* is not used by the software, although a dummy value must be defined. See Examples. See Remark 4.
DIFFK	Include the effects of differential stiffness in the solution. See Remarks 3 and 4.
n	For the PHYSICAL option, *n* is the set identification number of TIC bulk entries for structural analysis (SOLs 109, 129, 601, and 701) or TEMP and TEMPD bulk entries for heat transfer analysis (SOL 159). For the MODAL option, *n* is the set identification number of TIC bulk entries for modal transient analysis (SOL 112). For the STATSUB option, *n* is the ID of a static analysis subcase (SOL 109 and 112). For the TZERO option, *n* is not used by the software, although a dummy value must be defined. (Integer>0)

REMARKS:

1. TIC bulk entries will not be used (therefore, no initial conditions) unless selected in the case control section.

2. The IC case control command must be defined in the global subcase. However, when the TZERO describer is specified, the initial condition for each subcase is defined by the loads included in that subcase.

3. The DIFFK describer is meaningful only when used in conjunction with the STATSUB or TZERO describers.

4. The following table summarizes the solution sequences in which the describers may be specified.

	SOL 109	SOL 112	SOL 129	SOL 159	SOL 601	SOL 701
PHYSICAL	X		X	X	X	X
MODAL		X				
STATSUB*	X	X				
TZERO*	X	X				

DIFFK*	X	X				

* Not applicable in a DMP solution.

5. Initial condition definitions on extra points are not supported and will be ignored.

INCLUDE

Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

FORMAT:

INCLUDE ’filename’

DESCRIBERS:

Describer	Meaning
filename	External file to be inserted. The ‘*directory_path/filename*’ must begin and end with the single quote character.

EXAMPLES:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE=STATIC ANALYSIS
LOAD=100
INCLUDE ’MYCASE.DATA’
BEGIN BULK
ENDDATA
```

REMARKS:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.

2. Each line has a 72 character limit. Multiple lines can be used when file names are long. The entire string must begin and end with the single quote character.

For example,

D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat

can be defined on multiple lines:

```
INCLUDE 'D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\fold
er8\folder9\mydata.dat'
```

The following input format is also supported:

```
INCLUDE 'D:\folder1\folder2\folder3\
         folder4\folder5\
     folder6\folder7\
        folder8\folder9\mydata.dat'
```

INITS

Selects an initial stress or strain set.

Selects an initial stress or strain set.

FORMAT:

INITS=n

EXAMPLES:

```
INITS=12
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of an INITS or INITADD bulk entry. (Integer>0)

REMARKS:

1. INITS is only supported in SOL 401.
2. The INITS case control must be defined above the subcases (globally), and is reapplied in each static subcase. If you want to include the effects of initial stress or strain including any other load in a normal modes subcase, you should include a static subcase before the sequentially dependent normal modes subcase.

JINTEG

J-Integral computation and output for SOL 401.

Controls the computation and output of the j-integral.

FORMAT:

$$\text{JINTEG}\left[\left(\left[\begin{matrix}\text{PRINT}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
JINTEG=ALL
JINTEG(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print (.f06) file. (Default)
PLOT	Compute output only.
ALL	Compute j-integral.
NONE	Do not compute j-integral.

REMARKS:

1. Only supported in a static subcases for SOL 401.

K2GG

Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrices.

FORMAT:

K2GG=name

EXAMPLES:

```
K2GG=KDMIG
K2GG=KDMIG1, KDMIG2, KDMIG3
K2GG=1.25*KDMIG1, 1.0*KDMIG2, 0.75*KDMIG3
SET 100=K1, K2
K2GG=100
```

DESCRIBERS:

Describer	Meaning
name	Name of a [k^2_{gg}] matrix that is input on the DMIG Bulk Data entry. (Character) Scale factors may be included (see Remarks 4 and 5). See “Matrix Assembly Operations in SubDMAP SEMG” in the *NX Nastran User’s Guide*.

REMARKS:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the stiffness matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG,name entry must contain the integer 6.

4. The associated DMIG matrices can be scaled using either in-line scale factors on K2GG (for example, K2GG=1.25*KDMIG1), using the parameter CK2 (for example, PARAM,CK2,1.25), or both. See “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

   ```
   K2GG=1.25*KDMIG1,1.0*KDMIG2,0.75*KDMIG3
   ```

   The parameter CK2 when defined will scale all K2GG. For example, if PARAM,CK2,1.30 is defined with the K2GG example above, the result would be K2GG=1.30(1.25*KDMIG1 + 1.0*KDMIG2 + 0.75*KDMIG3).

6. The K2GG command is limited to use in the first subcase only.

K2PP

Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrices, which are not included in normal modes.

FORMAT:

K2PP=name

EXAMPLES:

```
K2PP=KDMIG
K2PP=KDMIG1, KDMIG2, KDMIG3
K2PP=1.25*KDMIG1, 1.0*KDMIG2, 0.75*KDMIG3
K2PP=(1.25,0.5)*KDMIG1, (1.0,0.0)*KDMIG2, (0.75,-2.2)*KDMIG3
SET 100=K1, K2
K2PP=100
```

DESCRIBERS:

Describer	Meaning
name	Name of a $[k^2_{pp}]$ matrix that is input on the DMIG Bulk Data entry. (Character) Scale factors may be included (see remarks 6 and 7). See “Formulations of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide*.

REMARKS:

1. DMIG entries will not be used unless selected by the K2PP command.
2. The matrix must be square or symmetric and field 4 on the DMIG,name entry must contain a 1 or 6.

3. It is recommended that PARAM,AUTOSPC,NO be specified. See “Constraint and Mechanism Problem Identification in SubDMAP SEKR” in the *NX Nastran User’s Guide*.

4. K2PP matrices are used only in dynamic response problems. They are not used in normal modes.

5. The K2PP command is supported across subcases. A K2PP command selecting a different DMIG matrix can be defined for each subcase.

6. The associated DMIG matrices can be scaled using in-line scale factors on K2PP (for example, K2PP=1.25*KDMIG1).

7. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

```
K2PP=1.25*KDMIG1,1.0*KDMIG2,0.75*KDMIG3
```

K42GG

Direct Input Structural Damping Matrix Selection

Selects direct input structural damping matrices.

FORMAT:

K42GG=name

EXAMPLES:

```
K42GG=KDMIG
K42GG=KDMIG1, KDMIG2, KDMIG3
K42GG=1.25*KDMIG1, 1.0*KDMIG2, 0.75*KDMIG3
SET 100=K1, K2
K42GG=100
```

DESCRIBERS:

Describer	Meaning
name	Name of a $[k^{4,2}_{gg}]$ matrix that is input on the DMIG Bulk Data entry. (Character) Scale factors may be included (see Remarks 4 and 5). See “Matrix Assembly Operations in SubDMAP SEMG” in the *NX Nastran User’s Guide*.

REMARKS:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the structural damping matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG,name entry must contain the integer 6.

4. The associated DMIG matrices can be scaled using either in-line scale factors on K42GG (for example, K42GG=1.25*KDMIG1), using the parameter CK42 (for example, PARAM,CK42,1.25), or both. See “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

   ```
   K42GG=1.25*KDMIG1,1.0*KDMIG2,0.75*KDMIG3
   ```

   The parameter CK42 when defined will scale all K42GG. For example, if PARAM,CK42,1.30 is defined with the K42GG example above, the result would be K42GG=1.30(1.25*KDMIG1 + 1.0*KDMIG2 + 0.75*KDMIG3).

6. The K42GG command is limited to use in the first subcase only.

LABEL

Output Label

Defines a character string that will appear on the third heading line of each page of printer output.

FORMAT:

LABEL=label

EXAMPLES:

```
LABEL=DEMONSTRATION PROBLEM
```

DESCRIBERS:

Describer	Meaning
label	Any character string.

REMARKS:

1. LABEL appearing at the subcase level will label output for that subcase only.
2. LABEL appearing before all subcases will label any outputs that are not subcase dependent.
3. If no LABEL command is supplied, the label line will be blank.
4. LABEL information is also placed on plotter output as applicable. Only the first 65 characters will appear.

LINE

Maximum Lines Per Printed Page

Defines the maximum number of output lines per printed page.

FORMAT:

LINE=n

EXAMPLES:

```
LINE=35
```

DESCRIBERS:

Describer	Meaning
n	Maximum number of output lines per page. (Integer>0; Default = 50)

REMARKS:

1. For 11 inch paper, 50 lines per page is recommended; for 8-1/2 inch paper, 35 lines per page is recommended.
2. The NASTRAN statement keyword NLINES may also be used to set this value. See the “nastran Command and NASTRAN Statement” .

LOAD

External Static Load Set Selection

Selects an external static load set.

For SOLs 601 and 701, selects a constant load.

FORMAT:

LOAD=n

EXAMPLES:

```
LOAD=15
```

DESCRIBERS:

Describer	Meaning
n	Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one ACCEL, ACCEL1, DAREA, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADE1, PLOADX1, QBDY1, QBDY2, QBDY3, QHBDY, QVECT, QVOL, RFORCE, RFORCE1, SELOAD, or SLOAD entry. (Integer>0)
n (For SOLs 601 and 701)	Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, LOAD, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1 (SOL 601), RFORCE or SPCD entry. (Integer>0)

REMARKS:

1. ACCEL, ACCEL1, GRAV entries cannot have the same set identification number as any other loading entries. If you want to combine these with other static loads, you must use the LOAD bulk entry.

2. LOAD is only applicable in linear and nonlinear statics, inertia relief, differential stiffness, buckling, and heat transfer problems.

3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) loads.

4. Static, thermal, and element deformation loads should have unique set identification numbers.

REMARKS RELATED TO SOLS 601 AND 701:

1. In SOL 601, LOAD may be used in a static or transient analysis.

2. Both DLOAD and LOAD case control commands can be specified in a SOL 601 or 701 analysis. DLOAD is used to select time-dependent loads and LOAD is used to select constant loads.

LOADSET

Static Load Set Selection

Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.

FORMAT:

LOADSET=n

EXAMPLES:

```
LOADSET=100
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of at least one LSEQ Bulk Data entry. (Integer>0)

REMARKS:

1. When used in superelement analysis, this command must be used for all superelements. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEID IDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique DAREA IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
2. When the LOADSET command is used in superelement statics, the residual structure should have as many loading conditions as the number of unique EXCITEID sets defined on all LSEQ entries. The subcases after the first should contain only SUBTITLE and LABEL information and residual structure output requests. SUBTITLE and LABEL information for all superelements will be obtained from the residual structure.

3. In SOL 101, the design sensitivity output will identify all expanded subcases by a sequence of unique integers beginning with n.

4. In the nonlinear static solution sequences (SOLs 106 and 153) the LOADSET must appear above all subcases and only one LOADSET may be specified.

5. Only one LOADSET command is allowed per superelement.

6. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LOADSET request in Case Control, all static loads whose load set IDs match the EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.

M2GG

Direct Input Mass Matrix Selection

Selects direct input mass matrices.

FORMAT:

M2GG=name

EXAMPLES:

```
M2GG=MDMIG
M2GG=MDMIG1, MDMIG2, MDMIG3
M2GG=1.25*MDMIG1, 1.0*MDMIG2, 0.75*MDMIG3
SET 100=M1, M2
M2GG=100
```

DESCRIBERS:

Describer	Meaning
name	Name of a [M^2_{gg}] matrix that is input on the DMIG Bulk Data entry. Scale factors may be included (see Remarks 4 and 5). See “Matrix Assembly Operations in SubDMAP SEMG” in the *NX Nastran User’s Guide*. (Character)

REMARKS:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the mass matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG, name entry must contain a 6.

4. The associated DMIG matrices can be scaled using either in-line scale factors on M2GG (for example, M2GG=1.25*MDMIG1), using the parameter CM2 (for example, PARAM,CM2,1.25), or both. See “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

```
M2GG=1.25*MDMIG1,1.0*MDMIG2,0.75*MDMIG3
```

   The parameter CM2 when defined will scale all M2GG. For example, if PARAM,CM2,1.30 is defined with the M2GG example above, the result would be M2GG=1.30(1.25*MDMIG1 + 1.0*MDMIG2 + 0.75*MDMIG3).

6. M2GG input is not affected by PARAM,WTMASS. M2GG input must either be in consistent mass units or PARAM,CM2 may be used.

7. The M2GG command is limited to use in the first subcase only.

M2PP

Direct Input Mass Matrix Selection

Selects direct input mass matrices, which are not included in normal modes.

FORMAT:

M2PP=name

EXAMPLES:

```
M2PP=MDMIG
M2PP=MDMIG1, MDMIG2, MDMIG3
M2PP=1.25*MDMIG1, 1.0*MDMIG2, 0.75*MDMIG3
M2PP=(1.25,0.5)*MDMIG1, (1.0,0.0)*MDMIG2, (0.75,-2.2)*MDMIG3
SET 100=M1, M2
M2PP=100
```

DESCRIBERS:

Describer	Meaning
name	Name of a [M^2_{pp}] matrix that is input on the DMIG Bulk Data entry. Scale factors may be included (see remarks 7 and 8). See “Dynamic Reduction and Component Mode Synthesis in SubDMAP SEMR3” in the *NX Nastran User’s Guide*. (Character)

REMARKS:

1. DMIG entries will not be used unless selected by the M2PP input.
2. M2PP input is not affected by PARAM,WTMASS. M2PP input must be in consistent mass units.
3. The matrix must be square or symmetric and field 4 on the DMIG, name entry must contain a 1 or 6.

4. It is recommended that PARAM,AUTOSPC,NO be specified. See “Constraint and Mechanism Problem Identification in SubDMAP SEKR” in the *NX Nastran User’s Guide*.

5. M2PP matrices are used only in dynamic response problems. They are not used in normal modes.

6. The M2PP command is supported across subcases. A M2PP command selecting a different DMIG matrix can be defined for each subcase.

7. The associated DMIG matrices can be scaled using in-line scale factors on M2PP (for example, M2PP=1.25*MDMIG1).

8. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

```
M2PP=1.25*MDMIG1,1.0*MDMIG2,0.75*MDMIG3
```

MASTER

Redefine the MASTER Subcase

Allows the redefinition of a MASTER subcase.

FORMAT:

SUBCASE n

MASTER

EXAMPLES:

```
SUBCASE 10
MASTER
```

REMARKS:

1. All commands in a MASTER subcase apply to the following subcases until a new MASTER subcase is defined.

2. Suppose that superelement 10 has SPC set 10, MPC set 10, and LOAD sets 101 and 102. Suppose also that superelement 20 has SPC set 20, MPC set 20, and LOAD sets 201 and 202.

   The following setup specifies the required subcase structure:

```
   TITLE=MY MODEL
   DISP=ALL
   SEALL=ALL
SUBCASE 101
   MASTER
   SPC=10
   MPC=10
   SUPER=10, 1
   LOAD=101
   LABEL=SUPER 10
   ESE=ALL
SUBCASE 102
   LOAD=102
   SUPER=10, 2
SUBCASE 201
```

```
   MASTER
   SPC=20
   MPC=20
   SUPER=20, 1
   LOAD=201
   LABEL=SUPER 20
SUBCASE 202
   LOAD=202
   SUPER=20, 2
```

3. MASTER may also be used to advantage with multiple boundary condition Case Control setups. Suppose that constraint sets 10 and 20 are to be solved with three loading conditions each: 1, 2, and 3 and 4, 5, and 6, respectively. The following Case Control Section may be used


```
TITLE=MULTIPLE BOUNDARY CONDITIONS
DISP=ALL
SYM 1
MASTER
   SPC=10
   LOAD=1
SYM 2
   LOAD=2
SYM 3
   LOAD=3
SYM 4
   MASTER
   SPC=20
   LOAD=4
SYM 5
   LOAD=5
SYM 6
   LOAD=6
SYMCOM 10
   SYMSEQ=1., 1., 1., -1., -1., -1.
SYMCOM 20
   SYMSEQ=-1., -1., -1., 1., 1., 1.
```

4. The MASTER command must appear immediately after a SUBCASE or SYM command.

MAXLINES

Maximum Number of Output Lines

Sets the maximum number of output lines.

FORMAT:

MAXLINES=n

EXAMPLES:

```
MAXLINES=150000
```

DESCRIBERS:

Describer	Meaning
n	Maximum number of output lines allowed. (Integer>0; Default=999999999)

REMARKS:

1. Any time MAXLINES is exceeded, the program will terminate.
2. MAXLINES does not override any system parameters such as those on Job Control Language commands.
3. MAXLINES may also be specified on the NASTRAN statement with SYSTEM(14). See the “nastran Command and NASTRAN Statement”..
4. The code actually counts the number of pages and assumes that the number of lines output is the number of lines allowed per page, specified by the “LINES” command, times the number of pages.

MAXMIN

MAXMIN Survey Output Request

Specifies options for max/min surveys of certain output data associated with grid points. (SOLs 101, 103, 105, 109, 114, 115, 129, 144, 145, 146, 187, and 200)

FORMAT:

$$\text{MAXMIN}\left(\left[\left\{\begin{matrix}\text{MAX}\\ \text{BOTH}\\ \text{MIN}\\ \text{VMAG}\end{matrix}\right\}[=\text{num}]\right],\left[\text{CID}=\left\{\begin{matrix}\text{GLOBAL}\\ \text{BASIC}\\ \text{cid}\end{matrix}\right\}\right],\right.$$
$$\left.\text{oplist},[\text{COMP}=\text{list}]\right)=\left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
MAXMIN (BOTH=10,CID=1000,DISP,COMP=T1/T3)=501
```

DESCRIBERS:

Describer	Meaning
MAX	Specifies only maximum values are to be output. See Remark 1 .
MIN	Specifies only minimum values are to be output. See Remark 1 .
BOTH	Specifies both maximum and minimum values are to be output. See Remark 1 .
VMAG	Specifies vector magnitude resultants are to be output. See Remark 2 .

Describer	Meaning
num	The maximum number of values that will be output. See Remark 3 . (Integer>0 , default=5)
CID	Specifies the coordinate system frame of reference in which the max/min values will be output. See Remarks 1 and 3.
GLOBAL	Requests output in the global coordinate system frame of reference.
BASIC	Requests output in the basic coordinate system frame of reference.
cid	Requests output in the local coordinate system defined by cid. (Integer>0)
oplist	Specifies a list of one or more standard Case Control output requests for which max/min results are to be produced. The list may include any of DISP, SPCDF, OLOAD, MPCF, VELO, ACCE or ALL. See Remark 6 . (Character, no default)
list	Specifies a list of grid point degree of freedom (DOF) component directions that will be included in the max/min survey output. The components are separated by slashes and are selected from T1, T2, T3, R1, R2, R3. See Remarks 4 and 5. (Character, default=/T1 /T2 /T3 /R1 /R2 /R3)
ALL	MAXMIN survey results for all points will be output.
NONE	MAXMIN survey results for no points will be output.
n	Set identification of a previously appearing SET command. The max/min results survey will be output only for the points specified in SET n. (Integer>0)

REMARKS:

1. The MAXMIN command produces an algebraically ascending sorted list of the output quantities specified for all of the points in the selected set. MAX refers to the largest magnitude positive values, while MIN refers to the largest magnitude negative values. The output format is similar to that of displacement output. All components will be output for a grid point and the order of the grid points will be

in sort on the particular component that was surveyed. The output title contains the identification number of the SET of points participating in the max/min output, the coordinate system frame of reference, the number of MAX and MIN values output and the component that was surveyed. When the output being surveyed is in the global output coordinate system reference frame and BASIC or a local output coordinate system is specified as cid, both the sorted system output and the original reference system output are displayed for the grid point if these systems are different.

2. Vector magnitude results are computed for both translations and rotations and displayed under the T1 and R1 column headings. The presence of the COMP keyword is ignored.

3. The default value of 5 generates a minimum of 10 output lines for the BOTH option. There will be 5 maximum values and 5 minimum values produced. In addition, if coordinate system are involved, both surveyed and original data will be output. This could result in as many as 10 more lines of output for each surveyed component.

4. Multiple MAXMIN commands may be specified for a subcase. This permits different output quantities to have different MAXMIN specification within a subcase. For example,

```
SET 501=1,3,5,7 THRU 99, 1001,2001
MAXMIN (DISP, COMP=T3)=501
MAXMIN (SPCF, COMP=T1/R3)=ALL
```

specifies different components and output sets for displacements and forces of single point constraint. When multiple component searches are specified using the COMP keyword, separate output tables are produced that contain the results of the max/min survey for each of the components in the list. For example, COMP=T1/R3 produces two surveys , one with the T1 component surveyed and the other with the R3 component surveyed.

5. Scalar point output is included only if component T1 is included in the list.

6. MAXMIN output will only be generated for items in the oplist when there is an associated case control command present. For example, a DISP Case Control command must be present in order for the MAXMIN (DISP)=ALL command to produce output. Use of the ALL keywords for the oplist requests MAXMIN output for all of the output commands acceptable to MAXMIN that are present in case control.

MBDEXPORT

Multi-Body Dynamics Export

Generates interface file for third-party multi-body dynamics and control system software during a solution 103, 111, or 112.

FORMAT:

$$\text{MBDEXPORT}\left[\begin{matrix}\text{RECURDYN}\\ \text{ADAMS}\\ \text{SIMPACK}\\ \text{OP4}=\text{unit}\\ \text{MATLAB}\end{matrix}\right],\left[\begin{matrix}\text{STANDARD}\\ \text{STATESPACE}\end{matrix}\right],\left[\text{FLEXBODY}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],$$

$$\left[\text{FLEXONLY}=\left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}\right],\left[\text{MINVAR}=\left\{\begin{matrix}\text{PARTIAL}\\ \text{CONSTANT}\\ \text{FULL}\\ \text{NONE}\end{matrix}\right\}\right],\left[\text{PSETID}=\left\{\begin{matrix}\text{NONE}\\ \text{setid}\\ \text{ALL}\\ \text{sktunit}\end{matrix}\right\}\right],$$

$$\left[\text{OUTGSTRS}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{OUTGSTRN}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{RECVROP2}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],$$

$$\left[\text{CHECK}=\left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right],\left[\text{NONCUP}=\left\{\begin{matrix}-1\\ -2\end{matrix}\right\}\right]$$

The general examples, describers, and remarks are an overview for all interface types. Below this are specific examples, describers, and remarks sections for each interface type.

GENERAL EXAMPLES:

```
MBDEXPORT ADAMS STANDARD FLEXBODY=YES FLEXONLY=NO
MBDEXPORT FLEXBODY=YES MINVAR=FULL
MBDEXPORT OP4=22 STANDARD FLEXBODY=YES
```

MBDEXPORT

```
MBDEXPORT OP4=22 STATESPACE FLEXBODY=YES
MBDEXPORT MATLAB STANDARD FLEXBODY=YES
MBDEXPORT MATLAB STATESPACE FLEXBODY=YES
MBDEXPORT SIMPACK FLEXBODY=YES
```

GENERAL DESCRIBERS:

Describer	Meaning
RECURDYN	Generate RecurDyn Flex Input (RFI) file. (default)
ADAMS	Generate Adams Interface Modal Neutral File (MNF).
SIMPACK	Generate SIMPACK Flexible Body Input (FBI) file.
OP4	Generate OP4 file.
MATLAB	Generate MATLAB script file.
STANDARD	Matrices are based on standard second-order differential equations of motion. (default)
STATESPACE	Matrices are based on first-order differential equations that represent the equations of motion, and are suitable for use with control system software.

GENERAL REMARKS:

1. Only one choice of RECURDYN, ADAMS, SIMPACK, OP4, or MATLAB is allowed and must immediately follow the MBDEXPORT command.
2. The describers can be truncated to the first 4 characters.
3. STATESPACE is not valid for RECURDYN, ADAMS or SIMPACK.
4. MBDEXPORT must appear above the subcase level.

The information from this point on is specific to each interface type.

RECURDYN STANDARD DESCRIBERS:

Describer	Meaning
FLEXBODY	Requests the generation of RFI (required).
NO	Standard NX Nastran solution without RFI creation. (default)
YES	RFI generation requested.
FLEXONLY	Determines if DMAP solution and data recovery runs or not after RFI creation is complete.
YES	Only RFI creation occurs. (default)
NO	RFI file creation occurs along with standard DMAP solution and data recovery.
MINVAR	Determines how mass invariants are computed.
PARTIAL	Mass invariants 6 and 8 are not computed. (default)
CONSTANT	Mass invariants 1,2,3 and 9 are computed.
FULL	All nine mass invariants are computed.
NONE	No mass invariants are computed.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) (including PLOTEL) whose connectivity is exported into the RFI. See Remark 16.
NONE	No specific sets are selected, thus all grids, geometry and associated modal data are written to RFI. (default)
setid	The connectivity of a specific element set is used to export face geometry.
ALL	The connectivity of all element sets are used to export face geometry.
OUTGSTRS	Determines if grid point stress is written to RFI.

Describer	Meaning
NO	Do not write grid point stress to RFI. (default)
YES	Write grid point stress to RFI.
OUTGSTRN	Determines if grid point strain is written to RFI.
NO	Do not write grid point strain to RFI. (default)
YES	Write grid point strain to RFI.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of RecurDyn/Flex results.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 20)
NO	No debug output will be written. (default)
YES	Debug output will be written.

RECURDYN STANDARD REMARKS:

1. The creation of the RecurDyn Flex Input file is applicable in a non-restart SOL 103, 111, or 112 analysis only. RFI files are named ‘jid_seid.rfi’, where seid is the integer number of the superelement (0 for residual). These files are located in the same directory as the jid.f06 file.

2. The creation of the RecurDyn Flex Input file is initiated by MBDEXPORT RECURDYN FLEXBODY=YES (other describers are optional) and the inclusion of the bulk entry DTI,UNITS.

3. Because RecurDyn is not a unitless code, the Data Table Input bulk entry DTI,UNITS is required for an MBDEXPORT RECURDYN FLEXBODY=YES run. The DTI,UNITS entry specifies the system of units of the original NX Nastran input file, and is then included with the data written to the RFI file. NX Nastran does not

do a units conversion of the nastran data when writing the RFI file. Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

All entries are required. Acceptable character strings are listed below.

Mass:

KG - kilogram

LBM – pound-mass (0.45359237 kg)

SLUG – slug (14.5939029372 kg)

GRAM – gram (1E-3 kg)

OZM – ounce-mass (0.02834952 kg)

KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

MGG – megagram (1E3 kg)

MG – milligram (1E-6 kg)

MCG – microgram (1E-9 kg)

NG – nanogram (1E-12 kg)

UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

CN – centinewton (1E–2 N)

P – poundal (0.138254954 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

4. Because DTI,UNITS determines all units for the RFI, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the RFI. For example, if the model mass is in kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1 ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the RFI would be: “DTI,UNITS,1,KG,N,M,S”.

5. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points; or for a residual-only type model, you can use NX Nastran ASET bulk entries to define the attachment points.

6. The eight mass variants are:

$$^{1}\underset{1x1}{I} = \sum_{p=1}^{N} m_p$$

$$^{2}\underset{3x1}{I} = \sum_{p=1}^{N} m_p s_p$$

$$^{3}\underset{3x3}{I} = \sum_{p=1}^{N} (m_p \tilde{s}_p \tilde{s}_p - I_p)$$

$$^{4}\underset{3xM}{I_j} = \sum_{p=1}^{N} m_p \Phi_p \quad j = 1,\ldots,M$$

$$^{5}\underset{3x3}{I_j} = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj} \quad j = 1,\ldots,M$$

$$^{6}\underset{3x3}{I_{jk}} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \phi_{pk} \quad j,k = 1,\ldots,M$$

$$^{7}\underset{3xM}{I} = \sum_{p=1}^{N} m_p \tilde{s}_p \Phi_p + \sum_{p=1}^{N} I_p \Phi_p^*$$

$$^{8}\underset{3xM}{I_j} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1,\ldots,M$$

$s_p = [xyz]^T$ are the coordinates of grid point p in the basic coordinate system.

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix} = \text{skew symmetric vector cross product operator.}$$

φ_p = partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid *p*.

I_p = inertia tensor *p*.

φ_p^* = partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid *p*.

$\tilde{\phi}_{pf}$ = skew-symmetric matrix formed for each grid translational degree of freedom for each mode.

M = number of modes.

N = number of grids.

7. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTS (ns) should be at least ns=n+(6+p), assuming that residual flexibility is on. In the above equation for ns, the number of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL bulk entries; the number of load cases is p. In general, you cannot have too many SPOINTs, as excess ones will be truncated with no performance penalty.

8. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in a more accurate representation of the component shapes with little additional computational effort.

9. OMIT or OMIT1 bulk entries are not supported.

10. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.

11. CBEND elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

12. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an RFI. It must have consistent units with regard to the DTI,UNITS bulk entry. Before generating the RFI, NX Nastran will appropriately scale the WTMASS from the physical mass matrix and mode shapes.

13. There is a distinction between how an MBDEXPORT RECURDYN FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

14. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran,

the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with MBDEXPORT RECURDYN FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This caution only applies to a superelement run and not to a residual-only or a part superelement run.

15. OUTGSTRS and OUTGSTRN entries require the use of standard NX Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN= commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain print to the NX Nastran .f06 file.

16. To reduce the FE mesh detail for dynamic simulations, PSETID can include the ID of a SET entry. The SET entry lists PLOTEL or element IDs, whose connectivity is exported into the RFI to display the components in RecurDyn. This option can significantly reduce the size of the RFI without compromising accuracy in the FunctionBay simulation providing that the mass invariant computation is requested. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids are attached only to a-set or exterior grids), a SEELT bulk entry must be specified to keep that display element with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

    If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements that are used to select grids to display the component in RecurDyn. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID then represents a list of OUTPUT(PLOT) defined elements' sets. The union of which will be used to define a set of PLOTELs or other elements used to select grids to display the component in RecurDyn. If you wish to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

    The following element types are not supported for writing to an RFI, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CMASS3, CMASS4, CRAC2D, CRAC3D, CTWIST, CWEDGE, CWELD, and GENEL.

17. Typical NX Nastran data entry requirements are described below.

    Typical Parameters:

    - PARAM,RESVEC,character_value – controls calculation of residual vector modes.

- PARAM,GRDPNT,value - mass invariants ^1I, ^2I, and ^3I will be computed using results of NX Nastran grid point weight generator execution in the basic coordinate system.

Typical Case Control:

- MBDEXPORT RECURDYN FLEXBODY=YES is required for RFI generation.
- METHOD=n is required before or in the first subcase for modal solutions.
- SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.
- OUTPUT(PLOT) is necessary to define elements used to select grids to display the component in RecurDyn when PSETID=ALL or setid.

  SET n=list of elements (including PLOTELs) is used to select grids to display the component.
- OUTPUT(POST) is necessary to define volume and surface for grid stress or strain shapes.

  SET n=list is a list of elements for surface definition for grid stress or strain shapes.

  Stress and strain data in the RFI is limited to the six components (that is, 3 normal and 3 shear) for a grid point for a given mode.

  SURFACE n SET n NORMAL z3 is used to define a surface for writing stress and strain data. Only one FIBER selection is allowed for each SURFACE, thus the use of the FIBER ALL keyword on the SURFACE case control command will write stresses to the RFI at the Z1 fiber location only.

  Because the FIBER keyword only applies to stresses, strain data will always be written to the RFI at the MID location.

  Stress and strain data at grid points can only be written to the RFI for surface and volume type elements (for example, CQUAD and CHEXA).

  VOLUME n SET n is a volume definition.

  The default SYSTEM BASIC is required with SURFACE or VOLUME.
- STRESS(PLOT) is necessary for stress shapes.
- STRAIN(PLOT) is necessary for strain shapes.
- GPSTRESS(PLOT) is necessary for grid point stress shapes to be included in the RFI.

- GPSTRAIN(PLOT) is necessary for grid point strain shapes to be included in the RFI.

  Typical Bulk Data:

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for RFI generation. For input files containing superelements, this command must reside in the main bulk data section.

- SPOINT,id_list defines and displays modal amplitude.

- SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.

- SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.

- RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.

- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).

- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main bulk data section.

- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).

- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).

- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in RecurDyn.

- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

18. MBDEXPORT and ADAMSMNF case control entries cannot be used in the same analysis run. In other words, a RecurDyn RFI file or an Adams MNF file can be generated during a particular NX Nastran execution, but not both files at the same time. Attempting to generate both files in the same analysis will cause an error to be issued and the execution to be terminated.

19. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from an RecurDyn/Flex analysis. This option requires the following assignment command:

ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be used as input into an NX Nastran SOL 103 run using the MBDRECVR case control capability to perform results recovery from an RecurDyn/Flex analysis. FLEXBODY=YES is required with its use.

The data blocks output are:

MGGEW - physical mass external sort with weight mass removed
MAAEW - modal mass
KAAE - modal stiffness
CMODEXT - component modes.

This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

If differential stiffness is included, the static portion of the results will not be included in the recovered results when using MBDRECVR.

20. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06.

21. The MBDEXPORT data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using MBDEXPORT. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

22. Preload conditions are not supported.

23. To request differential stiffness, include a static subcase that contains the stress-stiffening loads. In another subcase include STATSUB = n where n is the number of the static subcase.

ADAMS STANDARD DESCRIBERS:

Describer	Meaning
FLEXBODY	Requests the generation of MNF.
NO	Standard NX Nastran solution without MNF creation. (default)

Describer	Meaning
YES	MNF generation requested.
FLEXONLY	Determines if DMAP solution runs or not after MNF creation is complete.
YES	Only MNF creation occurs. (default)
NO	MNF file creation occurs along with standard DMAP solution.
MINVAR	Determines how mass invariants are computed.
PARTIAL	Mass invariants 5 and 9 are not computed. (default)
CONSTANT	Mass invariants 1,2,6 and 7 are computed.
FULL	All nine mass invariants are computed.
NONE	No mass invariants are computed.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) section (including PLOTEL) or on a sketch file whose connectivity is exported to face geometry to be used in Adams. See Remark 15.
NONE	All grids, geometry and associated modal data is written to MNF. (default)
setid	The connectivity of a specific element set is used to export face geometry.
ALL	The connectivity of all element sets are used to export face geometry.
sktunit	The connectivity of element faces defined on a sketch file is used to export face geometry. Note that the value must be a negative number to distinguish it from a setid value.
OUTGSTRS	Determines if grid point stress is written to MNF.
NO	Do not write grid point stress to MNF. (default)
YES	Write grid point stress to MNF.

Describer	Meaning
OUTGSTRN	Determines if grid point strain is written to MNF.
NO	Do not write grid point strain to MNF. (default)
YES	Write grid point strain to MNF.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of Adams/Flex results.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 18)
NO	No debug output will be written. (default)
YES	Debug output will be written.
NONCUP	Modal damping output control. See Remark 20.
–1	Output the full equivalent modal viscous damping matrix (default).
–2	Output only diagonal values of the equivalent modal viscous damping matrix.

ADAMS STANDARD REMARKS:

1. The creation of the Adams MNF, which is applicable in a non-restart SOL 103, 111, or 112 analysis only, is initiated by MBDEXPORT ADAMS FLEXBODY=YES (other describers are optional) and the inclusion of the bulk entry DTI,UNITS. MNF files are named ‘jid_seid.mnf’, where seid is the integer number of the superelement (0 for residual). The location of these files is the same directory as the jid.f06 file.

2. Because Adams is not a unitless code, the Data Table Input bulk entry DTI,UNITS is required for an MBDEXPORT ADAMS FLEXBODY=YES run. The DTI,UNITS entry specifies the system of units of the original NX Nastran input file, and is then included with the data written to the MNF file. NX Nastran does not do a units

conversion of the nastran data when writing the MNF file. Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

All entries are required. Acceptable character strings are listed below.

Mass:

KG - kilogram

LBM – pound-mass (0.45359237 kg)

SLUG – slug (14.5939029372 kg)

GRAM – gram (1E-3 kg)

OZM – ounce-mass (0.02834952 kg)

KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

MGG – megagram (1E3 kg)

MG – milligram (1E-6 kg)

MCG – microgram (1E-9 kg)

NG – nanogram (1E-12 kg)

UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

3. Because DTI,UNITS determines all units for the MNF, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the MNF. For example, if the model mass is in kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1, ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the MNF would be: “DTI,UNITS,1,KG,N,M,S”.

4. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points. For a residual-only type model, you can use standard NX Nastran ASET bulk entries to define the attachment points.

5. The nine mass variants are:

$$\underset{1 \times 1}{{}^{1}I} = \sum_{p=1}^{N} m_p$$

$$\underset{3 \times 1}{{}^{2}I} = \sum_{p=1}^{N} m_p s_p$$

$$\underset{3 \times M}{{}^{3}I_j} = \sum_{p=1}^{N} m_p \Phi_p \quad j = 1, \ldots, M$$

$$\underset{3 \times M}{{}^{4}I} = \sum_{p=1}^{N} m_p \tilde{s}_p \bar{\Phi}_p + I_p \Phi_p^*$$

$$\underset{3 \times M}{{}^{5}I_j} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1, \ldots, M$$

$$\underset{M \times M}{{}^{6}I} = \sum_{p=1}^{N} m_p \Phi_p^T \Phi_p + \Phi_p^{*T} I_p \Phi_p^*$$

$$\underset{3 \times 3}{{}^{7}I} = \sum_{p=1}^{N} m_p \tilde{s}_p^T \tilde{s}_p + I_p$$

$$\underset{3 \times 3}{{}^{8}I_j} = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj} \quad j = 1, \ldots, M$$

$$\underset{3 \times 3}{{}^{9}I_{jk}} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \phi_{pk} \quad j, k = 1, \ldots, M$$

$s_p = [xyz]^T$ are the coordinates of grid point p in the basic coordinate system.

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix} = \text{skew symmetric vector cross product operator.}$$

φ_p = partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid *p*.

I_p = inertia tensor *p*.

φ_p^* = partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid *p*.

$\tilde{\phi}_{pf}$ = skew-symmetric matrix formed for each grid translational degree of freedom for each mode.

M = number of modes.

N = number of grids.

6. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTS (ns) should be at least ns=n+(6+p), assuming that residual flexibility is on. In the above equation for ns, the number of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL bulk entries; the number of load cases is p. In general, you cannot have too many SPOINTs, as excess ones are truncated with no performance penalty.

7. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in a more accurate representation of the component shapes at little additional cost.

8. OMIT or OMIT1 bulk entries are not supported.

9. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.

10. CBEND elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

11. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an MNF. It must have consistent units with regard to the DTI,UNITS bulk entry. Before generating the MNF, NX Nastran will appropriately scale the WTMASS from the physical mass matrix and mode shapes.

12. There is a distinction between how an MBDEXPORT ADAMS FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

13. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran, the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with MBDEXPORT ADAMS FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This caution only applies to a superelement run and not to a residual-only or a part superelement run.

14. OUTGSTRS and OUTGSTRN entries require the use of standard NX Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN= commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain print to the NX Nastran .f06 file.

15. To reduce the FE mesh detail for dynamic simulations, PSETID (on the MBDEXPORT Case Control command) defined with a SET entry (i.e. setid) is used to define a set of PLOTELs or other elements used to select grids to display the components in Adams. This option can significantly reduce the size of the MNF without compromising accuracy in the Adams simulation providing that the mass invariant computation is requested. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids attached only to a-set or exterior grids), a SEELT bulk entry must be specified to keep that display element with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

    If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements used to select grids to display the component in Adams. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements' sets, the union of

which will be used to define a set of PLOTELs or other elements used to select grids to display the component in Adams. If the user wishes to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

The following element types are not supported for writing to an MNF, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CMASS3, CMASS4, CRAC2D, CRAC3D, CTWIST, CWEDGE, CWELD, and GENEL.

PSETID can also point to a sketch file using PSETID= – sktunit, where sktunit references an ASSIGN statement of the form:

```
ASSIGN SKT='sketch_file.dat',UNIT=sktunit.
```


The grids defined for the elements' faces in the sketch file, along with *all* external (i.e. boundary) grids for the superelements, will be the only grids (and their associated data) written to the MNF.

The format of the sketch file, which describes the mesh as a collection of faces, must be as follows:

```
face_count
face_1_node_count face_1_nodeid_1 face_1_nodeid_2 ...
face_2_node_count face_2_nodeid_1 face_2_nodeid_2 ...

<etc>
```

Faces must have a node count of at least two. For example, a mesh comprised of a single brick element might be described as follows:

```
6
4 1000 1001 1002 1003
4 1007 1006 1005 1004
4 1000 1004 1005 1001
4 1001 1005 1006 1002
4 1002 1006 1007 1003
4 1003 1007 1004 1000
```

Alternatively, the mesh might be described as a stick figure using a collection of lines (two node faces), as shown below:

```
8
2 101 102
2 102 103
2 103 104
2 104 105
2 105 106
2 106 107
2 107 108
2 108 109
```

16. Typical NX Nastran data entry requirements are described below.

    Typical Parameters:

    - PARAM,RESVEC,character_value – controls calculation of residual vector modes.
    - PARAM,GRDPNT, value - mass invariants ^1I, ^2I, and ^7I will be computed using results of NX Nastran grid point weight generator execution in the basic coordinate system.

    Typical Case Control:

    - MBDEXPORT ADAMS FLEXBODY=YES is required for MNF generation.
    - METHOD=n is required before or in the first subcase for modal solutions.
    - SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.
    - OUTPUT(PLOT) is necessary to define elements used to select grids to display the component in Adams when PSETID=ALL or setid.

      SET n=list of elements (including PLOTELs) is used to select grids to display the component.
    - OUTPUT(POST) is necessary to define volume and surface for grid stress or strain shapes.

      SET n=list is a list of elements for surface definition for grid stress or strain shapes.

      Stress and strain data in the MNF is limited to the six components (i.e. 3 normal and 3 shear) for a grid point for a given mode.

      SURFACE n SET n NORMAL z3 is used to define a surface for writing stress and strain data. Only one FIBER selection is allowed for each SURFACE, thus the use of the FIBRE ALL keyword on the SURFACE case control command will write stresses to the MNF at the Z1 fiber location only.

      Because the FIBRE keyword only applies to stresses, strain data will always be written to the MNF at the MID location.

      Stress and strain data at grid points can only be written to the MNF for surface and volume type elements (e.g. CQUAD and CHEXA).

      VOLUME n SET n is a volume definition.

      The default SYSTEM BASIC is required with SURFACE or VOLUME.
    - STRESS(PLOT) is necessary for stress shapes.

- STRAIN(PLOT) is necessary for strain shapes.
- GPSTRESS(PLOT) is necessary for grid point stress shapes to be included in the MNF.
- GPSTRAIN(PLOT) is necessary for grid point strain shapes to be included in the MNF.

  Typical Bulk Data:

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for MNF generation. For input files containing superelements, this command must reside in the main bulk data section.
- SPOINT,id_list defines and displays modal amplitude.SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.
- SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.
- RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.
- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).
- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main bulk data section.
- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).
- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).
- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in Adams.
- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

17. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from an Adams/Flex analysis. This option requires the following assignment command:

    ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

    be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be

used as input into an NX Nastran SOL 103 run using the MBDRECVR case control capability to perform results recovery from an Adams/Flex analysis. FLEXBODY=YES is required with its use.

The data blocks output are:

MGGEW - physical mass external sort with weight mass removed
MAAEW - modal mass
KAAE - modal stiffness
CMODEXT - component modes.

This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

If differential stiffness is included, the static portion of the results will not be included in the recovered results when using MBDRECVR.

18. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

19. The MBDEXPORT data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using MBDEXPORT. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

20. If any damping is defined in the model, an equivalent modal viscous damping will be determined for each mode and written to the MNF. This equivalent modal viscous damping is defined as:

    $D = \psi^T B_e \psi$

    where D is the equivalent modal viscous damping matrix, ψ is the eigenvector matrix, and B_e is the equivalent viscous damping matrix.

    The equivalent viscous damping matrix is given by:

    $$B_e = B^1_{AA} + B^2_{AA} + \frac{G}{W3} K^1_{AA} + \frac{1}{W4} K^4_{AA}$$

    where G, $W3$, and $W4$ are structural damping-related parameters described in the “Parameter Descriptions” section of this guide.

    By default, the full equivalent modal viscous damping matrix is written to the MNF. To write only the diagonal values of the equivalent modal viscous damping matrix to the MNF, specify NONCUP=–2 or specify PARAM,NONCUP,-2.

    If both the NONCUP describer and the NONCUP parameter are specified, the NONCUP describer specification takes precedence.

21. Preload conditions are not supported.

22. To request differential stiffness, include a static subcase that contains the stress-stiffening loads. In another subcase include STATSUB = n where n is the number of the static subcase.

SIMPACK STANDARD DESCRIBERS:

Describer	Meaning
FLEXBODY	Requests the generation and writing of standard matrices (CMS) to an FBI file.
NO	NX Nastran solution without standard matrix generation. (default)
YES	Standard matrix generation requested.
FLEXONLY	Determines if DMAP solution runs or not after standard matrix generation is complete.
YES	Only standard matrix generation occurs. (default)
NO	Standard matrix generation occurs along with the DMAP solution.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) section (including PLOTEL) whose connectivity is exported to the FBI file. See Remark 13.
NONE	All grids, geometry and associated modal data is written to the FBI file. (default)
setid	The connectivity of a specific element set is used to export geometry and associated model data.
ALL	The connectivity of all element sets are used to export geometry and associated model data.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of results. See Remark 16.
NO	OP2 file will not be generated. (default)

Describer	Meaning
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. See Remark 17.
NO	No debug output will be written. (default)
YES	Debug output will be written.

SIMPACK STANDARD REMARKS:

1. The creation of a SIMPACK Flexible Body Input (FBI) file is applicable in a non-restart SOL 103, 111, and 112 analysis only. FBI files are named ‘jid_seid.fbi’, where seid is the integer number of the superelement (0 for residual). The location of these files is the same directory as the jid.f06 file.

2. The creation of the FBI file is initiated by MBDEXPORT SIMPACK FLEXBODY=YES (other describers are optional) and the inclusion of the bulk entry DTI,UNITS. This is only valid for a Component Mode Synthesis (CMS) analysis. Thus, it is necessary to define the modal coordinates using the SPOINT bulk entry, and to define them to be in the q-set using the QSET/QSET1 or SEQSET/SEQSET1 bulk entries as appropriate.

3. The Data Table Input bulk entry DTI,UNITS, which is required for an MBDEXPORT SIMPACK FLEXBODY=YES run, specifies the system of units in the original NX Nastran input file. When NX Nastran creates the FBI file, it converts the nastran data from the units defined on the DTI,UNITS entry to SI units. Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

All entries are required. Acceptable character strings are listed below.

Mass:

KG - kilogram

LBM – pound-mass (0.45359237 kg)

SLUG – slug (14.5939029372 kg)

GRAM – gram (1E-3 kg)

OZM – ounce-mass (0.02834952 kg)

KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

MGG – megagram (1E3 kg)

MG – milligram (1E-6 kg)

MCG – microgram (1E-9 kg)

NG – nanogram (1E-12 kg)

UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

CN – centinewton (1E-2 N)

P – poundal (0.138254954 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

4. Because DTI,UNITS determines all units for the FBI file, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the FBI file. For example, if the model mass is in kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1, ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the FBI file would be: “DTI,UNITS,1,KG,N,M,S”.

5. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points; or for a residual-only type model, you can use NX Nastran ASET bulk entries to define the attachment points. Note that the values corresponding to these attachment points in the CMS-reduced mass and stiffness matrices written to the FBI file will be defined in the nodal displacement coordinate systems of these attachment points. The user must account for these coordinate systems when loading or restraining these attachment points within the SIMPACK run.

6. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTs (ns) should be at least ns=n+(6+p). In the above equation for ns, the number of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL bulk entries; the number of load cases is p. In general, you cannot have too many SPOINTs. Excess SPOINTs will be truncated with no performance penalty.

7. OMIT and OMIT1 bulk entries are not supported.

8. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass formulation.

9. CBEND elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

10. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an FBI file. It must have consistent units with regard to the DTI,UNITS bulk entry. Before generating the FBI file, NX Nastran will appropriately scale the WTMASS from the physical mass matrix and mode shapes.

11. There is a distinction between how an MBDEXPORT SIMPACK FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

12. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran, the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with MBDEXPORT SIMPACK FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This caution only applies to a superelement run and not to a residual-only or a part superelement run.

13. To reduce the FE mesh detail for dynamic simulations, PSETID can include the ID of a SET entry. PSETID is also used to define the grids to be included in the recovery matrix that is written to the FBI file. The SET entry lists PLOTEL or element IDs, whose connectivity is exported into the FBI file to display the components in SIMPACK. This option can significantly reduce the size of the FBI file without compromising accuracy in the SIMPACK simulation. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids are attached only to a-set or exterior grids), a SEELT bulk entry must be specified to keep that display element

with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements that are used to select grids to display the component in SIMPACK. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID then represents a list of OUTPUT(PLOT) defined elements' sets, the union of which will be used to define a set of PLOTELs or other elements used to select grids to display the component in SIMPACK. If you wish to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

The following element types are not supported for writing to an FBI file, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CMASS3, CMASS4, CPYRAM, CRAC2D, CRAC3D, CTWIST, CWEDGE, CWELD, and GENEL.

14. Typical NX Nastran data entry requirements are described below.

    Typical Case Control:

    - MBDEXPORT SIMPACK FLEXBODY=YES is required for FBI file generation.
    - METHOD=n is required before or in the first subcase for modal solutions.
    - SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.

    Typical Bulk Data:

    - DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for FBI file generation. For input files containing superelements, this command must reside in the main bulk data section.
    - SPOINT,id_list defines and displays modal amplitude.
    - SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.
    - SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.
    - RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.

- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).
- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main bulk data section.
- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).
- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).
- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in SIMPACK.
- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

15. MBDEXPORT and ADAMSMNF case control entries cannot be used in the same analysis run. In other words, a SIMPACK FBI file or an Adams MNF file can be generated during a particular NX Nastran execution, but not both files at the same time. Attempting to generate both files in the same analysis will cause an error to be issued and the execution to be terminated.

16. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from a SIMPACK analysis. This option requires the following assignment command:

    ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

    be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be used as input into an NX Nastran SOL 103 run using the MBDRECVR case control capability to perform results recovery from a SIMPACK analysis. FLEXBODY=YES is required with its use.

    The data blocks output are:

    MGGEW - physical mass external sort with weight mass removed
    MAAEW - modal mass
    KAAE - modal stiffness
    CMODEXT - component modes.

    This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

    If differential stiffness is included, the static portion of the results will not be included in the recovered results when using MBDRECVR.

17. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

18. The MBDEXPORT data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using MBDEXPORT. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

19. Preload conditions are not supported.

20. To request differential stiffness, include a static subcase that contains the stress-stiffening loads. In another subcase include STATSUB = n where n is the number of the static subcase.

OP4 DESCRIBERS:

Describer	Meaning
unit	The OP4 file is written to the specified logical unit number. (Integer ≠ 0) If *unit* > 0, matrices are written to the OP4 file in sparse format. If *unit* < 0, matrices are written to the OP4 file in full matrix format. The absolute value of the logical unit number must match the unit number on an ASSIGN statement.
FLEXBODY	Requests the generation and writing of standard or state-space matrices to an OP4 file.
NO	NX Nastran solution without standard or state-space matrix generation. (default)
YES	Standard or state-space matrix generation requested.
FLEXONLY	Determines if DMAP solution runs or not after standard or state-space matrix generation is complete.
YES	Only standard or state-space matrix generation occurs. (default)
NO	Standard or state-space matrix generation occurs along with the standard DMAP solution.

Describer	Meaning
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post-processing of controls results. See Remark 7.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. See Remark 8.
NO	No debug output will be written. (default)
YES	Debug output will be written.
NONCUP	Modal damping output control. See Remark 10.
–1	Output the full equivalent modal viscous damping matrix (default).
–2	Output only diagonal values of the equivalent modal viscous damping matrix.

OP4 REMARKS:

1. The generation of standard or state-space matrices and the writing of them to an OP4 file via OUTPUT4, which is applicable in a non-restart SOL 103, 111, or 112 analysis only, is initiated by MBDEXPORT OP4=unit STANDARD FLEXBODY=YES, or MBDEXPORT OP4=unit STATESPACE FLEXBODY=YES (other describers are optional) and the inclusion of the ASSIGN file management statement. This ASSIGN statement must be of the form:

   ASSIGN OUTPUT4='filename',UNIT=n,etc.

   where 'n' matches the absolute value for unit on the MBDEXPORT OP4=unit case control command.

   The number of digits of precision for matrix data is controlled by the DIGITS parameter.

   For a model with superelements, only one OP4 file will be generated. This OP4 file will be generated for the first superelement (or the residual) that satisfies the

conditions defined in Remarks 3 and 4. For standard matrices, if user-defined set U8 is not defined, the residual will be written to the OP4.

2. The parameters LFREQ/HFREQ or LMODES can be used to control which modes are used to derive the standard or state-space matrices.

3. For state-space matrices, user-defined set U7 is used for input DOF. User-defined set U8 is used for output DOF. Refer to the USET/USET1 bulk entries for partitioned superelements and refer to the SEUSET/SEUSET1 bulk entries for non-partitioned superelements.

4. For standard matrices, user-defined set U8 is used for output DOF. The mode shape output will be reduced to the DOF defined in DOF set U8. If DOF set U8 is not defined, the mode shape data for all DOF will be written. Refer to the USET/USET1 bulk entries for partitioned superelements and refer to the SEUSET/SEUSET1 bulk entries for non-partitioned superelements.

5. For the state-space option, the OP4 file contains the [A], [B], [C], and [E] state-space matrices. They are defined as AMAT, BMAT, CMAT, and EMAT, respectively. The input and output DOF are defined as U7DOF and U8DOF, respectively with the first column being the grid ID and the second column being the direction code (1 through 6).

6. For the standard option, the OP4 file contains the modal mass, equivalent modal viscous damping, modal stiffness, mode shapes, and modal forces defined as MMASS, MDAMP, MSTIF, U8PHIX, and MFORC, respectively. The physical DOF corresponding one-to-one with the rows of U8PHIX are defined as U8DOF. The first column contains the grid ID and the second column contains the direction code (1 through 6).

7. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from a system analysis. This option requires the following assignment command:

   ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

   be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which can then be used as an input into an NX Nastran SOL 103 run using the MBDRECVR case control command. FLEXBODY=YES is required when specifying RECVROP2=YES.

   The data blocks output are:

   MGGEW – physical mass external sort with weight mass removed

   MAAEW – modal mass

   KAAE – modal stiffness

   CMODEXT – component modes

This capability is limited to one superelement per NX Nastran model. Residual-only analyses are supported.

If differential stiffness is included, the static portion of the results will not be included in the recovered results when using MBDRECVR.

8. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

9. Differential stiffness is only supported for standard second-order system representation. To request differential stiffness, include a static subcase that contains the stress-stiffening loads. In another subcase include STATSUB = n where n is the number of the static subcase.

10. By default, the full equivalent modal viscous damping matrix is written to standard or state-space OP4 files. To write only the diagonal values of the equivalent modal viscous damping matrix to OP4 files, specify NONCUP=–2, or specify PARAM,NONCUP,-2.

    If both the NONCUP describer and the NONCUP parameter are specified, the NONCUP describer specification takes precedence.

MATLAB DESCRIBERS:

Describer	Meaning
FLEXBODY	Requests the generation and writing of standard or state-space matrices to a MATLAB script file.
NO	NX Nastran solution without standard or state-space matrix generation. (default)
YES	Standard or state-space matrix generation requested.
FLEXONLY	Determines if DMAP solution runs or not after standard or state-space matrix generation is complete.
YES	Only standard or state-space matrix generation occurs. (default)
NO	Standard or state-space matrix generation occurs along with the standard DMAP solution.

Describer	Meaning
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post-processing of controls results. See Remark 7.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. See Remark 8.
NO	No debug output will be written. (default)
YES	Debug output will be written.
NONCUP	Modal damping output control. See Remark 10.
–1	Output the full equivalent modal viscous damping matrix (default).
–2	Output only diagonal values of the equivalent modal viscous damping matrix.

MATLAB REMARKS:

1. The generation of standard or state-space matrices and the writing of them to a MATLAB script file, which is applicable in a non-restart SOL 103, 111, or 112 analysis only, is initiated by MBDEXPORT MATLAB STANDARD FLEXBODY=YES, or MBDEXPORT MATLAB STATESPACE FLEXBODY=YES (other describers are optional). The MATLAB script files are named jid_seid.m where seid is the integer number of the superelement (0 for residual). The location of the MATLAB script files is the same directory as the jid.f06 file.

2. The parameters LFREQ/HFREQ or LMODES can be used to control which modes are used to derive the standard or state-space matrices.

3. For state-space matrices, user-defined set U7 is used for input DOF. User-defined set U8 is used for output DOF. Refer to the USET/USET1 bulk entries for partitioned superelements and refer to the SEUSET/SEUSET1 bulk entries for non-partitioned superelements.

4. For standard matrices, user-defined set U8 is used for output DOF. The mode shape output will be reduced to the DOF defined in DOF set U8. If DOF set U8 is not defined, the mode shape data for all DOF will be written. Refer to the USET/USET1 bulk entries for partitioned superelements and refer to the SEUSET/SEUSET1 bulk entries for non-partitioned superelements.

5. For the state-space option, the MATLAB script file contains the [A], [B], [C], and [E] state-space matrices. They are defined as AMAT, BMAT, CMAT, and EMAT, respectively. The input and output DOF are defined as U7DOF and U8DOF, respectively with the first column being the grid ID and the second column being the direction code (1 through 6).

6. For the standard option, the MATLAB script file contains the modal mass, equivalent modal viscous damping, modal stiffness, mode shapes, and modal forces defined as MMASS, MDAMP, MSTIF, MSHAP, and MFORC, respectively. The physical DOF corresponding one-to-one with the rows of MSHAP are defined as U8DOF. The first column contains the grid ID and the second column contains the direction code (1 through 6).

7. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from a system analysis. This option requires the following assignment command:

   ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

   be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which can then be used as an input into an NX Nastran SOL 103 run using the MBDRECVR case control command. FLEXBODY=YES is required when specifying RECVROP2=YES.

   The data blocks output are:

   MGGEW – physical mass external sort with weight mass removed

   MAAEW – modal mass

   KAAE – modal stiffness

   CMODEXT – component modes

   This capability is limited to one superelement per NX Nastran model. Residual-only analyses are supported.

   If differential stiffness is included, the static portion of the results will not be included in the recovered results when using MBDRECVR.

8. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

9. To request differential stiffness, include a static subcase that contains the stress-stiffening loads. In another subcase include STATSUB = n where n is the number of the static subcase.

10. By default, the full equivalent modal viscous damping matrix is written to standard or state-space MATLAB script files. To write only the diagonal values of the equivalent modal viscous damping matrix to MATLAB script files, specify NONCUP=–2 or specify PARAM,NONCUP,-2.

    If both the NONCUP describer and the NONCUP parameter are specified, the NONCUP describer specification takes precedence.

MBDRECVR

Multi-Body Dynamics Results Recovery

Imports required files from third-party multi-body dynamics codes to perform results recovery.

FORMAT:

$$\text{MBDRECVR}\left[\left\{\begin{matrix}\text{BINARY}\\ \text{ASCII}\end{matrix}\right\}\right],\left[\text{MSRMODE} = \left\{\begin{matrix}0\\1\\2\end{matrix}\right\}\right],\left[\text{RGBODY} = \left\{\begin{matrix}\text{NO}\\ \text{YES}\\ \text{BOTH}\end{matrix}\right\}\right],$$

$$\left[\text{MSGLVL} = \left\{\begin{matrix}0\\1-4\end{matrix}\right\}\right],\left[\text{CHECK} = \left\{\begin{matrix}\text{NO}\\ \text{YES}\end{matrix}\right\}\right]$$

EXAMPLES:

RecurDyn Example:

```
ASSIGN INPUTT2='rfi_results.mdf' UNIT=13
...
CEND
STRESS(PLOT)=100
MBDRECVR
```

Adams Example:

```
ASSIGN INPUTT2=`adams_results.mdf' UNIT=13
...
CEND
STRESS(PLOT)=100
MBDRECVR
```

DESCRIBERS:

Describer	Meaning
BINARY	OUTPUT2 file format for Adams modal deformations file (default). (See Remarks 1 and 2)
ASCII	PUNCH file format for RecurDyn or Adams modal deformations file. (See Remarks 1 and 2).
MSRMODE	Specifies stress recovery type (see Remarks 6 and 7).
0	Component definitions are stored in an OUTPUT2 file (specifically, an *.out file created by using RECVROP2=YES on the MBDEXPORT ADAMS or RECURDYN case control in a pre-Adams/Flex or RecurDyn, respectively, NX Nastran run. The OUTPUT2 files used in this case do not contain data blocks used for MNF or RFI creation (default).
1	Same as option 0, except that the OUTPUT2 file will contain 10 additional data blocks used for RFI or MNF creation by a RecurDyn or Adams pre-processor (specifically, a *.out file created through use of the mnfx.alt DMAP alter capability).
2	No file reference (specifically, component definitions will be recomputed)
RGBODY	Requests the addition of rigid body motion with modal deformations (see Remark 5).
NO	Do not include rigid body motion (default).
YES	Include rigid body motion.
BOTH	Generate two output data blocks; one containing rigid body motion and the other without.
MSGLVL	Level of diagnostic output from Lanczos eigensolver when component definitions are determined (applies only when MSRMODE=2).
0	No output (default).
1	Warning and fatal messages.

Describer	Meaning
2	Summary output.
3	Detailed output on cost and convergence.
4	Detailed output on orthogonalization.
CHECK	Requests debug output be written to the f06 file (See Remark 9).
NO	No debug output will be written (default).
YES	Debug output will be written.

REMARKS:

1. The NX Nastran results recovery requires displacement, velocity, and acceleration data from a multi-body dynamics software product stored in a modal deformation file (MDF). NX Nastran interprets any missing data to be 0.0. The MDF format is described in “Multi-body Dynamics and Control System Software Interfaces” in the Advanced Dynamic Analysis User’s Guide.

2. When modal deformations to be read are in binary (OUTPUT2) format (specifically, BINARY), the following statement needs to be specified near the top of the NX Nastran input file in the file management section:

   ASSIGN INPUTT2=`<MDFilename>' UNIT=13

   where <MDFilename> is the name of the modal deformations file from Adams or RecurDyn.

3. To input the modal deformations file from Adams or RecurDyn in ASCII (Punch) format (specifically, ASCII), the following statement needs to be included in the bulk data section:

   INCLUDE `<MDFilename>' where `<MDFilename>' is the name of the modal deformations file.

4. Dynamic stress/strain output can either be in .f06, PUNCH, and/or OUTPUT2 according to standard NX Nastran functionality. However, stress recovery in NX Nastran from Adams or RecurDyn results do not support XYPLOT output.

5. If displacements, stresses, and/or strains are to be available for post processing, one or more of the following statements must appear in the case control section of the NX Nastran input file:

DISP(PLOT) = <set id>

STRAIN(FIBER,PLOT) = <set id>

STRESS(PLOT) = <set id>

Even though the recovery process is performed in SOL 103, the derived results are transient response results. The default output format for transient results is SORT2. If no SORT2 results are desired, then SORT1 must be requested for all output requests.

6. Rigid body motions from an Adams or RecurDyn simulation are included in the modal deformation file, but they are not applied unless the RGBODY keyword is set to YES or BOTH. Including rigid body motion affects the display and animation of the flexible component, but it has no effect on dynamic stresses. For RGBODY=YES, the displacements output data blocks OUGV1 or OUGV2 will contain a ‘0’ in word 3 of each header record. For RGBODY=NO, OUGV1 or OUGV2 will contain a ‘1’ in word 3 of each header record. For RGBODY=BOTH, two OUGV1 or OUGV2 data blocks will be generated; one for RGBODY=YES and one for RGBODY=NO.

7. For MSRMODE=0 or 1, stress recovery references the OUTPUT2 file obtained from the initial CMS analysis (specifically, RECVROP2=YES on the MBDEXPORT ADAMS or RECURDYN case control entry or use of the mnfx.alt (Adams only) DMAP alter capability). No other files are required. The geometric data needs to be included in the bulk data of the NX Nastran input file because geometry is missing from the OUTPUT2 file. This mode of stress recovery is faster than the MSRMODE=2 mode. To reference this OUTPUT2 file the following line needs to be included in the file specification section of the NX Nastran input file:

   ASSIGN INPUTT2=`<OUTPUT2_filename>' UNIT=20

8. For MSRMODE=2, no files are referenced for stress recovery. Instead, a full CMS reanalysis is performed to build the reference data for the stress recovery analysis. Obviously, the analysis time is significantly far greater compared to the MSRMODE=0 or 1 method, but this method frees up disk space. There is also risk in using this method. If the reanalysis generates slightly different component eigenvalues or eigenvectors than were generated during the creation of the Adams MNF or RecurDyn RFI in the initial NX Nastran run, then the Adams or RecurDyn results in the MDF (modal deformation file) will be inconsistent and incorrect results will be recovered. Something as simple as a sign change for one eigenvector will cause incorrect results to be recovered. It is, therefore, *highly recommended* that MSRMODE=0 or 1 *always* be used.

9. This capability must be performed in SOL 103 and is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

10. Setting CHECK=YES is *not* recommended for models of realistic size due to the amount of data that will be written to the f06.

MEFFMASS

Modal Effective Mass Output Request

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis. Optionally can limit mode output by effective mass fraction.

FORMAT:

$$\text{MEFFMASS}\left(\left(\left(\begin{bmatrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{bmatrix}\right),[\text{GRID}=g],[\text{THRESH}=f],\right.\right.$$

$$[\text{MINT1}=f1],[\text{MINT2}=f2],[\text{MINT3}=f3],[\text{MAXIT}=n],$$

$$\left.\left.\begin{bmatrix}\text{SUMMARY},\text{PARTFAC},\\ \text{MEFFM},\text{MEFFW},\\ \text{FRACSUM},\text{ALL}\end{bmatrix}\right)\right)=\begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

EXAMPLES:

```
MEFFMASS
MEFFMASS(GRID=12,SUMMARY,PARTFAC)
MEFFMASS(PLOT,ALL,THRESH=0.001)=YES
```

DESCRIBERS:

Describer	Meaning
PRINT	Writes output to the print file. (Default)
PUNCH	Writes output to the punch file.
PLOT	Writes output to the output2 file.

Describer	Meaning
GRID=*g*	Reference grid point for the calculation of the Rigid Body Mass Matrix. (Integer; Default is the origin of the basic coordinate system)
THRESH=*f*	Excludes modes that have effective mass fraction less than *f* from mode output. (Real 0.0 ≤ *f* ≤ 1.0; Default is all modes are output) See Remark 8 and Remark 10.
MINT1=*f*1	Calculation of modes continue until either the total effective mass fraction in the X-direction exceeds *f*1 or the maximum number of iterations is reached. (Real 0.0 ≤ *f*1 ≤ 1.0; No default) See Remark 9 and Remark 10.
MINT2=*f*2	Calculation of modes continue until either the total effective mass fraction in the Y-direction exceeds *f*2 or the maximum number of iterations is reached. (Real 0.0 ≤ *f*2 ≤ 1.0; No default) See Remark 9 and Remark 10.
MINT3=*f*3	Calculation of modes continue until either the total effective mass fraction in the Z-direction exceeds *f*3 or the maximum number of iterations is reached. (Real 0.0 ≤ *f*3 ≤ 1.0; No default) See Remark 9 and Remark 10.
MAXIT=*n*	Maximum number of iterations to achieve MINT1, MINT2, and MINT3 specification. (Integer > 0; Default = 5)
SUMMARY	Requests calculation of Total Effective Mass Fraction, Modal Effective Mass Matrix, and the Rigid Body Mass Matrix. (Default)
PARTFAC	Requests calculation of Modal Participation Factors.
MEFFM	Requests calculation of Modal Effective Mass in units of mass.
MEFFW	Requests calculation of Modal Effective Mass in units of weight.
FRACSUM	Requests calculation of Modal Effective Mass Fraction.
ALL	Requests calculation of Total Effective Mass Fraction, Modal Effective Mass Matrix, Rigid Body Mass Matrix, Modal Participation Factors, Modal Effective Mass in units of mass, Modal Effective Mass in units of weight, and Modal Effective Mass Fraction.

REMARKS:

1. The SUMMARY describer produces three outputs:

   Modal Effective Mass Matrix $[\varepsilon^T][m][\varepsilon]$ where

ε	=	Modal Participation Factors
	=	$[m]^{-1}[\varphi]^T[M_{aa}][D_{ar}]$
m	=	Generalized mass matrix
φ	=	Eigenvectors
M_{aa}	=	Mass matrix reduced to the a-set (g-set for superelements)
D_{ar}	=	Rigid body transformation matrix with respect to the a-set

   A-set Rigid Body Mass Matrix: $[D^T_{ar}][M_{aa}][D_{ar}]$. For a superelement this is computed at the g-set.

   Total Effective Mass Fraction: i.e., diagonal elements of the Modal Effective Mass Matrix divided by the Rigid Body Mass Matrix.

2. The PARTFAC describer outputs the Modal Participation Factors table: ε.

3. The MEFFM describer outputs the Modal Effective Mass table: ε^2, the term-wise square of the Modal Participation Factors table.

4. The MEFFW describer outputs the Modal Effective Weight table; i.e., the Modal Effective mass divided by the user parameter WTMASS.

5. The FRACSUM describer outputs the Modal Effective Mass Fraction table; i.e., the Generalized Mass Matrix (diagonal term) multiplied by the Modal Effective Mass and then divided by the Rigid Body Mass Matrix (diagonal term).

6. For superelements the MEFFMASS command uses the residual structure eigenvalues and eigenvectors, by default. If however, PARAM, FIXEDB, -1 is specified then MEFFMASS command uses the component mode eigenvalues and eigenvectors.

7. Effective mass is computed in the basic coordinate system.

8. Modes which have an effective mass fraction greater than the value of THRESH in at least one translational or rotational direction are output in the .f06 and .op2 files. THRESH does not limit modes for consecutive dynamic response solutions

(SOL 111 or 112). Use the MODSEL case control command to select/deselect modes in these solutions.

9. If any combination of MINT1, MINT2, and MINT3 are specified, the calculation of modes continue until either the total effective mass fraction of all those specified exceeds their respective *f* - value or the maximum number of iterations is reached.

10. The THRESH describer cannot be specified in combination with the MINT1, MINT2, or MINT3 describers.

11. On the EIRG and EIGRL bulk entries, you can use combinations of the F1 (V1), F2 (V2), and ND fields to specify:

    - The frequency range for mode calculation.
    - The number of modes to calculate.
    - The number of modes to calculate above a frequency limit.

    If any of the MINT1, MINT2, and MINT3 describers is specified, the software calculates additional modes when either of the following conditions is true:

    - The upper limit to the frequency range is reached prior to satisfying the describer specification. The upper limit is increased to accommodate additional mode calculation.
    - The number of modes calculated does not satisfy the describer specification.

12. For SOL 110, MEFFMASS is not applicable to models that contain fluid elements.

METHOD

Real Eigenvalue Extraction Method Selection

Selects the real eigenvalue extraction parameters.

FORMAT:

$$\text{METHOD}\begin{bmatrix}\text{STRUCTURE}\\ \text{FLUID}\end{bmatrix} = n$$

EXAMPLES:

```
METHOD=33
METHOD(FLUID)=34
```

DESCRIBERS:

Describer	Meaning
STRUCTURE or FLUID	The referenced EIGR or EIGRL Bulk Data entry is applied to the structural or fluid portion of the model. (Default = STRUCTURE)
n	Set identification number of an EIGR or EIGRL Bulk Data entry for normal modes or modal formulation, or an EIGB or EIGRL entry for buckling. (Integer>0)

REMARKS:

1. An eigenvalue extraction method must be selected when extracting real eigenvalues using DMAP modules READ or REIGL.
2. If the set identification number selected is present on both EIGRL and EIGR and/or EIGB entries, the EIGRL entry will be used. This entry requests the Lanczos eigensolution method.
3. When a fluid material is included in the model:

- METHOD(STRUCTURE)=n or METHOD=n selects the eigenvalue method for the structure, and METHOD(FLUID)=n selects the eigenvalue method for the fluid.

- When METHOD(STRUCTURE)=n and METHOD(FLUID)=n are both defined for a modal response solution, the software will perform a modal reduction for both the structure and the fluid.

- When METHOD(STRUCTURE)=n is defined but METHOD(FLUID)=n is not defined for a modal response solution, the software performs a modal reduction for the structure only. When computing the response, the software uses the modal degrees-of-freedom computed for the structure, and the physical degrees-of-freedom belonging to the fluid.

- When a fluid only model is used in a modal response solution, any of METHOD=n, METHOD(FLUID)=n, or METHOD(STRUCTURE)=n can be specified to select the eigenvalue method for the fluid.

- The METHOD(FLUID) and METHOD(STRUCTURE) may be specified simultaneously in the same subcase for the residual structure only. Do not specify METHOD(FLUID) in a superelement subcase even if the superelement contains fluid elements.

MFLUID

Fluid Boundary Element Selection

Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

FORMAT:

MFLUID=n

EXAMPLES:

```
MFLUID=919
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of one or more MFLUID Bulk Data entries. (Integer>0)

REMARKS:

1. For a further discussion, see the *NX Nastran User's Guide*.
2. MFLUID must exist above all subcases.

MODALE

Energy output selection for SOL 111

Energy output selection for SOL 111. This is not supported for the constraint mode method of enforced motion.

FORMAT:

$$\text{MODALE}\left[\left(\left[\text{TYPE}=\left\{\begin{matrix}\text{ALL}\\ \text{KE}\\ \text{SE}\\ \text{BOTH}\\ \text{TOTAL}\end{matrix}\right\}\right],\left[\begin{matrix}\text{BOTH}\\ \text{CONST}\\ \text{OSCILL}\end{matrix}\right],\left[\begin{matrix}\text{SORT2}\\ \text{SORT1}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{NOPRINT}\end{matrix}\right],\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right],\left[\text{FREQ}=\left\{\begin{matrix}\text{ALL}\\ r\end{matrix}\right\}\right]\right)\right]=\left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
MODALE=ALL
SET 200=1,3,4,5,7
SET 300=0.1 0.3 0.5
MODALE(TYPE=BOTH,FREQ=300)=200
```

DESCRIBERS:

Describer	Meaning
TYPE	Type of energy output: All=Strain energy, kinetic energy, and total energy will be output. (Default) SE=Only strain energy will be output. KE=Only kinetic energy will be output.

Describer	Meaning
	BOTH=Both strain and kinetic energy will be output.
	TOTAL=Only total energy will be output.
BOTH	Both constant and oscillating energy results will be output. (Default)
CONST	Only constant energy results will be output.
OSCILL	Only oscillating energy results will be output.
SORT1	Output will be presented as a tabular listing of energy values for the set of frequencies defined by the FREQ describer at each mode in the specified set of modes.
SORT2	Output will be presented as a tabular listing of energy values for the specified set of modes at each frequency defined by the FREQ describer. (Default)
PRINT	Prints the modal energy results to the .f06 file. (Default)
NOPRINT	Writes the modal energy results to the OP2 file without writing to the f06 or punch file.
PUNCH	Writes the modal energy results to the punch (.pch) file.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
FREQ	Computes energies at ALL frequencies or the set of frequencies defined by SET r. (Default=ALL)
ALL, n, NONE	Computes modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.

REMARKS:

1. The FREQ describer selects from the set of forcing frequencies. If a frequency in the selected set is not identical to a frequency in the set of forcing frequencies, energy values at the closest frequency will be provided. If a frequency in the selected set is greater or less than the range of forcing frequencies, energy values will be provided at the highest or lowest forcing frequency, respectively.

2. Modal energy will be calculated and reported for (1) constant energy and/or (2) oscillating energy (depending on whether BOTH, CONST, or OSCILL is requested) at each frequency for each mode and at each frequency for all modes summed together. The frequencies used are those requested by the FREQ describer; the modes used are those requested by ALL/n (i.e. SET n). Combined with the value of the TYPE describer, one or more of the following equations will be output (the modal energy computation is valid for the arbitrary loading case denoted by superscript ‘s’, and the arbitrary excitation radian frequency denoted by subscript ‘r’):

$$MSE_{j,const} = \frac{1}{4}k_j(\xi_{j,re}^2 + \xi_{j,im}^2)$$
$$MSE_{j,osc} = \sqrt{(MSE_{j,osc,cos}^2 + MSE_{j,osc,sin}^2)}$$
$$\phi_{j,MSE} = atan\left(\frac{MSE_{j,osc,cos}}{MSE_{j,osc,sin}}\right)$$
$$MKE_{j,const} = \frac{1}{4}m_j(\dot{\xi}_{j,re}^2 + \dot{\xi}_{j,im}^2)$$
$$MKE_{j,osc} = \sqrt{(MKE_{j,osc,cos}^2 + MKE_{j,osc,sin}^2)}$$
$$\phi_{j,MKE} = atan\left(\frac{MKE_{j,osc,cos}}{MKE_{j,osc,sin}}\right)$$
$$ME_{j,const} = MSE_{j,const} + MKE_{j,const}$$

$ME_{j,osc}\cos(2\omega t + \phi_{j,ME}) = MSE_{j,osc}\cos(2\omega t + \phi_{j,MSE}) + MKE_{j,osc}\cos(2\omega t + \phi_{j,MKE})$
${}^{s}_{r}MSE_{const} = \sum_{j=1}^{n} {}^{s}_{r}MSE_{j,const}$
${}^{s}_{r}MKE_{const} = \sum_{j=1}^{n} {}^{s}_{r}MKE_{j,const}$
${}^{s}_{r}MSE_{osc} = \sqrt{({}^{s}_{r}MSE^{2}_{osc,cos} + {}^{s}_{r}MSE^{2}_{osc,sin})}$
${}^{s}_{r}MKE_{osc} = \sqrt{({}^{s}_{r}MKE^{2}_{osc,cos} + {}^{s}_{r}MKE^{2}_{osc,sin})}$
${}^{s}_{r}\phi^{MSE} = \operatorname{atan}\frac{{}^{s}_{r}MSE_{osc,sin}}{{}^{s}_{r}MSE_{osc,cos}}$
${}^{s}_{r}\phi^{MKE} = \operatorname{atan}\frac{{}^{s}_{r}MKE_{osc,sin}}{{}^{s}_{r}MKE_{osc,cos}}$
${}^{s}_{r}ME_{const} = {}^{s}_{r}MSE_{const} + {}^{s}_{r}MKE_{const}$
${}^{s}_{r}ME_{osc}\cos(2\omega t + {}^{s}_{r}\phi^{ME}) = {}^{s}_{r}MSE_{osc}\cos(2\omega t + {}^{s}_{r}\phi^{MSE}) + {}^{s}_{r}MKE_{osc}\cos(2\omega t + {}^{s}_{r}\phi^{MKE})$

3. Both PRINT and PUNCH may be requested.

MODCON

Modal Contribution Request

Requests modal contribution results for residual.

FORMAT:

$$\text{MODCON}\left[\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{REALorIMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix},\text{PUNCH}\right],$$

$$\begin{bmatrix}\text{ABS}\\ \text{NORM}\\ \text{BOTH}\end{bmatrix},\text{TOPS}=\text{ps},\text{TOPF}=\text{pf},\text{SOLUTION}=\begin{Bmatrix}\text{ALL}\\ \text{setout}\end{Bmatrix},$$

$$\text{PANELMC}=\begin{Bmatrix}\text{NONE}\\ \text{setp}\\ \text{ALL}\end{Bmatrix}\Bigg]=\begin{Bmatrix}\text{n}\\ \text{ALL}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
MODCON=123
MODCON(SORT1,PHASE,PRINT,PUNCH,BOTH,TOPS=5)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of modal dof for each frequency or time. (Default)
SORT2	Output will be presented as a tabular listing of frequency or time for each modal dof. This option is not available for SOL 110.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PRINT	The print file (.f06) will be the output medium. (Default)
PUNCH	The standard punch file (.pch) will be the output medium.
NOPRINT	Generates, but does not print, modal contribution results.
ABS	Output modal contributions in absolute terms. (Default)
NORM	Output modal contributions in normalized terms.
BOTH	Output modal contributions in both absolute and normalized terms.
TOPS (or TOP)	The number of structural modes to list in the output that have the greatest contribution to the response at each frequency or time. The output is sorted in descending order from the structural mode having the greatest contribution when *ps* > 0. If *ps* = 0, no structural mode contributions will be output, only totals. (Default is *ps* = 5)
TOPF	The number of fluid modes to list in the output that have the greatest contribution to the response at each frequency or time. The output is sorted in descending order from the fluid mode having the greatest contribution when *pf* > 0. If *pf* = 0, no fluid mode contributions will be output, only totals. (Default is *pf* = 5)
SOLUTION	SOLUTION = ALL (default) requests that modal contribution calculations be performed at all frequencies or times defined by either the FREQUENCY or TSTEP case control commands, respectively. For SOLUTION = *setout*, modal contribution calculations are performed at the frequencies or times specified by a SET case control command having the identification number of *setout*.

Describer	Meaning
PANELMC	Request modal contributions by panels; only applies to acoustic responses and the contributions from structural modes. PANELMC = ALL requests modal contributions from all panels defined by PANEL bulk entries. PANELMC = *setp* requests modal contributions from all panels listed in the SET case control command having the identification number of *setp*. PANELMC = NONE (default) requests that no modal contributions by panels be calculated.
n	Calculate modal contributions for the list defined by the SETMC case control command having set identification number *n*.
ALL	Calculate modal contributions for the lists defined by all SETMC case control commands specified in and above the current subcase.
NONE	Do not calculate modal contributions. This is useful to turn off modal contribution output for a specific subcase.

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. MODCON = NONE overrides an overall output request.
3. SOL 110, 111, 112, and 146 are supported. For SOL 110, modal contributions for superelements are not supported. The TOPF and PANELMC keywords are only supported for SOL 111. The SOLUTION keyword is only supported for SOL 111, 112, and 146.
4. MODCON supports results for MICPNT microphone points (0D microphone elements).
5. Results for SPC forces do not include the effect of any enforced motion applied at the DOF.
6. The parameters LFREQ, LFREQFL, HFREQ, HFREQFL, LMODES, and LMODESFL are supported.
7. The SOLUTION and PANELMC keywords can be abbreviated to SOLU and PANE, respectively.

8. The SET case control command referenced by SOLUTION = *setout* must contain real values for frequencies or times. Using integer values may lead to erroneous results.

MODES

Subcase Repeater

Repeats a subcase.

FORMAT:

MODES=n

EXAMPLES:

```
MODES=3
```

DESCRIBERS:

Describer	Meaning
n	Number of times the subcase is to be repeated. (Integer>0)

REMARKS:

1. This command can be illustrated by an example. Suppose stress output is desired for the first five modes only and displacements for the next two modes and forces for the remaining modes. The following example would accomplish this.

```
SUBCASE 1 $ FOR MODES 1 THRU 5
   MODES=5
   STRESS=ALL
SUBCASE 6 $ FOR MODES 6 AND 7
   DISPLACEMENTS=
   MODES=2
SUBCASE 8 $ FOR MODE 8 AND REMAINING MODES
   FORCE=ALL
```

2. This command causes the results for each mode to be considered as a separate, successively numbered subcase, beginning with the subcase number containing

the MODES command. In the example above, this means that subcases 1, 2, 3, etc. are assigned to modes 1, 2, 3, etc., respectively.

3. If this command is not used, eigenvalue results are considered to be a part of a single subcase. Therefore, any output requests for the single subcase will apply for all eigenvalues.

4. All eigenvectors with mode numbers greater than the number of subcases defined in Case Control are printed with the descriptors of the last subcase. For example, to suppress all printout for modes beyond the first three, the following Case Control could be used:

```
SUBCASE 1
   MODES=3
   DISPLACEMENTS=ALL
SUBCASE 4
   DISPLACEMENTS=NONE
BEGIN BULK
```

5. This command may be of no use in non eigenvalue analysis and may cause data recovery processing to be repeated.

MODSEL

Mode Selection

Used to select mode numbers to include in a modal dynamic response solution.

FORMAT:

$$\text{MODSEL}\begin{pmatrix}\text{STRUCTURAL}\\ \text{FLUID}\end{pmatrix}=\begin{Bmatrix}\text{ALL}\\ \text{n}\end{Bmatrix}$$

EXAMPLES:

```
MODSEL = 3
MODSEL(FLUID) = 4
MODSEL(STRUCTURAL) = ALL
MODSEL = -10
```

DESCRIBERS:

Describer	Meaning
STRUCTURAL	Specifies the structural modes to include in the response solution. (Default)
FLUID	Specifies the fluid modes to include in the response solution.
ALL	Designates that all the structural or fluid modes be used in the response solution. (Default)
n	Identification number of the SET case control command containing a list of either structural or fluid modes to be used in the response solution. The mode numbers not included in the SET case control command are removed from the modal space. By preceding the identification number of the SET case control command with a negative sign, the mode numbers listed in the SET case control command are omitted from the response solution. Mode numbers listed in the SET case control command that are larger than the number of computed modes are ignored. (Integer≠0)

REMARKS:

1. All structural and fluid modes are used in the response solution if a MODSEL case control command is not included in the input file.

2. The use of MODSEL is supported at the subcase level and for restarts.

3. Multiple MODSEL case control commands can be included in any subcase.

4. MODSEL is supported for restarts and at the subcase level for SOLs 111, 112, 145, and 146. MODSEL is supported for restarts, but not at the subcase level for SOLs 103, 110, and 187. For SOLs 103, 110, and 187, MODSEL must be above the subcase level.

MODTRAK

Mode Tracking Request

Selects mode tracking options in design optimization (SOL 200).

FORMAT:

MODTRAK=n

EXAMPLES:

```
MODTRAK=100
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a MODTRAK Bulk Data entry. (Integer>0)

REMARKS:

Selection of a MODTRAK Bulk Data entry with the MODTRAK Case Control command activates mode tracking for the current subcase. This request is limited to normal modes subcases (ANALYSIS = MODES) in design optimization (SOL 200).

MONVAR

Selects degree-of-freedom for a displacement monitor plot in a SOL 401 run with Simcenter Multiphysics.

Selects degree-of-freedom for a displacement monitor plot in a SOL 401 run with Simcenter Multiphysics.

FORMAT:

$$\text{MONVAR} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
MONVAR=ALL
SETMC 21 = DISP/ 1001(T1), 1002(T1), 1204(T1)
SETMCNAME 21 = Blade 3
SETMC 25 = DISP/ 1001(T2), 1002(T2), 1204(T2)
SETMCNAME 25 = Compressor Disk
```

DESCRIBERS:

Describer	Meaning
ALL	Requests displacement monitoring of all defined SETMC IDs. See the Examples.
n	Set identification number of a previously appearing SETMC command. The SETMC command lists a grid point ID and component for each degree-of-freedom to be included in the displacement monitoring. See the Examples. (Integer>0)
NONE	No displacement monitoring.

REMARKS:

1. Only supported in a SOL 401 statics subcase.

2. A displacement monitor plot is created for each SETMC ID selected. Each displacement monitor plot includes a plot of the maximum, the minimum, and the average displacement. The software uses the displacement values for all degree-of-freedom listed on a particular SETMC to determine the maximum and minimum displacement, and to compute the average displacement for each converged time step. If you want to monitor the displacement of a single degree-of-freedom, you should select a SETMC which includes only this degree-of-freedom.

3. You can optionally define a title for each displacement monitor plot using the SETMCNAME command. The ID on the SETMCNAME command should match the ID on the SETMC command. See the Examples. When the SETMCNAME is undefined, the ID of the SETMC is used for the plot title.

MPC

Multipoint Constraint Set Selection

Selects a multipoint constraint set.

FORMAT:

MPC=n

EXAMPLES:

```
MPC=17
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a multipoint constraint set. This set identification number must appear on at least one MPC or MPCADD Bulk Data entry. (Integer>0)

REMARKS:

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary (MPC sets) conditions are not allowed in superelement analysis. If more than one MPC set is specified per superelement (including the residual), then the second and subsequent sets will be ignored.

MPCFORCES

Multipoint Forces of Constraint Output Request

Requests the form and type of multipoint force of constraint vector output.

FORMAT:

$$\text{MPCFORCES}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
MPCFORCES=5
MPCFORCES(SORT2,PUNCH,PRINT,IMAG)=ALL
MPCFORCES(PHASE)=NONE
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.

Describer	Meaning
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, multipoint constraint forces.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 9.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 9.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 9.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 9.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 9.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 9.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 9.
RPUNCH	Writes random analysis results to the punch file. See Remark 9.

Describer	Meaning
ALL	Multipoint forces of constraint for all points will be output. See Remark 3 and Remark 6.
NONE	Multipoint forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only multipoint forces constraint for points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.

4. MPCFORCES=NONE overrides an overall output request.

5. In SORT1 format, MPCFORCES recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of

output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.

6. MPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).

7. MPCFORCES results are not available in SOL 129.

8. In inertia relief analysis, the MPCFORCES output includes both the effects of applied and inertial loads.

9. The following applies to random solutions:

   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

MPRES

Fluid Pressure Output Request

Requests the pressure for selected surface elements in fluid-structure interaction problems.

FORMAT:

$$\text{MPRES}\left[\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right], \left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\right] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
MPRES=5
MPRES(IMAG)=ALL
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print or punch, data.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Fluid pressures for all elements will be output.
NONE	Fluid pressures for no elements will be output.

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only fluid pressures for elements in this set will be output. (Integer>0)

REMARKS:

PARAM,DDRMM,-1 is also required in the modal solution sequences 111, 112, 146, and 200.

NLCNTL

Analysis parameter selection for SOL 401.

Selects the parameters used in SOL 401.

FORMAT:

NLCNTL=n

EXAMPLES:

```
NLCNTL=10
```

DESCRIBERS:

Describer	Meaning
n	Set identification of NLCNTL bulk entries. (Integer>0)

REMARKS:

1. Only supported in SOL 401.
2. NLCNTL may be defined above the subcases (globally) or within a subcase.
3. NLCNTL entries in the Bulk Data will not be used unless selected with a NLCNTL case control command.

NLLOAD

Nonlinear Load Output Request

Requests the form and type of nonlinear load output for transient problems.

FORMAT:

$$\text{NLLOAD}[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
NLLOAD=ALL
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Nonlinear loads for all solution points will be output.
NONE	Nonlinear loads will not be output.
n	Set identification of a previously appearing SET command. Only nonlinear loads for points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Nonlinear loads are output only in the solution (d or h) set.

2. The output is available in SORT2 format only.
3. Both PRINT and PUNCH may be used.
4. NLLOAD=NONE allows overriding an overall output request.

NLPARM

Nonlinear Static Analysis Parameter Selection

Selects the parameters used for nonlinear static analysis.

FORMAT:

NLPARM=n

EXAMPLES:

```
NLPARM=10
```

DESCRIBERS:

Describer	Meaning
n	Set identification of NLPARM and NLPCI Bulk Data entries. (Integer>0)

REMARKS:

1. NLPARM and NLPCI entries in the Bulk Data will not be used unless selected.
2. NLPARM may appear above or within a subcase.

NLSTRESS

Nonlinear Element Stress Output Request

Requests the form and type of nonlinear element stress output in SOL 106.

FORMAT:

$$\text{NLSTRESS}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
NLSTRESS=5
NLSTRESS (SORT1,PRINT,PUNCH,PHASE)=15
NLSTRESS(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load.
SORT2	Output will be presented as a tabular listing of load for each element type.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates nonlinear element stresses for requested set but no printer output.
ALL	Stresses for all nonlinear elements will be output. (Default)

Describer	Meaning
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	No nonlinear element stress will be output.

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. ALL should be used with caution in transient solutions as it can produce a large amount of output.

3. See “DISPLACEMENT” for a discussion of SORT1 and SORT2.

4. SOLs 106 and 129 can write the stress and strain output for nonlinear elements into both a linear and a nonlinear format. The STRESS case control command requests both the linear and nonlinear stress format. The NLSTRESS case control command requests only the nonlinear format. The formats have a different data organization. For example, the nonlinear format includes stresses together with strains, and it provides more information with regard to nonlinear material laws (effective strain, equivalent stress). See the *Handbook of Nonlinear Analysis* for details on the computation of effective strain and equivalent stress.

5. For the output coordinate system of CQUAD4 and CTRIA3 elements, see “CQUAD4” and “CTRIA3”. For the output coordinate system of CHEXA, CPENTA, CPYRAM, and CTETRA elements, see “PSOLID” and “PLSOLID”.

NONLINEAR

Nonlinear Dynamic Load Set Selection

Selects a nonlinear dynamic load set for transient problems.

FORMAT:

NONLINEAR=n

EXAMPLES:

```
NONLINEAR=75
```

DESCRIBERS:

Describer	Meaning
n	Set identification of NOLINi or NLRGAP Bulk Data entry. (Integer>0)

REMARKS:

NOLINi Bulk Data entry will be ignored unless selected in the Case Control Section.

NOUTPUT

Normal Output Request in Cyclic Symmetry Problems

Requests physical output in cyclic symmetry problems.

FORMAT:

$$\text{NOUTPUT}\left\{ k, \begin{matrix} R \\ L \end{matrix} \right\} = \left\{ \begin{matrix} \text{ALL} \\ m \end{matrix} \right\}$$

EXAMPLES:

```
NOUTPUT (R)=ALL
NOUTPUT (2)=5
NOUTPUT (4,L)=10
```

DESCRIBERS:

Describer	Meaning
ALL	Output for all segments is desired.
m	Output for segments specified in SET m is desired. (Integer>0)
k	Used in eigenvalue analysis to request eigenvector and internal force output for harmonics specified in SET k. (Integer>0)
R, L	Output for only the right- or left-half of segments specified as ALL or in SET m. R and L are used in dihedral symmetry only.

REMARKS:

1. Sets k and m are defined on SET commands.
2. In cyclic symmetry analysis, this command or the HOUTPUT command is required to obtain data recovery.

NSM

Nonstructural Mass

Selects Nonstructural Mass (NSM) set for mass generation.

FORMAT:

NSM=n

EXAMPLES:

```
NSM=5
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a nonstructural mass that appears on a NSM, NSM1, or NSMADD bulk data entry. (Integer > 0)

REMARKS:

1. In a non-superelement analysis, the NSM request must be consistent in all subcases. This is achieved with a single NSM request above the subcase level, or by including the same NSM request in each subcase. A fatal error will occur if the NSM request changes in any subcase. In a superelement analysis, each superelement subcase can have a different NSM request, but the NSM request must be consistent in all residual subcases.

OFREQUENCY

Output Frequency Set

Selects a set of frequencies for output requests.

FORMAT:

$$\text{OFREQUENCY} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

EXAMPLES:

```
OFREQUENCY=ALL
OFREQUENCY=15
```

DESCRIBERS:

Describer	Meaning
ALL	Output for all frequencies will be computed.
n	Set identification of a previously appearing SET command. Output for frequencies closest to those given on this SET command will be output. (Integer>0)

REMARKS:

1. In real eigenvalue, buckling, and complex eigenvalue analyses, the OMODES Case Control command allows for an alternate way of selecting the modes to be output based on their mode numbers. In these cases, if both the OMODES and OFREQUENCY requests appear, the OMODES request takes precedence.

2. If this command is not specified in the Case Control Section (or, in the case of real eigenvalue, buckling, and complex eigenvalue analyses, if neither the OMODES nor the OFREQUENCY request is specified), then output will be generated for all frequencies.

3. The number of solutions selected will always be equal to the number of quantities in the selected set. The closest values are used. This is accomplished by working in increasing order of magnitude in the OFREQ set, assigning the closest frequency value from the frequency set for each value in the OFREQ set, and if already previously assigned, then assigning the next closest value. Thus, for example, if the frequency set is 2.5; 3.5; …; 14.5; 15.5, and the OFREQ set is 3.8; 17.4; 21.7, the program will select 3.5; 15.5; 14.5 from the frequency set in that order, and will sort to end up with 3.5; 14.5; 15.5.

4. In flutter analysis (SOL 145), the selected set refers to the imaginary part of the complex eigenvalues. The physical interpretation of this quantity depends on the method of flutter analysis as follows:

   - K- or KE-method: Velocity (input units).

   - PK-method: Frequency.

5. In aeroelastic response analysis (SOL 146) with RLOAD selection, the selected set refers to the frequency (cycles per unit time).

6. In complex eigenvalue analysis (SOLs 107 and 110), the selected set refers to the imaginary part of the complex eigenvalues.

7. In the modal solution sequences, if this command is specified in more than one subcase, then it is recommended that the first subcase contain OFREQ=ALL and subsequent subcases contain OFREQ=n. Instead of OFREQ=ALL, a super-set of all OFREQ sets to follow may be used in the first subcase (see exceptions described in Remark 8). Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
   OFREQ=ALL $ 0.0  through 0.5
SUBCASE 2
   SET 10=0.0 0.1 0.3
   OFREQ=10
   DISP=ALL
SUBCASE3
   SET 20=0.4 0.5
   OFREQ=20
   STRESS=ALL
```

8. An OFREQ selection in a given subcase is valid for all subsequent subcases of a given batch of frequency response subcases, until a new OFREQ selection is encountered. However, with SOL 200, when a new boundary condition and/or a new frequency set is encountered, a new batch of frequency response subcases starts, and a new OFREQ super-set (see Remark 7) needs to be entered with the first subcase of this new batch, if there was a previous definition of OFREQ.

OLOAD

Applied Load Output Request

Requests the form and type of applied load vector output.

FORMAT:

$$\text{OLOAD}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
OLOAD=ALL
OLOAD(SORT1,PHASE)=5
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 11.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 11.
RPUNCH	Writes random analysis results to the punch file. See Remark 11.
ALL	Applied loads for all points will be output. See Remark 2 and Remark 8.
NONE	Applied load for no points will be output.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. In a statics problem, a request for SORT2 causes loads at all requested points (zero and nonzero) to be output.

4. OLOAD=NONE overrides an overall output request.

5. In the statics superelement solution sequences, and in the dynamics SOLs 107 through 112, 118, 145, 146, and 200, OLOADs are available for superelements and the residual structure. Only externally applied loads are printed, and not loads transmitted from upstream superelements. Transmitted loads can be obtained with GPFORCE requests.

   - In the nonlinear transient analysis solution sequences SOLs 129 and 159, OLOADs are available only for residual structure points and include loads transmitted by upstream superelements.

6. In nonlinear analysis, OLOAD output will not reflect changes due to follower forces.

7. Loads generated via the SPCD Bulk Data entry do not appear in OLOAD output.

8. In SORT1 format, OLOADs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and it is zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.

9. OLOAD results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).

10. In inertia relief analysis, the OLOAD output includes both the inertia loads and applied loads.

11. The following applies to random solutions:

    - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

    - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

    - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

OMODES

Output Modes Set

Selects a set of modes for output requests.

FORMAT:

$$\text{OMODES} = \begin{Bmatrix} \text{ALL} \\ \text{n} \end{Bmatrix}$$

EXAMPLES:

```
OMODES=ALL
OMODES=20
```

DESCRIBERS:

Describer	Meaning
ALL	Output for all extracted modes will be computed. (Default)
n	Set identification of a previously appearing SET command. Output for those extracted modes appearing on this SET command will be computed.

REMARKS:

1. OMODES is supported in SOLs 103, 105, 107, 110, 111, 112, 200, and 401. It is ignored in all other solution types.
2. In contrast to the OFREQENCY Case Control request, which affords an alternate way of selecting the modes to be output based on their frequencies, the OMODES command allows mode selection based on integer mode ID. For example:

```
SUBCASE 10
      ...
```

```
        SET 11=1,3,5,7
        OMODES=11
        DISP=ALL
        …
SUBCASE 20
        …
        SET 21=25., 28., 31.
        OFREQ=21
        DISP=ALL
…
```

The “THRU” option on the SET definition can be used with the OMODES command. For example:

```
SET 20 = 5 THRU 30
OMODES=20
```


3. If both the OMODES and the OFREQUENCY requests appear, the OMODES request takes precedence.

4. If neither the OMODES nor the OFREQUENCY request is specified, output will be generated for all modes.

5. It is important to note that the OMODES request has no effect whatsoever on the number of modes computed. Instead, it only provides a means for selecting a subset of the computed modes for which output is to be generated.

6. In superelement analysis, the set definition of an OMODES request for an upstream superelement will not be honored unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OMODES default is ALL. Note that the code does not check to see if this condition is satisfied.

7. If the parameter SECOMB is set to “YES”, then OMODES must be the same for all superelements.

8. In SOL 401, for the cyclic and Fourier subcase types, a unique modal solution occurs for the requested harmonic numbers. As a result of the inherent symmetry with these solutions, mode pairs exist for specific harmonic numbers. For a cyclic subcase, mode pairs exist for harmonic numbers 1 through N/2 -1. For the Fourier subcase, mode pairs exist for all harmonic numbers > 0. The software automatically outputs the mode pairs for these subcase types for each mode number referenced by the OMODES command.

   For example, if you define OMODES=1, where SET 1 = 1,2,5,7 and you request 10 modes on the EIGRL bulk entry:

   For a cyclic subcase:

For harmonic index 0 and N/2, 10 modes are computed. Modes 1, 2, 5, and 7 are output.

For harmonic numbers 1 through N/2 -1, 20 modes are computed (10 distinct modes). Modes 1, 2, 3, 4, 9, 10, 13, and 14 are output (corresponding to the 1st, 2nd, 5th and 7th unique frequency).

For a Fourier subcase:

For harmonic index 0, 10 modes are computed. Modes 1, 2, 5, and 7 are output.

For harmonic index > 0, 20 modes are computed (10 distinct modes). Modes 1, 2, 3, 4, 9, 10, 13, and 14 are output (corresponding to the 1st, 2nd, 5th and 7th unique frequency).

OPRESS

Requests solution set pressure output for SOL 401 in the Simcenter Multiphysics environment.

Requests the form and type of solution set pressure output.

FORMAT:

$$\text{OPRESS}\left[\left(\text{PRINT}, \text{PLOT}\right)\right] = \left\{\begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix}\right\}$$

EXAMPLES:

```
OPRESS=ALL
OPRESS(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PLOT	Compute output.
ALL	Requests output for all elements.
n	Set identification number of a previously appearing SET command. Only elements with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401 in the context of a coupled structural-thermal analysis within the Simcenter Multiphysics environment.

2. In the Simcenter Multiphysics environment, you can optionally request that the pressures computed by the Simcenter Thermal solver be passed to the SOL 401 structural solution as a mechanical loading. These pressures are passed to NX Nastran directly and not written to the NX Nastran input file. The OPRESS command output only includes the applied pressure loads as a result of this data exchange with the Simcenter Thermal solver. This output does not include any other pressure loading. For example, pressures defined with the PLOAD or PLOAD4 entries are not included in this output. The OLOAD case control command can be used to request the output of all applied loads including pressures defined with the PLOAD and PLOAD4 entries.

3. Only SORT1 data is output.

OSTNINI

Requests initial strain output in SOL 401.

Requests initial strain output in SOL 401.

FORMAT:

$$\text{OSTNINI}\left[\left(\begin{bmatrix}\text{GRID}\\ \text{GAUSS}\\ \text{BOTH}\end{bmatrix}, \text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
OSTNINI=ALL
OSTNINI(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
GRID	Output at grid points on elements.
GAUSS	Output at gauss points on elements.
BOTH	Output at both grid and gauss points on elements.
ALL	Requests output for all elements.
n	Set identification number of a previously appearing SET command. The SET command lists element IDs. (Integer>0)
NONE	No output requested.

REMARKS:

1. Only supported in SOL 401.
2. The OSTNINI command should be defined above the subcases (globally). The output only occurs once, at the beginning of the solution before any subcase solutions begin.
3. Strains reported are engineering strain.
4. Initial strains are written to the .op2 file in the OES data block in SORT1 format.
5. The strain components are output in the basic coordinate system.

OTEMP

Requests temperature output on grid points for SOL 401.

Requests temperature output on grid points.

FORMAT:

$$\text{OTEMP}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
OTEMP=ALL
OTEMP(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

OTIME

Output Time Set

Selects a set of times for output requests.

FORMAT:

$$\text{OTIME} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

EXAMPLES:

```
OTIME=ALL
OTIME=15
```

DESCRIBERS:

Describer	Meaning
ALL	Output for all times will be computed.
n	Set identification number of a previously appearing SET command. Output for times closest to those given on this SET command will be computed. (Integer>0)

REMARKS:

1. If the OTIME command is not supplied in the Case Control Section, then output for all times will be computed.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses at only peak times, etc.).
3. This command can be used in conjunction with the MODACC module to limit the times for which modal acceleration computations are performed.

4. If this command is specified in more than one subcase in the modal solution sequences, then it is recommended that the first subcase contain OTIME=ALL and subsequent subcases contain OTIME=n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
    OTIME ALL
SUBCASE 2
    OTIME=10
    SET10=. . .
    DISP=ALL
SUBCASE 3
    OTIME=20
    SET 20=. . .
    STRESS=ALL
```

5. The OTIME command is not effective in nonlinear transient analysis.

6. In superelement analysis, the set definition of an OTIME request for an upstream superelement will not be honored unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OTIME default is ALL. Note that the program does not check to see if this condition is satisfied.

OUTPUT

Case Control Delimiter

Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.

FORMAT:

$$\text{OUTPUT}\left[\left(\left\{\begin{array}{c}\text{PLOT}\\ \text{POST}\\ \text{XYOUT}\\ \text{XYPLOT}\\ \text{CARDS}\end{array}\right\}\right)\right]$$

EXAMPLES:

```
OUTPUT
OUTPUT(PLOT)
OUTPUT(XYOUT)
```

DESCRIBERS:

Describer	Meaning
PLOT	Beginning of the structure plotter request. This command must precede all structure plotter control commands.
POST	Beginning of grid point stress SURFACE and VOLUME commands. This command must precede all SURFACE and VOLUME commands.
XYOUT or XYPLOT	Beginning of curve plotter request. This command must precede all curve plotter control commands. XYPLOT and XYOUT are entirely equivalent. Curve plotter commands are described in “X-Y PLOT Commands” .

Describer	Meaning
CARDS	The OUTPUT(CARDS) packet is used by the MSGSTRESS program. These commands have no format rules. This package must terminate with the command ENDCARDS (starting in column 1).

REMARKS:

1. The structure plotter request OUTPUT(PLOT), the curve plotter request OUTPUT(XYOUT or XYPLOT), and the grid point stress requests (OUTPUT(POST)) must follow the standard Case Control commands.

2. If OUTPUT is specified without a describer, then the subsequent commands are standard Case Control commands.

3. Case Control commands specified after OUTPUT(POST) are SURFACE and VOLUME.

P2G

Direct Input Load Matrix Selection

Selects direct input load matrices.

FORMAT:

P2G=name

EXAMPLES:

```
P2G=LDMIG
P2G=L1, L2, L3
SET 100=LDMIG, L1, L8
P2G=100
P2G=1.25*L1, 1.0*L2, 0.75*L3
```

DESCRIBERS:

Describer	Meaning
name	Name of a $[P^2_g]$ matrix that is input on the DMIG Bulk Data entry. (Character) Scale factors may be included (see Remarks 4 and 5). See “Applied Static Loads in SubDMAPs SELG, SELR, and SELRRS” in the *NX Nastran User’s Guide*.

REMARKS:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the load matrix before any constraints are applied.
3. The matrix must be columnar in form (field 4 on the DMIG entry must contain the integer 9).

4. The associated DMIG matrices can be scaled using either in-line scale factors on P2G (for example, P2G =1.25*L1), using the parameter CP2 (for example, PARAM,CP2,1.25), or both. See “Parameter Descriptions”.

5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. A scale factor of 1.0 should be used for matrices in the list which are not scaled. For example,

```
P2G=1.25*L1, 1.0*L2, 0.75*L3
```

   The parameter CP2 when defined will scale all P2G. For example, if PARAM,CP2,1.30 is defined with the P2G example above, the result would be P2G=1.30(1.25*L1 + 1.0*L2 + 0.75*L3).

6. The P2G command is supported across subcases. A P2G command selecting a different DMIG matrix can be defined for each subcase.

PANCON

Acoustic Panel Contribution Request

Requests acoustic panel contribution results for residual.

FORMAT:

$$\text{PANCON}\left[\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{REALorIMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix}, \text{PUNCH}\right],$$

$$\begin{bmatrix}\text{ABS}\\ \text{NORM}\\ \text{BOTH}\end{bmatrix}, \text{TOPP} = \text{pp}, \text{TOPG} = \text{pg}, \text{SOLUTION} = \begin{Bmatrix}\text{ALL}\\ \text{setf}\end{Bmatrix},$$

$$\text{PANEL} = \begin{Bmatrix}\text{ALL}\\ \text{setp}\\ \text{NONE}\end{Bmatrix}, \text{GRID}=\begin{Bmatrix}\text{NONE}\\ \text{setg}\\ \text{ALL}\end{Bmatrix}\Bigg] = \begin{Bmatrix}\text{n}\\ \text{ALL}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
PANCON=123
PANCON(SORT1,PHASE,PRINT,PUNCH,BOTH,TOPP=5)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of panels or grids for each frequency. (Default)
SORT2	Output will be presented as a tabular listing of frequency for each panel or grid.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PRINT	The print file (.f06) will be the output medium. (Default)
PUNCH	The standard punch file (.pch) will be the output medium.
NOPRINT	Generates, but does not print, contribution results.
ABS	Output contributions in absolute terms. (Default)
NORM	Output contributions in normalized terms.
BOTH	Output contributions in both absolute and normalized terms.
TOPP	The number of structural panels to list in the output that have the greatest contribution to the response at each frequency. The output is sorted in descending order from the structural panel having the greatest contribution when *pp* > 0. If *pp* = 0, no structural panel contributions will be output, only totals. (Default is *pp* = 5)
TOPG	The number of structural grids to list in the output that have the greatest contribution to the response at each frequency. The output is sorted in descending order from the structural grid having the greatest contribution when *pg* > 0. If *pg* = 0, no structural grid contributions will be output, only totals. (Default is *pg* = 20)
SOLUTION	SOLUTION = ALL (default) requests that contribution calculations be performed at all frequencies defined by the FREQUENCY case control commands. For SOLUTION = *setf*, contribution calculations are performed at the frequencies specified by a SET case control command having the identification number of *setf*.
PANEL	Specifies the set of panels to output panel contributions. PANEL = ALL (default) requests that contributions from all panels defined by PANEL bulk entries be output. PANEL = *setp* requests contributions from panels included in the SET case control command having the identification number of *setp*. PANEL = NONE requests that no contributions from panels be output.

Describer	Meaning
GRID	Specifies the set of grids to output contributions. GRID = ALL requests contributions from all structural grids that are part of the acoustic coupling matrix. GRID = *setg* requests contributions from structural grids included in the SET case control command having the identification number of *setg*. Any grids included in *setg* that are not part of the acoustic coupling matrix will be ignored. GRID = NONE (default) requests that no structural grid contributions be output.
n	Calculate panel and/or grid contributions for the list defined by the SETMC case control command having set identification number *n*. Any response listed in SETMC = *n* that is acoustic will generate panel and/or grid contributions as requested. Any response listed in SETMC = *n* that is structural will generate panel reciprocal contributions if panel contributions are requested.
ALL	Calculate panel and/or grid contributions for the lists defined by all SETMC case control commands specified in and above the current subcase. Any response listed in SETMC case control commands that is acoustic will generate panel and/or grid contributions as requested. Any response listed in SETMC case control commands that is structural will generate panel reciprocal contributions if panel contributions are requested.
NONE	Do not calculate panel or grid contributions. This is useful to turn off contribution output for a specific subcase.

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. PANCON = NONE overrides an overall output request.
3. SOL 108 and 111 are supported.
4. PANCON supports results for MICPNT microphone points (0D microphone elements).
5. The parameters LFREQ, LFREQFL, HFREQ, HFREQFL, LMODES, and LMODESFL are supported.

6. The SOLUTION and PANEL keywords can be abbreviated to SOLU and PANE, respectively.

7. The SET case control command referenced by SOLUTION = *setf* must contain real values for frequencies. Using integer values may lead to unintended results.

8. The SET case control command referenced by PANEL = *setp* must contain the alphanumeric name of existing panels defined by PANEL bulk entries.

PARAM

Parameter Specification

Specifies values for parameters. Parameters are described in Parameters and Parameter Applicability Tables.

FORMAT:

PARAM,n,V1,V2

EXAMPLES:

```
PARAM,GRDPNT,0
PARAM,K6ROT,1.0
```

DESCRIBERS:

Describer	Meaning
n	Parameter name (1 to 8 alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single precision	Real	Blank
Character	Character	Blank
Real, double precision	Real, Double Precision	Blank
Complex, single precision	Real or Blank	Real or Blank
Complex, double precision	Real, Double Precision	Real, Double Precision

REMARKS:

1. The PARAM command is normally used in the Bulk Data Section and is described in Bulk Data Entries.

2. The parameter values that may be defined in the Case Control Section are described in Parameters and Parameter Applicability Tables. Case Control PARAM commands in user-written DMAPs require the use of the PVT module, described in the *NX Nastran DMAP Programmer's Guide*.

3. The following should be considered when external superelements or part superelements are included in the Bulk Data section of your input file.

   - Parameters specified in the Bulk Data, but not in a BEGIN SUPER portion of the Bulk Data, apply only to the residual and not to external or part superelements.

   - Parameters specified in a BEGIN SUPER portion of the Bulk Data apply only to that external or part superelement.

   - To ensure that parameters apply to the residual and all superelements, you should specify them in the Case Control section above the subcases. For example, specifying the PARAM,POST setting above the subcases will request a consistent output format for the residual and the superelements.

PARTN

Partitioning Vector Specifications

Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17). In SOLs 111 and 200, the PARTN command specifies the points at which modal participation factors are to be computed.

FORMAT:

PARTN=n

EXAMPLES:

```
PARTN=10
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only modal participation factors for points with identification numbers that appear on the SET=n command will be output. (Integer>0)

REMARKS:

1. The PARTN command and the DMAP module MATMOD provide a convenient method for building a partitioning vector for use in the DMAP modules such as PARTN and MERGE.

2. Modal participation factors are computed automatically in SOLs 111 and 200 when coupled fluid-structure analysis is performed. See the *NX Nastran User's Guide*.

PFRESULTS

Progressive Failure Output Request

Requests progressive failure results output for composite solid elements in SOL 401.

FORMAT:

$$\text{PFRESULTS}\left[\left(\text{DAMAGE, STATUS, ENERGY, }\begin{bmatrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
PFRESULTS(DAMAGE,ENERGY,PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
DAMAGE	Requests damage value results. See Remarks 1 and 2. (Default)
STATUS	Requests damage status. See Remark 3.
ENERGY	Requests damage energy results. See Remark 4.
PRINT	The printer is the output medium. (Default)
PUNCH	The punch file is the output medium.
PLOT	Generates results for requested set but no printer output.
ALL	Results are output for all composite solid elements.
n	Identification number of a SET case control command that lists the composite solid elements for which results are output. (Integer>0)
NONE	No results are output for any composite solid elements.

REMARKS:

1. Results are reported at the corner grid points for each ply in the laminate on an individual element basis. The results for each ply in the laminate are reported at the middle of the ply.

2. Damage values are calculated at the Gauss points and extrapolated to the corner grid points. At the Gauss points, the calculated damage can range from 0.0 and 0.999. A damage value of 0.999 indicates that the elastic modulus is reduced to 0.1% of its original value. The software limits the damage to 0.999 to avoid numerical problems that can arise if the damage is 1.0. However, because the values at the Gauss points are extrapolated to the corner grids, the reported damage values at the corner grids can be less than 0.0 and greater than 0.999.

3. Damage status is reported on an individual element basis as follows:

   “0” indicates that the laminate is undamaged.

   “1” indicates that some plies in the laminate are damaged, but have not failed.

   “2” indicates that some plies in the laminate have failed and other plies in the laminate may be damaged.

   “3: indicates that all plies in the laminate have failed.

   A ply has failed when the software calculates that the ply is damaged at all Gauss points. A ply is damaged when the software calculates that the ply is damaged at one or more, but not all of the Gauss points.

4. Damage energy is the energy that is expended to cause the damage in the laminate. Damage energy is output on an element-by-element basis.

PLOTID

Plotter Identification

Defines a character string that will appear on the first frame of any plotter output.

FORMAT:

PLOTID=title

EXAMPLES:

```
PLOTID=BLDG. 125 BOX 91
```

DESCRIBERS:

Describer	Meaning
title	Any character string.

REMARKS:

1. PLOTID must appear before the OUTPUT(PLOT) or OUTPUT(XYOUT) commands.
2. The presence of PLOTID causes a special header frame to be plotted with the supplied identification plotted several times. The header frame allows plotter output to be identified easily.
3. If no PLOTID command appears, no ID frame will be plotted.
4. The PLOTID header frame will not be generated for the table plotters.

PLSTRN

Requests plastic strain output at grid points for SOL 401.

Requests plastic strain at grid points.

FORMAT:

$$\text{PLSTRN}\left[\left(\text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \left\{\begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix}\right\}$$

EXAMPLES:

```
PLSTRN=ALL
PLSTRN(PRINT,PLOT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

REMARKS:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

PRESSURE

Pressure Output Request

Requests form and type pressure output. Analogous to the DISPLACEMENT Case Control command.

See the description for DISPLACEMENT.

RANDOM

Random Analysis Set Selection

Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.

FORMAT:

RANDOM[(RPOSTS1 = i,RANCPLX = j,RMSSF = r)] = n

EXAMPLES:

```
RANDOM=177
RANDOM(RPOSTS1=1,RANCPLX=1,RMSSF=2.0)=123
```

DESCRIBERS:

Describer	Meaning
RPOSTS1 = i	Specifies the output format for random results. (Integer; Default = 0) RPOSTS1 = 0 for SORT2 output format. RPOSTS1 = 1 for SORT1 output format.
RANCPLX = j	Specifies the data format for random output. (Integer; Default = 0) RANCPLX = 0 for real data. RANCPLX = 1 for complex data.
RMSSF = r	Specifies a scaling factor for RMS and CRMS random results. (Real > 0.0; Default = 1.0)
n	Set identification number of RANDPS and RANDT1 Bulk Data entries to be used in random analysis. (Integer>0)

REMARKS:

1. RANDOM must select RANDPS Bulk Data entries to perform random analysis.

2. RANDOM must appear in the first subcase of the current loop. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command or changes in the K2PP, M2PP, or B2PP commands.

3. If RANDPS entries are used in a superelement analysis, the RANDOM command may be specified above the subcase level if a condensed subcase structure (SUPER=ALL) is used. If a condensed subcase structure is not used, then a unique RANDOM selection of a unique RANDPS entry must be specified within each of the desired superelement subcases.

4. If a describer is specified, it takes precedence over the corresponding parameter.

RCROSS

Cross-Power Spectral Density and Cross-correlation Function Output Requests

Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.

FORMAT:

$$\text{RCROSS}\left[\left(\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right],\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right],[\text{PUNCH}],\left[\begin{matrix}\text{PSDF}\\ \text{CORF}\\ \text{RALL}\end{matrix}\right]\right)\right]=\text{n}$$

EXAMPLES:

```
RCROSS(PHASE)=10
RCROSS(PSDF,NOPRINT,PUNCH)=20
```

DESCRIBERS:

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output (for cross-power spectral density function). Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output (for cross-power spectral density function). Phase output is in degrees.
PRINT	Writes output to print file. (Default)
NOPRINT	Does not write output to the print file.
PUNCH	Writes output to punch file.
PSDF	Requests the cross-power spectral density function be calculated and output for random analysis post-processing. (Default)

Describer	Meaning
CORF	Requests the cross-correlation function be calculated and output for random analysis post-processing.
RALL	Requests both the cross-power spectral density and cross-correlation function be calculated and output for random analysis post-processing.
n	Identification number of RCROSS and RCROSSC bulk data entries to be used in random analysis. (Integer > 0)

REMARKS:

1. The case control RCROSS entry must be used in conjuction with the case control RANDOM entry. See remarks under the RANDOM case control entry.

2. Response quantities, such as DISPLACEMENT, STRESS, and FORCE, must be requested by corresponding case control entries in order to compute cross-power spectral density and cross-correlation functions between the two response quantities specified by the RCROSS bulk data entry. It is recommended that the DISPLACEMENT, STRESS, and FORCE requests be put above the subcase level to ensure that these response quantities exist when the random analysis post-processing occurs.

3. The response quantities must belong to the same superelement. The cross-power spectral density and cross-correlation functions between two responses that belong to different superelements are not supported.

REPCASE

Repeat Output Subcase Delimiter

Delimits and identifies a repeated output subcase.

FORMAT:

REPCASE=n

EXAMPLES:

```
REPCASE=137
```

DESCRIBERS:

Describer	Meaning
n	Subcase identification number. (Integer>1)

REMARKS:

1. n must be strictly increasing (i.e., must be greater than all previous subcase identification numbers).
2. REPCASE defines a subcase that is used to make additional output requests for the previous real subcase. This command is required because multiple output requests for the same item are not permitted in the same subcase.
3. REPCASE may only be used in statics and normal modes problems. However, in normal modes, only output for the first mode is obtained. This feature is not supported in SOLs 106 or 153.
4. One or more repeated subcases (REPCASEs) must immediately follow the subcase (SUBCASE) to which they refer.

5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the REPCASE with a TEMP(LOAD) Case Control command or the element deformation state with a DEFORM command.

RESVEC

Residual Vector Request

Used to control the computation of residual vectors.

FORMAT:

$$\text{RESVEC}\left[\left(\begin{bmatrix}\text{APPLOD}\\ \text{NOAPPL}\end{bmatrix}, \begin{bmatrix}\text{RVDOF}\\ \text{NORVDO}\end{bmatrix}, \begin{bmatrix}\text{RVEL}\\ \text{NORVEL}\end{bmatrix}, \begin{bmatrix}\text{INRLOD}\\ \text{NOINRL}\end{bmatrix}, \begin{bmatrix}\text{DAMPLOD}\\ \text{NODAMP}\end{bmatrix}, \begin{bmatrix}\text{DYNRSP}\\ \text{NODYNRSP}\end{bmatrix}\right)\right] = \left\{\begin{matrix}\text{SYSTEM}\\ \text{NOSYSTEM}\\ \text{COMPONENT}\\ \text{NOCOMPONENT}\\ \text{BOTH or YES}\\ \text{NO}\end{matrix}\right\}$$

EXAMPLES:

```
RESVEC = SYSTEM
RESVEC(NOINRL) = COMPONENT
RESVEC = YES
```

DESCRIBERS:

Describer	Meaning
APPLOD	Compute residual vectors for applied loads. See Remark 5. (Default)
NOAPPL	Do not compute residual vectors for applied loads.
RVDOF	Compute residual vectors for degree-of-freedom included in RVDOF and RVDOF1 bulk entries. See Remark 6. (Default)

Describer	Meaning
NORVDO	Do not compute residual vectors for degree-of-freedom included in RVDOF and RVDOF1 bulk entries.
RVEL	Compute residual vectors for degree-of-freedom defining CBUSHi, CDAMPi, CELASi, and CVISC elements referenced by a RVEL bulk entry. See Remarks 7 and 9. (Default)
NORVEL	Do not compute residual vectors for degree-of-freedom defining CBUSHi, CDAMPi, CELASi, and CVISC elements referenced by a RVEL bulk entry.
INRLOD	Compute residual vectors for inertia relief load. See Remark 4. (Default)
NOINRL	Do not compute residual vectors for inertia relief load.
DAMPLOD	Compute residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements. See Remark 9. (Default)
NODAMP	Do not compute residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements.
DYNRSP	Include the dynamic effect of residual vector modes in the forced response solution. See Remark 8. (Default)
NODYNRSP	Do not include the dynamic effect of residual vector modes in the forced response solution.

The defaults for the following describers are solution dependent. The defaults apply even when the RESVEC case control entry is undefined. See Remark 10.

SYSTEM	Request residual vectors for residual structure degree-of-freedom only (a-set).
NOSYSTEM	Do not compute residual vectors for residual structure degree-of-freedom (a-set).
COMPONENT	Request residual vectors for superelement degree-of-freedom only (o-set). (Default for SOLs 103, 106 with PARAM, NMLOOP=0, 110, 115, 153 with PARAM, NMLOOP=0, 187, and 200 with ANALYSIS=MODES)
NOCOMPONENT	Do not compute residual vectors for superelement degree-of-freedom (o-set).

Describer	Meaning
BOTH (or YES)	Request residual vectors for both residual structure DOF (a-set) and for superelement DOF (o-set). (Default for SOLs 111, 112, 118, 146, and 200 with ANALYSIS=MCEIG, MTRAN, or MFREQ)
NO	Turns off calculation of residual vectors.

REMARKS:

1. If the RESVEC case control command is present, the parameters RESVEC and RESVINER are ignored.

2. A unique RESVEC case control definition can be defined in different superelement subcases. You can use the NOSYSTEM and NOCOMPONENT describers in a superelement subcase to turn off a setting which may have been defined globally.

3. If a RESVEC case control command is present in a cold start analysis, then only the RESVEC case control command can be used in a restart. Similarly, if parameter RESVEC/RESVINER is used in a cold start, then only the same can be used in a restart.

4. INRLOD designates that inertia load residual vectors are to be calculated. INRLOD is functionally equivalent to PARAM,RESVINER,YES. Inertia loads are computed for each of the 6 basic coordinate system directions and residual vectors are computed for each load.

5. APPLOD designates that the applied load residual vectors are to be calculated. There are two input scenarios to determine the applied loads for computing residual vectors.

   a. The load set IDs selected with the EXCITEID field on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries in the bulk data are processed. No DLOAD case control is required in this case. If a LOADSET case control command exists, these are all ignored.

   b. If a LOADSET case control command exists, and it selects an LSEQ bulk entry, the load set ID selected in the LID field in the LSEQ bulk entry is processed.

6. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is

different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.

The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

7. If a CBUSHi, CDAMPi, CELASi, and CVISC element is grounded, the unit load is applied to only the ungrounded end.

8. DYNRSP/NODYNRSP is only applicable to modal forced response solutions (SOLs 111, 112, 146, and 200 when ANALYSIS=MFREQ or MTRAN). For models containing superelements, DYNRSP/NODYNRSP is only applicable to the residual structure. For modal transient analysis, this control is functionally equivalent to the RESVALT parameter. For example, PARAM,RESVALT,YES is the same as NODYNRSP, and PARAM,RESVALT,NO is the same as DYNRSP.

9. DAMPLOD requests residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements. RVEL requests residual vectors for the degree-of-freedom associated with the CBUSHi, CDAMPi, CELASi, and CVISC elements selected on a RVEL bulk entry, regardless if damping is defined or not. If DAMPLOD and RVEL are both defined (default), and RVEL bulk data entries are defined, residual vectors will be requested for the combined degree-of-freedom list.

10. For SOLs 111, 112, 118, 146, and 200 (ANALYSIS=MCEIG, MTRAN, or MFREQ), the default is RESVEC=YES. For SOLs 103, 106 (PARAM,NMLOOP,0), 110, 115, 153 (PARAM,NMLOOP,0), 187, and 200 (ANALYSIS=MODES), the default is RESVEC=COMPONENT. These defaults apply even when the RESVEC case control entry is undefined.

11. Residual vectors are not supported in the rotor dynamics capability.

RIGID

Rigid Element Method

For SOLs 101, 103, 106 through 112, selects the rigid element processing method for RBAR, RBE1, RBE2, RBE3, RROD and RTRPLT elements.

For SOL 401, selects the rigid element processing method for RBAR and RBE2 elements.

FORMAT FOR SOLS 101, 103, 106 THROUGH 112:

$$\text{RIGID} = \left\{ \begin{array}{c} \text{LINEAR} \\ \text{LAGRAN} \end{array} \right\}$$

FORMAT FOR SOL 401:

$$\text{RIGID} = \left\{ \begin{array}{c} \text{AUTO} \\ \text{STIFF} \\ \text{LINEAR} \end{array} \right\}$$

EXAMPLE:

```
RIGID=LAGRAN
```

DESCRIBERS FOR SOLS 101, 103, 106 THROUGH 112:

Describer	Meaning
LINEAR	Selects the linear elimination method.
LAGRAN	Selects the Lagrange multiplier method.

DESCRIBERS FOR SOL 401:

Describer	Meaning
AUTO	The software automatically determines the RBE2 and RBAR element behaviour based on the PARAM,LGDISP setting.
STIFF	RBE2 and RBAR elements include thermal expansion. Large displacement effects are determined by the PARAM,LGDISP setting.
LINEAR	RBE2 and RBAR elements do not include large displacement effects or thermal expansion.

REMARKS:

1. The RIGID command must be above the SUBCASE level.

2. The RIGID command is supported in SOLs 101, 103, 106 through 112, and 401. For all other solution sequences, the RIGID command is ignored and RIGID = LINEAR is used.

3. For SOLs 101, 103, 106 through 112:

   - The LAGRAN method allows for the thermal expansion of the rigid elements.

   - If the RIGID command is not specified, RIGID = LINEAR is used.

   - LINEAR processing will not compute the thermal loads. Also, in SOLs 103, 106 through 112, LAGRAN method must be used to compute the differential stiffness due to the thermal expansion of the rigid elements.

   - The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

     The rigid element thermal strains are calculated from

     $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$

     If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

- When using RIGID = LAGRAN, K6ROT must be defined as non-zero.
- For additional information, see “Rigid Element Processing Options” in the *Element Library Reference*.

4. In SOL 401, the RIGID command and the LGDISP parameter settings determine the RBE2 and RBAR element behavior.
   - When RIGID=AUTO and PARAM,LGDISP,-1, the software automatically applies the RIGID=LINEAR option. RBE2 and RBAR elements do not include large displacement effects or thermal expansion.
   - When RIGID=AUTO and PARAM,LGDISP,1, the software automatically applies the RIGID=STIFF option. RBE2 and RBAR elements include large displacement effects and thermal expansion.
   - When RIGID=STIFF and PARAM,LGDISP,-1, the large displacement effects are not included, in general. RBE2 and RBAR elements include thermal expansion.
   - When RIGID=STIFF and PARAM,LGDISP,1, the RBE2 and RBAR element behavior is the same as RIGID=AUTO and PARAM,LGDISP,1. RBE2 and RBAR elements include large displacement effects and thermal expansion.
   - When RIGID=LINEAR, the RBE2 and RBAR elements do not include large displacement effects or thermal expansion. This behavior is independent of the PARAM,LGDISP setting.
   - The TEMP(LOAD) and TEMP(INIT) value used on RBAR elements is an average calculated from the grid point values. On RBE2 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

     The rigid element thermal strains are calculated from

     $\varepsilon_{thermal} = \alpha(AVGTEMP(LOAD) - AVGTEMP(INIT))$

     If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.
   - MPCFORCE and GPFORCE output are supported with all of the rigid elements. Since the software internally replaces an RBAR or RBE2 with a stiff beam or spring element when RIGID=STIFF, these elements are no longer represented as MPC equations. As a result, MPCFORCE output is not applicable to these elements. GPFORCE and FORCE output is applicable.
   - For additional information, see Rigid Element Processing in the *Multistep Nonlinear User’s Guide*.

RMAXMIN

Requests output of MIN/MAX and RMS values from solutions 101, 109, and 112.

Defines parameters to output the minimum, maximum, absolute value maximum, average, and RMS value of stress, force and displacement results generated during solutions 101, 109, and 112.

FORMAT:

$$\text{RMAXMIN}\left(\{\text{FORCE,STRESS,DISP}\},\ \begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix},\ [\text{PUNCH,PLOT}],\ \begin{bmatrix}\text{ABSOLUTE}\\ \text{MINIMUM}\\ \text{MAXIMUM}\end{bmatrix},\ \text{RMXTRAN} = \begin{Bmatrix}yes\\ no\end{Bmatrix},\ \text{NPAVG=n, START=a, END=b}\right) = \begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

EXAMPLES:

```
RMAXMIN(NOPRINT,STRESS,ABSOLUTE)=YES
RMAXMIN(STRESS,PRINT,MAX,NPAVG=3,START=.3,END=1.2) = yes
```

DESCRIBERS:

Describer	Meaning
FORCE, STRESS, DISP	Result tables to be searched.
PRINT	Writes output to the print file. (Default)
NOPRINT	Does not write output to the print file.
PUNCH	Writes output to the punch file.
PLOT	Writes output to .plt file.

Describer	Meaning
ABSOLUTE	Specifies output of absolute maximum values. See Remarks 1, 2 & 4.
MINIMUM	Specifies output of minimum values. See Remarks 1 & 2.
MAXIMUM	Specifies output of maximum values. (Default) See Remarks 1 & 2.
RMXTRAN	Turns on or off the normal printing of output data when RMAXMIN output is requested. See Remark 7.
= YES	Results will be output according to the DISPLACEMENT, STRESS, and FORCE case control entries.
= NO	Results are not output if RMAXMIN is activated.
NPAVG	Defines the number of extremes in a subcase history (SOL 101) or peaks in a time history (SOL 109 and 112) to be averaged. See Remarks 2 & 3. (Default=1)
START=a,END=b	Limits the RMAXMIN evaluation over a specific range in the subcase history (SOL 101) or the time history (SOL 109 and 112). See Remarks 1 & 4.
YES	Activates RMAXMIN.
NO	Disables RMAXMIN.

REMARKS:

1. For RMAXMIN to process force output, the FORCE describer should be defined on the RMAXMIN command, and force output must be requested with either the FORCE case control command or using the XYPUNCH/XYPLOT commands.

   For RMAXMIN to process stress output, the STRESS describer should be defined on the RMAXMIN command, and stress output must be requested with either the STRESS case control command or using the XYPUNCH/XYPLOT commands.

   For RMAXMIN to process one of displacement, velocity, or acceleration output, the DISP describer should be defined on the RMAXMIN command, and only one of displacement, velocity, or acceleration output must be requested with either the DISPLACEMENT, VELOCITY, or ACCELERATION case control commands or using the XYPUNCH/XYPLOT commands.

RMAXMIN will process displacement, velocity, or acceleration output, if requested, together as a single output type, and not separately. As a result, you should only request one of displacement, velocity, or acceleration output at a time for RMAXMIN processing.

2. RMAXMIN outputs maximum/minimum/absolute tables and RMS tables. Both tables are evaluated using the START and/or END values when they are defined. Only one of MAXIMUM (default), MINIMUM and ABSOLUTE can be selected, and the one selected determines the values reported in the maximum/minimum/absolute tables. The RMS tables are independent of this selection.

3. NPAVG is the number of extremes in a subcase range (SOL 101) or the number of peak responses (see Remark 3) in a time range (SOLs 109 and 112) to be averaged when calculating the maximum/minimum/absolute tables. The extremes/peak responses used depend on which of MAXIMUM, MINIMUM, or ABSOLUTE is defined. When NPAVG=1, an extreme value is reported with no averaging. NPAVG is ignored by the RMS calculation.

4. In solutions 109 and 112, a peak search algorithm is used to determine the peak responses in the time range. This algorithm is comparing peaks relative to one another to find “NPAVG” peak responses for averaging.

5. When ABSOLUTE is defined, the software does the following.
   a. NPAVG maximum peaks and NPAVG minimum peaks are found.
   b. NPAVG maximum peaks are determined from the absolute values of all peaks found in “a”.

6. The START=a and END=b describers can be used to limit the RMAXMIN evaluation to a specific subcase or time range. By default, START is the beginning, and END is the final, such that one can be defined without the other.

7. RMAXMIN output is always SORT1.

8. The parameter RMXTRAN can also be used to control the output of transient results for STRESS, FORCE, DISP case control when RMAXMIN is activated, but only if the RMXTRAN input on RMAXMIN is not defined. The parameter default, RMXTRAN=NO will suppress output of transient results. The RMXTRAN describer on the RMAXMIN case control command, when defined, takes precedence over the RMXTRAN parameter.

9. The output datablocks generated by RMAXMIN are OES1MX, OEF1MX, and OUGV1MX for stress, force and displacement respectively.

10. The maximum, minimum, absolute, and RMS results are captured and stored on a component basis in the OES1MX, OEF1MX, and OUGV1MX datablocks. For example, the maximum displacement is computed and stored for all six DOF on a grid and each value may correspond to different times in the transient history.

11. When the RMAXMIN case control command is defined in a SOL 112 run, the parameter entry PARAM,RMXPANEL,PSIZE can be used to process the data recovery in a series of smaller “panels” rather than the entire output set at once. This will reduce the overall amount of scratch disk space required for the run.

    “PSIZE” is the number of timesteps to process in a single panel. The highwater disk space used will be a function of psize, and not of the total output request. A good initial value for psize is the greater of 100 or the square root of the number of timesteps. Setting psize =0 (default) will disable the panel method and process the data recovery for all the time steps at once. The START, END, and RMXTRAN=YES keyword options on the RMAXMIN command are not supported when using RMXPANEL. The RMXPANEL method is not compatible with PARAM,DDRMM,-1.

RMETHOD

ROTORD Selection

Selects the ROTORD bulk entry used for rotor dynamics analysis. Optionally selects complex modal reduction when using SOL 107 for rotor dynamics analysis.

FORMATS:

RMETHOD[(CMR=m)]=n

EXAMPLES:

```
RMETHOD=77
RMETHOD(CMR=3)=77
```

DESCRIBERS:

Describer	Meaning
CMR = m	Request complex modal reduction using the complex eigenvalue method and parameters specified on the EIGC bulk entry having identification number m. See Remark 4. (Integer ≥ 0; If m = 0, complex modal reduction is not used; Default = 0)
n	Identification number of a ROTORD bulk entry. (Integer>0)

REMARKS:

1. ROTORD bulk entries will not be used unless selected in the case control section.
2. RMETHOD must be located above any subcases.
3. RMETHOD is supported in SOLs 107 – 112.
4. The CMR = m describer is ignored for any solution sequence other than SOL 107.

RSMETHOD

Real Eigenvalue Extraction Method Selection

Selects the real eigenvalue extraction parameters in the component mode reduction of the residual structure for direct and modal solutions.

FORMAT:

RSMETHOD = n

EXAMPLES:

```
RSMETHOD=33
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of an EIGR or EIGRL Bulk Data entry. (Integer>0)

REMARKS:

1. RSMETHOD specifies the set identification number of an EIGR or EIGRL entry to be used in the computation of the normal modes on the o-set (fixed boundary modes) or v-set (mixed boundary modes) of the residual structure depending on the b-set and/or c-set entries. These modes are calculated for a component mode reduction of the residual structure when QSET type entries exist. If QSET type entries do not exist in the residual structure a component mode reduction for SE 0 is not performed.

2. Component mode reduction of a residual structure is used to account for local residual flexibilitiy in the residual. Similar to component mode reduction of a superelement in which residual flexibility effects are obtained for the exterior DOF, a component mode reduction on the residual accounts for residual flexibility at DOF identified with ASET, BSET, and CSET type entries.

3. If the set identification number selected is present on both EIGRL and EIGR entries, the EIGRL entry will be used. This entry requests the Lanczos eigensolution method.

4. For CMR of RS the user must have QSET type entries along with ASET, BSET, and CSET type entries. In contrast, when using CMR of a superelement other than 0, the user must use SEBSET, SECSET, SEQSET, or SESET type entries.

5. If any of the following conditions exist, a Guyan reduction is executed and the results may be poor because of the absence of a dynamic portion of the reduction:

   - DOF referenced by QSET do not exist.

   - RSMETHOD specified but QSET does not exist.

SACCELERATION

Solution Set Acceleration Output Request

Requests the form and type of solution set acceleration output.

FORMAT:

$$\text{SACCELERATION}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \text{PRINT}, \text{PUNCH}\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
SACCELERATION=ALL
SACCELERATION(PUNCH,IMAG)=142
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Acceleration for all solution set points (modes) will be output.

Describer	Meaning
NONE	Acceleration for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Acceleration output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis and are discussed under DISPLACEMENT. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SACCELERATION=NONE allows an overall output request to be overridden.
4. Due to the differences in formulations, the modal responses for the constraint mode method of enforced motion will be different than the modal responses for the absolute displacement method of enforced motion. In order to highlight this difference, the f06 output for the constraint mode method will be labeled *alternative* solution set.

SDAMPING

Structural Damping Selection

Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.

FORMAT:

$$\text{SDAMPING}\left[\left(\begin{array}{c}\text{STRUCTURE}\\ \text{FLUID}\end{array}\right)\right] = \text{n}$$

EXAMPLES:

```
SDAMPING=77
```

DESCRIBERS:

Describer	Meaning
STRUCTURE or FLUID	Modal damping is requested for the structural or fluid portion of the model.
n	Set identification number of a TABDMP1 (modal solutions such as 111 or MFREQ in SOL 200) or TABLEDi (direct solutions such as SOL 108 or DFREQ in SOL 200) bulk data entry. (Integer>0) See Remarks 1 and 2.

REMARKS:

1. In the modal solutions (e.g., SOLs 110, 111, 112, 145, 146, and 200), SDAMPING must reference a TABDMP1 entry.

2. In direct frequency response (e.g., SOL 108), SDAMPING must reference a TABLEDi entry with ID=n and another TABLEDi entry with ID=n+1, which define

viscoelastic (frequency-dependent) material properties. See the *NX Nastran Advanced Dynamic Analysis User's Guide*.

3. SDAMPING may be requested for superelements as long as PARAM,SESDAMP,YES is specified.

4. The SDAMPING command is supported across subcases. A SDAMPING command selecting a different TABDMP1 or TABLEDi bulk data entry can be defined for each subcase.

SDISPLACEMENT

Solution Set Displacement Output Request

Requests the form and type of solution set displacement output.

FORMAT:

$$\text{SDISPLACEMENT}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \text{PRINT}, \text{PUNCH}\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}\right)\right] = \begin{bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{bmatrix}$$

EXAMPLES:

```
SDISPLACEMENT=ALL
SDISPLACEMENT(SORT2,PUNCH,PHASE)=NONE
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
ALL	Displacements for all solution set points (modes) will be output.
NONE	Displacements for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only displacements on points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. The defaults for SORT1 and SORT2 depend on the type of analysis and are discussed in Remark 2 under the "DISPLACEMENT" . If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.

2. SDISPLACEMENT=NONE allows an overall output request to be overridden.

3. The SDISPLACEMENT command is required to output normalized complex eigenvectors.

4. Due to the differences in formulations, the modal responses for the constraint mode method of enforced motion will be different than the modal responses for the absolute displacement method of enforced motion. In order to highlight this difference, the f06 output for the constraint mode method will be labeled *alternative* solution set.

SEALL

Superelement Generation and Assembly

Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.

FORMAT:

$$\text{SEALL} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEALL=ALL
SEALL=7
```

DESCRIBERS:

Describer	Meaning
ALL	Generate and assemble all superelements.
n	Set identification number of a previously appearing SET command. Only superelements with identification numbers that appear on this SET command will be generated and assembled. (Integer>0)
i	Identification number of a single superelement that will be generated and assembled. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. This command combines, in one command, the functions of the SEMG, SELG, SEKR, SELR, and SEMR commands.

6. This command does not control superelement data recovery (Phase 3). See SEDR.

7. SEALL=ALL is the default in the Structured Solution Sequences (SOLs 101 through 200). This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEDR

Superelement Data Recovery

Specifies the superelement identification numbers for which data recovery will be performed.

FORMAT:

$$\text{SEDR} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEDR=ALL
SEDR=7
```

DESCRIBERS:

Describer	Meaning
ALL	Performs data recovery for all superelements.
n	Set identification number of a previously appearing SET command. Data recovery will be performed for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which data recovery will be performed. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. If this command is not present, data recovery is performed for all superelements for which there are output requests (i.e., the default for this command is SEDR=ALL).

SEDV

Superelement Design Variable Processing

Specifies the superelement identification numbers for which the design variables will be processed.

FORMAT:

$$\text{SEDV} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEDV=ALL
SEDV=18
```

DESCRIBERS:

Describer	Meaning
ALL	Requests design variable processing for all superelements.
n	Set identification number of a previously appearing SET command. Design variable processing will be performed for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which design variable processing will be performed. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.

3. For a further discussion of superelement sensitivity analysis, see the *NX Nastran Design Sensitivity and Optimization User's Guide*.

4. Zero (0) can only appear as a member of a SET.

5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

6. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

SEEXCLUDE

Superelement Matrix and Load Assembly Exclusion

Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.

FORMAT:

$$\text{SEEXCLUDE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEEXCLUDE=ALL
SEEXCLUDE=18
```

DESCRIBERS:

Describer	Meaning
ALL	All upstream superelements will be excluded.
n	Set identification number of a previously appearing SET command. Only those superelements with identification numbers that appear on this SET command will be excluded. (Integer>0)
i	Identification number of a single superelement for which matrices will be excluded. (Integer>0)

REMARKS:

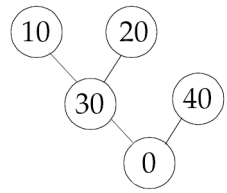
1. If this command is present, then it must be located before the first SUBCASE command.

2. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

3. This command is not meaningful when applied to the residual structure.

4. For a further discussion of this command, see the *NX Nastran Superelement User's Guide.*

5. If the SEEXCLUDE command is specified in a restart of SOLs 101 through 200, then PARAM,SERST,MANUAL must be specified. Also the SEKR command must be specified for the superelement immediately downstream from the excluded superelement. For example, if superelement 10 is excluded in the superelement tree:



then you must specify the following commands in the Case Control Section:

```
SEKR=30
PARAM,SERST,MANUAL
```

SEFINAL

Final Superelement for Assembly

Specifies the superelement identification number for the final superelement to be assembled.

FORMAT:

$$\text{SEFINAL} = \left\{ \begin{array}{c} n \\ i \end{array} \right\}$$

EXAMPLES:

```
SEFINAL=14
```

DESCRIBERS:

Describer	Meaning
n	Set identification of a previously appearing SET command. Each superelement identification number appearing on the SET command must belong to a disjoint model. (Integer>0)
i	Identification number of the final superelement to be assembled. (Integer>0)

REMARKS:

1. If this command is not present, the program selects the order of the superelements for assembly operations.
2. If this command is present, then it must be located before the first SUBCASE command.
3. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

4. This command can be used on restarts to ensure that minor modeling changes do not also change the processing order. For this usage, inspect the SEMAP table to determine which superelements were final superelements on the prior run.

5. See the *NX Nastran Superelement User's Guide* for a further discussion of this command.

6. If all of the superelements in the model are on a single level, (i.e. No superelement connections except to residual structure), then SEFINAL is ignored and superelement processing order is set according to the superelement ID.

SEKREDUCE

Superelement Stiffness Matrix Assembly and Reduction

Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.

FORMAT:

$$\text{SEKREDUCE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEKREDUCE=ALL
SEKREDUCE=9
```

DESCRIBERS:

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the stiffness matrix will be assembled and reduced. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. SEKREDUCE is an alternate form and is entirely equivalent to the obsolete command SEMASSEMBLE.

5. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELGENERATE

Superelement Load Generation

Specifies the superelement identification numbers for which static loads will be generated.

FORMAT:

$$\text{SELGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SELGENERATE=ALL
SELGENERATE=18
```

DESCRIBERS:

Describer	Meaning
ALL	Generates static loads for all superelements.
n	Set identification number of a previously appearing SET command. Static load matrices will only be generated for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which load matrices will be generated. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELREDUCE

Superelement Load Assembly and Reduction

Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.

FORMAT:

$$\text{SELREDUCE}=\left\{\begin{array}{c}\text{ALL}\\ \text{n}\\ \text{i}\end{array}\right\}$$

EXAMPLES:

```
SELREDUCE=ALL
SELREDUCE=9
```

DESCRIBERS:

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will be assembled only for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the load matrices will be assembled and reduced. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number for the residual structure and can only be appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. This command is used on restarts to selectively assemble and reduce load matrices.

5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

6. In superelement static analysis, SELREDUCE is equivalent to SELASSEMBLE.

7. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.

8. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMGENERATE

Superelement Matrix Generation

Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.

FORMAT:

$$\text{SEMGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEMGENERATE=ALL
SEMGENERATE=7
```

DESCRIBERS:

Describer	Meaning
ALL	Generates structural matrices for all superelements.
n	Set identification number of a previously appearing SET command. Structural matrices will only be generated for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which structural matrices will be generated. (Integer>0)

REMARKS:

1. If this command is present, then is must be located before the first SUBCASE command.

2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMREDUCE

Superelement Mass and Damping Assembly and Reduction

Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices will be assembled and reduced.

FORMAT:

$$\text{SEMREDUCE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SEMREDUCE=ALL
SEMREDUCE=9
```

DESCRIBERS:

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the load matrices or the mass and damping matrices will be assembled and reduced. (Integer>0)

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number for the residual structure and can only appear as a member of a set.

3. This command is used on restart to selectively assemble and reduce mass and damping matrices. For a further discussion of this command, see the *NX Nastran Superelement User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.

6. This command has no function in static analysis.

7. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEQDEP

Solution sequence dependency selection for SOL 401

Defines whether or not a subcase is sequentially dependent on the previous subcase.

FORMAT:

SEQDEP = {YES or NO}

EXAMPLES:

```
SEQDEP = NO
```

DESCRIBERS:

Describer	Meaning
SEQDEP=YES (Default)	Subcases are sequentially dependent. See Remark 260-3.
SEQDEP=NO	Subcases are non-sequentially dependent.

REMARKS:

1. Only supported in SOL 401.
2. SEQDEP may be defined above the subcases (globally) or within a subcase.
3. A sequentially dependent static subcase (SEQDEP=YES) uses the end time from the previous static subcase for its start time. In addition, a sequentially dependent static or modal subcase uses the displacement/stress/strain state from the previous static subcase for its starting state.

   A non-sequentially dependent static subcase (SEQDEP=NO) has a start time of zero. In addition, a non-sequentially dependent static or modal subcase does not use the displacement/stress/strain state from the previous static subcase.

   See Subcase sequencing in the Multi-Step Nonlinear User's Guide.

4. SEQDEP is ignored in the first subcase of a cold start. That is, in a solution which is not a restart.

5. SEQDEP=YES is the default. The default applies to a subcase when the SEQDEP case control entry is undefined.

SERESP

Superelement Response Sensitivity

Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.

FORMAT:

$$\text{SERESP} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

EXAMPLES:

```
SERESP=ALL
SERESP=18
```

DESCRIBERS:

Describer	Meaning
ALL	Requests design sensitivity matrix generation for all superelements.
n	Set identification number of a previously appearing SET command. Design sensitivity matrices will be generated for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the design sensitivity matrix will be generated.

REMARKS:

1. If this command is present, then it must be located before the first SUBCASE command.

2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.

3. For a further discussion of this command, see the *NX Nastran Design Sensitivity and Optimization User's Guide*.

4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.

5. If both SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

SET

Set Definition, General Form

Sets are used to define the following lists:

LISTS

1. Identification numbers (point, element, or superelement) for processing and output requests.

2. Times for which output will be printed in transient response problems using the OTIME case control command or the MODCON case control command with the SOLUTION = *setout* describer specified.

3. Frequencies for which output will be printed in frequency response problems using the OFREQ case control command or the MODCON case control command with the SOLUTION = *setout* describer specified or the PANCON case control command with the SOLUTION = *setf* describer specified.

4. Panels for which output will be printed in frequency response problems using either the PANCON or ERP case control commands.

   - For the PANCON command, the PANEL = *setp* describer references the identification number of the SET that lists the alphanumeric names of panels defined by PANEL bulk entries.

   - For the ERP command, the ERP = *setp* describer references the identification number of the SET that lists the alphanumeric names of panels defined by PANEL bulk entries.

5. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD case control commands.

6. With SOL 200, DRESP1 design responses that are assigned to a specific subcase via a DRSPAN case control command which refers to a particular SET.

Formats:

SET n = {i_1[,i_2, i_3, THRU i_4, EXCEPT i_5, i_6, i_7, i_8, THRU i_9]}

SET n = {r_1, [r_2, r_3, r_4]}

SET n = {$name_1$,[$name_2$,$name_3$,$name_4$]}

SET n = ALL

EXAMPLES:

```
SET 77=5
SET 88=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU 300
SET 99=1 THRU 100000
SET 101=1.0, 2.0, 3.0
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
SET 5=PANEL1,PANEL3,PANEL4
```

DESCRIBERS:

Describer	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only. (Integer>0)
i_1, i_2, etc.	Identification numbers. If no such identification number exists, the request is ignored. (Integer ≥ 0)
i_3 THRU i_4	Identification numbers (i_4>i_3). (Integer>0)
EXCEPT	Set identification numbers following EXCEPT will be deleted from the output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.
r_1, r_2, etc.	Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ, OTIME, MODCON (with SOLUTION = *setout* describer specified), or PANCON (with SOLUTION = *setf* describer specified) case control command references the set, then the values must be listed in ascending sequences, r_1>r_2>r_3>r_4...etc., otherwise some output may be missing. (Real>0.0)
$name_1$, $name_2$, etc.	Alphanumeric names of panels defined by PANEL bulk entries. (Character)
ALL	All members of the set will be processed. This option may not be used in a DRSPAN referenced set.

REMARKS:

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.

2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.

3. In SET 88 in the example section above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.

SET

Set Definition OUTPUT(PLOT)

Defines a set of element or grid point numbers to be plotted.

FORMAT:

$$\text{SET n} = \left[\text{ALL}\left[\begin{matrix}\text{ELEMENTS}\\ \text{GRID POINTS}\end{matrix}\right]\left[\text{EXCEPT}\left\{\begin{matrix}\text{type1 type2 } \ldots \text{ typej}\\ \text{k1 k2 } \ldots \text{ kj THRU kk BY incj}\end{matrix}\right\}\right]\right],$$

$$\left[\left[\begin{matrix}\text{INCLUDE}\\ \text{EXCLUDE}\end{matrix}\right]\left[\begin{matrix}\text{ELEMENTS}\\ \text{GRID POINTS}\end{matrix}\right]\left\{\begin{matrix}\text{type1 type2 } \ldots \text{ typej}\\ \text{k1 k2 } \ldots \text{ kj THRU kk BY incj}\end{matrix}\right\},\right.$$

$$\left.\left[\text{EXCEPT}\left\{\begin{matrix}\text{type1 typem } \ldots \text{ typen}\\ \text{k1 km } \ldots \text{ kn THRU ko BY incn}\end{matrix}\right\}\right]\right]$$

EXAMPLES:

1. SET 1 consists of elements 1, 5, 10, 11, 13, 14, 15, 20, 22, 24, and 26.

```
SET 1=INCLUDE 1, 5, 10 THRU 15 EXCEPT 12, INCLUDE 20
THRU 26 BY 2
```

2. SET 2 consists of all CTRIA3 and CQUAD4 elements except element 21.

```
SET 2=QUAD4 TRIA3 EXCEPT 21
```

3. SET 10 includes all CTRIAR elements plus elements 70 through 80.

```
SET 10  TRIAR  INCLUDE ELEMENTS 70 THRU 80
```

4. SET 15 includes all elements from 15 to 20 and 26 to 100.

```
SET 15=15 THRU 100 EXCEPT 21 THRU 25
```

5. SET 2 includes all elements except CTETRA elements.

```
SET 2=ALL EXCEPT TETRA
```

DESCRIBERS:

Describer	Meaning
n	Set identification number. (0<Integer<999999)
ALL	Selects all elements or grid points. See typei.
ELEMENTS	Specifies that all identification numbers refer to elements.
GRID POINTS	Specifies that all identification numbers refer to grid points.
INCLUDE	Includes specified element or grid point identification numbers or elements in the set.
EXCLUDE	Excludes specified element or grid point identification numbers or element types in the set.
EXCEPT	Modifies a prior ALL, INCLUDE, or EXCLUDE specification.
typei	Element types. The allowed element types are (Character):

typei	Element Entry Name
TRIA3	CTRIA3
TRIA6	CTRIA6
TRIAR	CTRIAR
QUAD4	CQUAD4
QUAD8	CQUAD8
QUADR	CQUADR
HEXA	CHEXA
PENTA	CPENTA
TETRA	CTETRA

THRU	Specifies a range of identification numbers.
BY	Specifies an increment for a THRU specification.
inci	Increment for THRU range. (Integer>0)

REMARKS:

1. This form of the SET command must and can only be specified after an OUTPUT(PLOT) delimiter.

2. The INCLUDE, EXCLUDE, and EXCEPT specifications may be specified more than once in the same set. See Examples above.

3. Commas or spaces may be used as separators.

4. Not all of the identification numbers in a THRU range have to correspond to elements or grid points. For example, elements 2, 4, 7, and 9 may be selected with 2 THRU 9, even if elements 3, 5, 6, and 8 do not exist. This is called an open set. It should be noted that large open sets can cause higher computational costs.

SETMC

Modal Contribution Set Definition

Set definitions for modal contribution results.

FORMATS:

SETMC n = rtype / id1 (item1), id2 (item2), …

EXAMPLES:

```
SETMC 121 = ACCE/99(T3),1200(T1),1399(R2)
SETMC 222 = STRESS/134(22)
SETMC 343 = ACCE/99(T3),1200(T1),1399(R2),STRESS/134(22)
```

DESCRIBERS:

Describer	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETMCs specified under a SUBCASE command are recognized for that SUBCASE only. (Integer>0)
rtype	

Response type (Character)	
ACCE	Acceleration
DISP	Displacement
ELFO	Element force
SPCF	Single-point force of constraint
STRA	Strain
STRE	Stress
VELO	Velocity

idj	Element or grid point identification number for j-th value. (Integer>0)

Describer	Meaning
Itemj	Item code for j-th value. (Character or integer>0) For elements, the code represents a component of the element stress or force and is described in the Item Codes topic in this guide. For grid points, the code is one of the mnemonics T1, T2, T3, R1, R2, or R3, where Ti stands for the i-th translation component and Ri stands for the i-th rotational component.

REMARKS:

1. The SETMC command may be continued on the next line as long as “SETMC n = rtype/” is specified on the first line.

2. A SETMC command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. A slash (i.e. “/”) or a parenthesis i.e. (“(” or “)”) may not begin a continuation line.

SETMCNAME

Specifies the title of a displacement monitor plot in a SOL 401 run with Simcenter Multiphysics.

Specifies the title of a displacement monitor plot in a SOL 401 run with Simcenter Multiphysics.

FORMAT:

SETMCNAME *n* = *title of plot*

EXAMPLES:

```
MONVAR=ALL
SETMC 21 = DISP/ 1001(T1), 1002(T1), 1204(T1)
SETMCNAME 21 = Blade 3
SETMC 25 = DISP/ 1001(T2), 1002(T2), 1204(T2)
SETMCNAME 25 = Compressor Disk
```

DESCRIBERS:

Describer	Meaning
n	ID of a corresponding SETMC case control command. The SETMC and SETMCNAME commands with matching IDs are referenced by a MONVAR case control command. See the examples.
plot title	Displacement monitor plot title. (Character string)

REMARKS:

1. Only supported in a SOL 401 run with Simcenter Multiphysics.
2. A displacement monitor plot is created for each SETMC ID selected by the MONVAR bulk entry. Each displacement monitor plot includes a plot of the maximum, the minimum, and the average displacement. The software uses

the displacement values for all degree-of-freedom listed on a particular SETMC to determine the maximum and minimum displacement, and to compute the average displacement for each converged time step. If you want to monitor the displacement of a single degree-of-freedom, you should select a SETMC which includes only this degree-of-freedom.

3. You can optionally define a title for each displacement monitor plot using the SETMCNAME command. The ID on the SETMCNAME command should match the ID on the SETMC command. See the Examples. When the SETMCNAME is undefined, the ID of the SETMC is used for the plot title.

SETS DEFINITION

Case Control Processing Delimiter

Delimits the various type of commands under grid point stress. This command is synonymous with OUTPUT(POST)

FORMAT:

SETS DEFINITION

EXAMPLES:

```
SETS DEFINITION
```

REMARKS:

SETS DEFINITION is synonymous with OUTPUT(POST). Either SETS DEFINITION or OUTPUT(POST) may be specified but not both.

SHELLTHK

Shell Thickness Output Request

Requests the form of shell thickness output (SOLs 601 and 701).

FORMAT:

$$\text{SHELLTHK}\left(\left\{\begin{matrix}\text{PRINT}\\ \text{PLOT}\end{matrix}\right\}\right)=\text{ALL or NONE}$$

EXAMPLES:

```
SHELLTHK(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PLOT	Computes and puts shell thickness results in op2 file only.
ALL	Shell thickness results will be output for all applicable shell elements. See remark 1.
NONE	Shell thickness results will not be output.

REMARKS:

1. Shell thickness results are output only for large strain analysis, i.e., PARAM,LGSTRN,1. Note that large strain formulation is not available for 8-node shell elements. However, by specifying ELCV=1 in the NXSTRAT entry, 8-node shell elements will be converted to 9-node shell elements which support large strain.

2. Large strain shell elements require the use of an elastic-plastic material through the MATS1 entry with TYPE=PLASTIC. On the MATS1 entry, either TID must be specified to define a multilinear plastic material, or H and LIMIT1 must be specified to define a bilinear plastic material for the shell elements.

3. Shell thickness results are output at nodes on elements.

SKIP

Case Control Processing Delimiter

Activates or deactivates the execution of subsequent commands in the Case Control (including plot commands).

FORMAT:

$$\text{SKIP}\left\{\begin{array}{c}\text{ON}\\ \text{OFF}\end{array}\right\}$$

EXAMPLES:

```
SKIPOFF
```

REMARKS:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control.

2. Commands that are skipped will be printed.

3. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows the user to omit requests without deleting them from his data. In the following example, plot commands will be skipped.

```
TITLE = EXAMPLE
SPC = 5
LOAD = 6
SKIPON $ SKIP PLOT REQUEST
OUTPUT (PLOT)
SET 1 INCLUDE ALL
FIND
PLOT
BEGIN BULK
```

SMETHOD

Iterative Solver Method Selection

Selects iterative solver parameters.

FORMAT:

SMETHOD=n

EXAMPLES:

```
SMETHOD=77
```

DESCRIBERS:

Describer	Meaning
n	Set identification of an ITER Bulk Data entry. (Integer>0)

REMARKS:

1. The iterative solver is requested by specifying ITER=YES on the NASTRAN statement. The SMETHOD command is optional and only required to override defaults shown on the ITER Bulk Data entry.
2. SMETHOD is only available in SOLs 101, 106, 108, 111, and 401.

SPC

Single-Point Constraint Set Selection

Selects a single-point constraint set to be applied.

FORMAT:

SPC=n

EXAMPLES:

```
SPC=10
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a single-point constraint that appears on a SPC, SPC1, or SPCADD Bulk Data entry. (Integer>0)

REMARKS:

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary conditions are only supported in SOLs 101, 103, 105, 106, 145 and 200. Multiple boundary conditions are not allowed for upstream superelements. The BC command must be specified to define multiple boundary conditions for the residual structure in SOLs 103, 105, 145 and 200.

SPCFORCES

Single-Point Forces of Constraint Output Request

Requests the form and type of single-point force of constraint vector output.

FORMAT:

$$\text{SPCFORCES}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
SPCFORCES=5
SPCFORCES(SORT2,PUNCH,PRINT,IMAG)=ALL
SPCFORCES(PHASE)=NONE
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.

Describer	Meaning
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, single-point forces of constraint.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 11.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 11.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 11.
RPUNCH	Writes random analysis results to the punch file. See Remark 11.

Describer	Meaning
ALL	Single-point forces of constraint for all points will be output. See Remark 2 and Remark 5.
NONE	Single-point forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only single-point forces constraint for points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

3. In a statics solution, a SORT2 request will output at the grids which have both zero and nonzero SPCFORCES. A SORT1 (default) request will only output at the grids which have nonzero SPCFORCES.

4. SPCFORCES=NONE overrides an overall output request.

5. In SORT1 format, SPCFORCESs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of

output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.

6. SPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).

7. In SOLs 129 and 159, SPCFORCES results do not include the effects of mass and damping elements.

8. In all solution sequences except SOLs 129 and 159, SPCFORCES results do include the effects of mass and damping, except damping selected by SDAMPING command. PARAM,DYNSPCF,OLD may be specified to obtain SPCFORCES results, which do not include mass and damping effects.

9. In inertia relief analysis, the SPCFORCE output includes both the effects due to inertial loads and applied loads.

10. In SOL 106, the moments in the SPCFORCE output are computed in the undeformed configuration. They do not include the effects of deformations.

11. The following applies to random solutions:

    - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

    - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

    - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

REMARKS RELATED TO SOLS 601 AND 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC, RMS, and RALL are ignored.

2. SPCFORCES is not supported for SOLs 601,153 and 601,159.

STATSUB

Static Solution Selection for Differential Stiffness

Selects the static solution to use in forming the differential stiffness for buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.

FORMAT:

$$\text{STATSUB}\left[\left(\begin{matrix}\text{BUCKLING}\\ \text{PRELOAD}\end{matrix}\right)\right] = \text{n}$$

EXAMPLES:

```
STATSUB=23
STAT=4
STATSUB(PREL)=7
```

DESCRIBERS:

Describer	Meaning
BUCKLING	Subcase id number corresponding to static subcase of buckling or varying load. (Default in buckling analysis.)
PRELOAD	Subcase id number corresponding to static subcase of preload or constant load. (Default in dynamic analysis.)
n	Subcase identification number of a prior SUBCASE specified for static analysis (Integer>0)

REMARKS:

1. STATSUB may be used in SOLs 103, 105, 107 through 112, and SOL 200 (ANALYSIS = BUCKLING) only.

2. STATSUB must be specified in the same subcase that contains the METHOD selection for buckling or normal modes, CMETHOD for complex eigenvalue analysis, TSTEP for transient response and FREQ for frequency response.

3. In SOL 105, if it is intended that results from the first static subcase are used to compute the differential stiffness, then the STATSUB command is not required. In other words, the default for STATSUB is the first static subcase identification. In SOLs 103 and 107 through 112, 115, and 116, STATSUB must reference a separate static subcase.

4. In dynamic analysis, only one STATSUB command may be specified in each dynamic subcase. In buckling analysis with a pre-load, both STATSUB (BUCKLING) and STATSUB(PRELOAD) must be specified in each buckling subcase.

5. If superelements are used , the static subcase which is identified in the STATSUB command must include the residual superelement or the solution will fail with an error.

6. If multiple superelements are used with the STATSUB command, each must be used in a static subcase or their differential stiffness will not be included. All superelements can be used in a single static subcase using SUPER=ALL, or a separate subcase can exist for each one (order of subcases is not important). In either case, the STATSUB command should point to the subcase which references the residual (see above remark).

STATVAR

Requests output of state variables with SOL 401.

Requests output of state variables used by a user material subroutine with SOL 401.

FORMAT:

$$\text{STATVAR}\left[\left(\begin{bmatrix}\text{GRID}\\ \text{GAUSS}\\ \text{BOTH}\end{bmatrix}, \text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
STATVAR=ALL
STATVAR(PRINT,PUNCH)=17
```

DESCRIBERS:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
GRID	Output at grid points on elements.
GAUSS	Output at gauss points on elements.
BOTH	Output at both grid and gauss points on elements.
ALL	Requests output for all elements.
n	Set identification number of a previously appearing SET command. The SET command lists element IDs. (Integer>0)
NONE	No output requested.

REMARKS:

1. Only supported in SOL 401.

STRAIN

Element Strain Output Request

Requests the form and type of strain output.

FORMAT:

$$\text{STRAIN}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{VONMISES}\\ \text{MAXS or SHEAR}\end{bmatrix},\right.\right.$$

$$\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix},\begin{bmatrix}\text{RPUNCH}\end{bmatrix},$$

$$\left.\left.\begin{bmatrix}\text{CPLYMID}\\ \text{CPLYBT}\\ \text{CPLYBMT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
STRAIN=5
STRAIN(CORNER)=ALL
STRAIN(PRINT,PHASE)=15
STRAIN(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.

Describer	Meaning
SORT2	Output will be presented as a tabular listing of frequency or time for each element.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates strain for the requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 10.

Describer	Meaning
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 10.
RPUNCH	Writes random analysis results to the punch file. See Remark 10.
VONMISES	von Mises strain is output.
MAXS or SHEAR	Maximum shear strains are output.
STRCUR	Strain at the reference plane and curvatures is output for plate elements.
FIBER	Strain at locations Z1, Z2 is computed for plate elements.
CENTER	Outputs strains at the center of the element only. See Remark 16.
CORNER or BILIN	Outputs strains at the center and corner grid points using extrapolation. See Remark 17.
SGAGE	Outputs strains at the center and corner grid points using strain gage approach. See Remark 18.
CUBIC	Outputs strains at the center and corner grid points using cubic bending correction. See Remark 19.
CPLYMID	Requests element strains at the middle of each ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15.
CPLYBT	Requests element strains at the bottom and top of the ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15.
CPLYBMT	Requests element strains at the bottom, middle, and top of each ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15
ALL	Output strain for all elements.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	No element strain will be output.

REMARKS:

1. In SOLs 106 and 129, nonlinear strains for nonlinear elements are requested by the STRESS/NLSTRESS commands and appear in the nonlinear stress output. The STRAIN command will generate additional output for total strain except for hyperelastic elements. The additional STRAIN output request will also be ignored for nonlinear material elements when the parameter LGDISP is -1, which is the default (strains will appear in the nonlinear stress output).

2. Both PRINT and PUNCH may be requested.

3. STRAIN=NONE overrides an overall output request.

4. The PLOT option is used when strains are requested for postprocessing but no printed output is desired.

5. Definitions of stress, strain, curvature, and output locations are given in the *NX Nastran Element Library Reference Manual*.

6. If the STRCUR option is selected, the values of Z1 will be set to 0.0 and Z2 will be set to -1.0 on the output.

7. The VONMISES, MAXS, and SHEAR describers are ignored in the complex eigenvalue and frequency response solution sequences. Although, von Mises stress and strain are computed by default in the frequency response solutions 108 and 111 when stress and strain results are requested. The system cell setting NASTRAN SYSTEM(579)=1 can be defined to disable the von Mises stress and strain request in SOLs 108 and 111.

8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also the FORCE command for further discussion. These options

are discussed in “Understanding Plate and Shell Element Output” in the *NX Nastran Element Library Reference*.

9. The defaults for SORT1 and SORT2 depend on the type of analysis:
   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.
   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.
   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.
10. The following applies to random solutions:
   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.
   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.
   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.
11. If some combination of the CPLYMID, CPLYBT, or CPLYBMT describers are specified, the describer producing the most output data is used.
12. If different CPLYMID, CPLYBT, or CPLYBMT describers are specified on STRESS and STRAIN case control commands in the same input file, the describer specified on the STRESS case control command takes precedence.
13. Failure indices and strength ratios for in-plane ply failure are output for the locations corresponding to the CPLYMID/CPLYBT/CPLYBMT specification and the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification. Failure indices

and strength ratios for inter-laminar failure are always output at the top and bottom of each ply at the locations corresponding to the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification.

14. A PSHELL entry with a MID1 or MID2 greater than or equal to 10^8 requires the parameter NOCOMPS to be 0 or -1 for any stress or strain recovery. See the parameter NOCOMPS for more information.

15. For elements referencing PCOMPS property entries, the REAL, IMAG, PHASE, VONMISES, MAXS, SHEAR, STRCUR, FIBER, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, NOPRINT, and RPUNCH describers are not supported.

16. For the CENTER option, strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center of the element for each ply.

17. For the CORNER (or BILIN) option, strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center and corner grid locations for each ply.

18. For the SGAGE option, strains are output at the center and grid points for CQUAD4 elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

19. For the CUBIC option, strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

20. Shear strain is output as engineering shear strain, which is twice the tensor shear strain.

STRESS

Element Stress Output Request

Requests the form and type of element stress output.

FORMAT:

$$\text{STRESS}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{VONMISES}\\ \text{MAXS or SHEAR}\end{bmatrix},\right.\right.$$

$$\left.\left.\begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix},\begin{bmatrix}\text{RPUNCH}\end{bmatrix},\begin{bmatrix}\text{CPLYMID}\\ \text{CPLYBT}\\ \text{CPLYBMT}\end{bmatrix}\right)\right]$$

$$= \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
STRESS=5
STRESS(CORNER)=ALL
STRESS  (SORT1,PRINT,PUNCH,PHASE)=15
STRESS(PLOT)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.

Describer	Meaning
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates stresses for requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 12.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 12.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 12.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 12.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 12.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 12.

Describer	Meaning
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 12.
RPUNCH	Writes random analysis results to the punch file. See Remark 12.
VONMISES	von Mises stress is output.
MAXS or SHEAR	Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.
CENTER	Outputs stresses at the center of the element only. See Remark 18.
CORNER or BILIN	Outputs stresses at the center and corner grid points using extrapolation. See Remark 19.
SGAGE	Outputs stresses at center and corner grid points using strain gage approach. See Remark 20.
CUBIC	Outputs stresses at the center and corner grid points using cubic bending correction. See Remark 21.
CPLYMID	Requests element stresses at the middle of each ply for elements referencing PCOMPS property entries. See Remark 13, Remark 14, Remark 15, and Remark 17.
CPLYBT	Requests element stresses at the bottom and top of the ply for elements referencing PCOMPS property entries. See Remark 13, Remark 14, Remark 15, and Remark 17.
CPLYBMT	Requests element stresses at the bottom, middle, and top of each ply for elements referencing PCOMPS property entries. See Remark 13, Remark 14, Remark 15, and Remark 17.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output. (Integer >0)
NONE	No element stress will be output.

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. ALL should be used with caution in transient solutions as it can produce a large amount of output.

3. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

4. STRESS=NONE overrides an overall output request.

5. The PLOT option is used when contour plots of stresses are requested but no printed output of stresses is desired. However in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.

6. The VONMISES option is ignored for ply stresses.

7. The VONMISES, MAXS, and SHEAR describers are ignored in the complex eigenvalue and frequency response solution sequences. Although, von Mises stress and strain are computed by default in the frequency response solutions 108 and 111 when stress and strain results are requested. The system cell setting NASTRAN SYSTEM(579)=1 can be defined to disable the von Mises stress and strain request in SOLs 108 and 111.

8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be

ignored. See also the FORCE command for further discussion. These options are discussed in the section “Understanding Plate and Shell Element Output” in Chapter 4 of the *NX Nastran Element Library Reference*.

9. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).

10. For nonlinear analysis, the grid point option for CQUAD4 elements will be reset to the default (CENTER) option for nonlinear elements.

11. MAXS for shell elements is not an equivalent stress.

12. The following applies to random solutions:

    - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

    - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

    - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

13. If some combination of the CPLYMID, CPLYBT, or CPLYBMT describers are specified, the describer producing the most output data is used.

14. If different CPLYMID, CPLYBT, or CPLYBMT describers are specified on STRESS and STRAIN case control commands in the same input file, the describer specified on the STRESS case control command takes precedence.

15. Failure indices and strength ratios for in-plane ply failure are output for the locations corresponding to the CPLYMID/CPLYBT/CPLYBMT specification and the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification. Failure indices and strength ratios for inter-laminar failure are always output at the top and bottom of each ply at the locations corresponding to the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification.

16. A PSHELL entry with a MID1 or MID2 greater than or equal to 10^8 requires the parameter NOCOMPS to be 0 or -1 for any stress or strain recovery. See the parameter NOCOMPS for more information.

17. For elements referencing PCOMPS property entries, the REAL, IMAG, PHASE, VONMISES, MAXS, SHEAR, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, NOPRINT, and RPUNCH describers are not supported.

18. For the CENTER option, stresses are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that are referenced by a PSHELL entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply stresses are always reported at the center of the element. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center of the element. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply stresses are output at the center of the element for each ply.

19. For the CORNER (or BILIN) option, stresses are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply stresses are output at the center and corner grid locations for each ply. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points.

20. For the SGAGE option, stresses are output at the center and grid points for CQUAD4 elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points of the element.

21. For the CUBIC option, stresses are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points of the element.

22. For the output coordinate system of CQUAD4 and CTRIA3 elements, see CQUAD4 and CTRIA3. For the output coordinate system of CHEXA, CPENTA, CPYRAM, and CTETRA elements, see PSOLID and PLSOLID.

23. In SOL 401, for elements in which plasticity is defined, the von Mises stress defines the yield surface. When kinematic or combined hardening is used and there is loading and unloading which causes the center of the yield surface to translate, the von Mises stress output is computed relative to the new yield surface origin. The stress components are computed from the original yield surface origin. In this case, the von Mises stress computed in a post processor using the component stresses may not match the von MIses stress reported by NX Nastran in the .f06 file.

REMARKS RELATED TO SOLS 601 AND 701:

1. Output is restricted to REAL format and CENTER or CORNER for CQUAD4 element stresses. IMAG, PHASE, CUBIC, SGAGE, PSDF, ATOC, RMS, and RALL are ignored.

2. The same request is used for element strain output in both linear and nonlinear analysis, i.e., the STRAIN command is ignored for linear analysis.

3. For both linear and nonlinear analysis, stress and strain results at grid points may be requested for single-ply elements by specifying the CORNER option.

4. If there is any nonlinearity in the model, i.e., large displacements, contact, or nonlinear material models, elements results are output in the op2 file using nonlinear element stress/strain data block, including elements with linear material models. Exceptions are the CQUAD8 and CTRIA6 elements, where both linear and nonlinear element stress/strain data blocks are output. However, in a future release, the output of linear stress/strain data blocks for CQUAD8 and CTRIA6 elements may be discontinued for a nonlinear analysis.

STRFIELD

Grid Point Stress Output Request

Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.

FORMAT:

$$\text{STRFIELD} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

EXAMPLES:

```
STRFIELD=ALL
STRFIELD=21
```

DESCRIBERS:

Describer	Meaning
ALL	Grid point stress requests for all surfaces and volumes defined in the OUTPUT(POST) section will be saved for postprocessing.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command and in the OUTPUT(POST) section will be included in the grid point stress output request for postprocessing. (Integer>0)

REMARKS:

1. The STRFIELD command is required for the graphical display of grid point stresses in postprocessors that use the .xdb file (PARAM,POST,0), or when the GPSDCON or ELSDCON commands are specified. The GPSTRESS command can be used to obtain printed output.

2. Only grid points connected to elements used to define the surface or volume are output. See the SURFACE and VOLUME commands.

3. Element stress output (STRESS) must be requested for elements referenced on requested SURFACE and VOLUME commands.

4. In nonlinear static and transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default. Also, in nonlinear transient analysis, grid point stresses are computed for elements with linear material properties only.

SUBCASE

Subcase Delimiter

Delimits and identifies a subcase.

FORMAT:

SUBCASE=n

EXAMPLES:

```
SUBCASE=101
```

DESCRIBERS:

Describer	Meaning
n	Subcase identification number. (Integer>0)

REMARKS:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot requests and RANDPS requests refer to n.
3. See the MODES command for use of this command in normal modes analysis.
4. If a comment follows n, then the first few characters of the comment will appear in the subcase label in the upper right-hand corner of the output.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 601,106, SOL 601,129 and SOL 701, only one subcase is used. If more than one subcase is specified, the first subcase is used.

2. For SOL 601,153 and SOL 601,159, two subcases are required. The first two subcases must be one with ANALYSIS=STRUC (default) and one with ANALYSIS=HEAT. The parameter COUP in TMCPARA bulk entry is used to specify the type of coupling between the structural and heat transfer analysis.

SUBCOM

Combination Subcase Delimiter

Delimits and identifies a combination subcase.

FORMAT:

SUBCOM = n

EXAMPLES:

```
SUBCOM = 125
```

DESCRIBERS:

Describer	Meaning
n	SUBCOM identification number. (Integer>2)

REMARKS:

1. The SUBCOM identification number, n, must be greater than all previous subcase and SUBCOM identification numbers.
2. A SUBSEQ command must follow this command.
3. SUBCOM may only be used in statics problems.
4. Output requests above the subcase level will be used.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SUBCOM with a TEMP(LOAD) command or the element deformations with a DEFORM command.
6. SUBCOMs may be specified in superelement analysis with the following recommendations:

- For each superelement, specify its SUBCASEs consecutively, directly followed by its SUBCOM(s).
- Specify a SUPER command with a new load sequence number under each SUBCOM command.

The following example is given for a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SUBCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SUBSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SUBCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SUBSEQ=1.,1.
```

SUBSEQ

Subcase Sequence Coefficients

Gives the coefficients for forming a linear combination of the previous subcases.

FORMAT:

SUBSEQ=R1 [, R2, R3, ..., Rn]

EXAMPLES:

```
SUBSEQ=1.0, -1 .0, 0.0, 2.0
```

DESCRIBERS:

Describer	Meaning
Ri	Coefficients of the previously occurring subcases. See Remark 4 . (Real)

REMARKS:

1. The SUBSEQ command can only appear after a SUBCOM command.
2. This command may only be used in statics problems.
3. This command list is limited to a maximum of 200 numbers.
4. R1 to Rn refer to the immediately preceding subcases. In other words Rn is applied to the most recently appearing subcase and R(n - 1) is applied to the second most recently appearing subcase, and so on. The comments ($) describe the following example:

```
DISPL=ALL
SUBCASE 1
SUBCASE 2
SUBCOM 3
SUBSEQ=1.0, -1.0 $ SUBCASE 1 - SUBCASE 2
```

```
SUBCASE 11
SUBCASE 12
SUBCOM 13
SUBSEQ=0.0, 0.0, 1.0, -1 .0 $ SUBCASE 11 - SUBCASE 12
or
SUBSEQ=1.0, - 1.0 $ EQUIVALENT TO PRECEDING COMMAND.
USE ONLY ONE.
```

SUBTITLE

Output Subtitle

Defines a subtitle that will appear on the second heading line of each page of printer output.

FORMAT:

SUBTITLE=subtitle

EXAMPLES:

```
SUBTITLE=PROBLEM NO. 5-1A
```

DESCRIBERS:

Describer	Meaning
subtitle	Any character string.

REMARKS:

1. SUBTITLE appearing under a SUBCASE command will appear in the output for that subcase only.
2. SUBTITLE appearing before all SUBCASE commands will appear in the output for all subcases except those in Remark 1 .
3. If no SUBTITLE command is present, the subtitle line will be blank.
4. The subtitle also appears on plotter output.

SUPER

Superelement Subcase Assignment

Assigns a subcase(s) to a superelement or set of superelements.

FORMAT:

$$\text{SUPER} = \left\{ \begin{array}{c} \text{ALL} \\ \left\{ \begin{array}{c} \text{n} \\ \text{i} \end{array} \right\} [\, , l \,] \end{array} \right\}$$

EXAMPLES:

```
SUPER=17, 3
SUPER=15
SUPER=ALL
```

DESCRIBERS:

Describer	Meaning
i	Superelement identification number. (Integer≥0)
ALL	The subcase is assigned to all superelements and all loading conditions. (Default)
n	Set identification number of a previously appearing SET command. The subcase is assigned to all superelements with identification numbers that appear on this SET command. (Integer>0)
l	Load sequence number. (Integer>0; Default = 1)

REMARKS:

1. All subcases with requests for specific superelement(s) must contain the SUPER command. If no SUPER command is specified in the Case Control Section then all subcases will be assigned to all superelements. In other words, SUPER=ALL is the default.

2. All subcases associated with superelements must precede those for the residual structure except when SUPER=ALL or SUPER=n and the selected set includes the residual structure.

3. The load sequence number is only used in static analysis and frequency response analysis when there are multiple loading conditions. Also, the residual structure must have a subcase specified for each unique load condition. This is required because the number of residual structure subcases is used to determine the number of load conditions for all superelements.

4. The load sequence number is associated with the order of the subcases for the residual structure; i.e., the third loading condition is associated with the third subcase for the residual structure.

5. Subcases are required for superelements when there is a load, constraint, or output request.

6. If a set is referenced by n, then the SET identification number must be unique with respect to any superelement identification numbers. In addition, the same sets must be used for all loading conditions.

7. If the ALL option is used, it must be used for all loading conditions.

8. When i=0, the residual structure is used instead of a superelement. See the *NX Nastran Superelement User's Guide* for a complete description of this option.

SUPORT1

Fictitious Support Set Selection

Selects the fictitious support set (SUPORT1 entries only) to be applied to the model.

FORMAT:

SUPORT1=n

EXAMPLES:

```
SUPORT1=15
SUPO=4
```

DESCRIBERS:

Describer	Meaning
n	Set identification of fictitious support set defined on the SUPORT1 Bulk Data entry. (Integer>0)

REMARKS:

1. SUPORT1 entries will not be used unless selected in Case Control by the SUPORT1 command.
2. SUPORT entries will be applied in all subcases.

SURFACE

Surface Definition

Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.

FORMAT:

$$\text{SURFACE id SET sid,}\left[\text{FIBRE}\left\{\begin{matrix}\text{ALL}\\ \text{Z1}\\ \text{Z2}\\ \text{MID}\end{matrix}\right\}\right],$$

$$\left[\text{SYSTEM}\left\{\begin{matrix}\text{ELEMENT}\\ \text{BASIC}\\ \text{CORD cid}\end{matrix}\right\}\right],\left[\text{AXIS}\left\{\begin{matrix}X1\\ \text{X2}\\ \text{X3}\end{matrix}\right\}\right],$$

$$\left[\text{NORMAL}\ [\text{M}]\left\{\begin{matrix}\text{R}\\ \text{X1}\\ \text{X2}\\ \text{X3}\end{matrix}\right\}\right],\left[\begin{matrix}\text{TOPOLOGICAL}\\ \text{GEOMETRIC}\end{matrix}\right][\text{TOLERANCE}\{\text{THETA}\}],$$

$$\left[\text{BRANCH}\left[\left[\left\{\begin{matrix}\text{MESSAGE}\\ \text{NOMESSAGE}\end{matrix}\right\}\right],\left[\begin{matrix}\text{BREAK}\\ \text{NOBREAK}\end{matrix}\right]\right]\right]$$

EXAMPLES:

```
SURFACE 10 SET 9 NORMAL X3
```

DESCRIBERS:

Describer	Meaning
id	Surface identification number (required).
SET	References a SET command that defines the elements in the surface (required). Either form of the SET command may be used.
sid	Set identification number.
FIBRE	Specifies the fiber location at which stresses will be calculated.
ALL	Requests output at all fiber locations; i.e., z=Z1, Z2, and MID.
Z1	Requests output z=Z1 only.
Z2	Requests output z=Z2 only.
MID	Requests output z=(Z1+Z2)/2 only.
SYSTEM	Specifies the coordinate system to be used to define the output coordinate system.
ELEMENT	Specifies the element coordinate system for output.
CORD cid	Specifies the coordinate system defined on a CORDij Bulk Data entry for output.
BASIC	Specifies the basic coordinate system for output.
AXIS	Specifies the axis of the coordinate system to be used as the x output axis and the local x-axis when geometric interpolation is used.
X1, X2, X3	Specifies the direction of the axis or the normal. X, Y, and Z may be substituted for X1, X2, and X3, respectively.
NORMAL	Specifies the reference direction for positive fiber and shear stress output, but has no effect when ELEMENT is specified.
M	Specifies the reverse of the direction given by R, X1, X2, or X3.

Describer	Meaning
R	Specifies the radius vector from the origin of reference coordinate system to the grid point.
TOPOLOGICAL GEOMETRIC	Specifies the method to calculate the average grid point stress or strain. The default is TOPOLOGICAL.
theta	Specifies the interelement slope difference tolerance (in degrees) to detect stress discontinuity between elements, (not used with TOPOLOGICAL). (Real; Default = 0.0)
BRANCH	Selects whether multiple element intersections (BREAK/NOBREAK) are to be treated as discontinuities and if warning messages (MESSAGE/NOMESSAGE) are to be issued.
BREAK NOBREAK	Multiple element intersections are (or are not) to be treated as stress discontinuities.
MESSAGE NOMESSAGE	A warning message will (or will not) be issued when multiple element intersections are encountered.

REMARKS:

1. SURFACE commands must be specified after OUTPUT(POST).

2. The surface identification number must be referenced on a SET command, specifically the SET command which comes after OUTPUT(POST). This SET identification number may then be referenced on the GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands.

3. The surface normal is also used in the definition of the local reference surface for geometric interpolation. Two options are available. In the first option, the radius vector (R) from the origin of the reference coordinate system to the grid point is used. In the second option, one axis (X1, X2, or X3) of the coordinate system is used. The direction can be reversed using the modification parameter, M. The positive side of an element is defined as a side from which the NORMAL direction emerges rather than the side determined by the connection specified on the element connection entries.

4. When the parameter ELEMENT is present, the element stresses or strains are used unmodified (defaults to output stresses in the output system). The

CORD keyword references a CORDij Bulk Data entry with coordinate system identification number cid.

5. When theta=0, no testing is made. When theta is negative, grid point stresses will be calculated for each element connected to an exception point; otherwise, the best estimation of the grid point stress will be output.

6. BREAK is the default if theta is nonzero.

7. For all elements defined in SET 9 in the Examples section:

   - All fiber locations are output.
   - The basic output system is used.
   - The x-axis is x-axis of the basic system.
   - The surface normal direction point is z-axis of the basic system.
   - The topological interpolation method is used.
   - No tolerance test is made.
   - No branch test is made.

   The example illustrates a good choice for regular two-dimensional problems in the x-y plane.

SVECTOR

Solution Set Eigenvector Output Request

Requests the form and type of solution set eigenvector output.

FORMAT:

$$\text{SVECTOR}[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

EXAMPLES:

```
SVECTOR=ALL
SVECTOR(PUNCH)=NONE
```

DESCRIBERS:

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. SVECTOR=NONE overrides an overall output request.

3. Output will be presented as a tabular listing of grid points for each eigenvector.

4. SVECTOR can be useful, for example, if you would like to output eigenvectors for all grids in the OP2 file, yet only print the eigenvectors for a specific grid set to the f06 file. In this example, both DISPLACEMENT(PLOT) = ALL and SVECTOR(PRINT) = 101 would be included in the case control.

SVELOCITY

Solution Set Velocity Output Request

Requests the form and type of solution set velocity output.

FORMAT:

$$\text{SVELOCITY}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \text{PRINT}, \text{PUNCH}\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
SVELOCITY=5
SVELOCITY(SORT2,PUNCH,PRINT,PHASE)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
ALL	Velocity for all solution points (modes) will be output.
NONE	Velocity for no solution points (modes) will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.
2. Velocity output is only available for transient and frequency response problems.
3. The defaults for SORT1 and SORT2 depend on the type of analysis and are discussed in DISPLACEMENT. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
4. SVELOCITY=NONE overrides an overall output request.
5. Due to the differences in formulations, the modal responses for the constraint mode method of enforced motion will be different than the modal responses for the absolute displacement method of enforced motion. In order to highlight this difference, the f06 output for the constraint mode method will be labeled *alternative* solution set.

SYM

Symmetry Subcase Delimiter

Delimits and identifies a symmetry subcase.

FORMAT:

SYM=n

EXAMPLES:

```
SYM=123
```

DESCRIBERS:

Describer	Meaning
n	Subcase identification number. (Integer>0)

REMARKS:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot commands should refer to n.
3. Overall output commands will not propagate into a SYM subcase (i.e., any output desired must be requested within the subcase).
4. SYM may only be used in statics or inertia relief problems.

SYMCOM

Symmetry Combination Subcase Delimiter

Delimits and identifies a symmetry combination subcase.

FORMAT:

SYMCOM=n

EXAMPLES:

```
SYMCOM=123
```

DESCRIBERS:

Describer	Meaning
n	Subcase identification number. (Integer>2)

REMARKS:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. SYMCOM may only be used in statics problems.
3. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SYMCOM by use of a TEMP(LOAD) command, or the element deformations by a DEFORM command.
4. An alternate command is the SUBCOM command.
5. SYMCOMs may be specified in superelement analysis with the following recommendations:
   - For each superelement, specify its SUBCASEs consecutively, directly followed by its SYMCOM(s).

- Specify a SUPER command with a new load sequence number under each SYMCOM command.

  The following example is given for a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SYMCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SYMSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SYMCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SYMSEQ=1.,1.
```

SYMSEQ

Symmetry Sequence Coefficients

Gives the coefficients for combining the symmetry subcases into the total structure.

FORMAT:

SYMSEQ=R1 [,R2,R3,..., Rn]

EXAMPLES:

```
SYMSEQ=1.0, -2.0, 3.0, 4.0
```

DESCRIBERS:

Describer	Meaning
Ri	Coefficients of the previously occurring n SYM subcases. (Real)

REMARKS:

1. SYMSEQ may only appear after a SYMCOM command.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. SYMSEQ may only be used in statics or inertia relief.
4. Ri is limited to a maximum of 200 numbers.

TEMPERATURE

Temperature Set Selection

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

FORMAT:

$$\text{TEMPERATURE}\left[\begin{pmatrix}\text{INITIAL}\\ \text{MATERIAL}\\ \text{LOAD}\\ \text{BOTH}\end{pmatrix}\right] = n$$

EXAMPLES:

```
TEMPERATURE(LOAD)=15
TEMPERATURE(MATERIAL)=7
TEMPERATURE=7
```

DESCRIBERS:

Describer	Meaning
MATERIAL	The selected temperature set will be used to determine temperature-dependent material properties indicated on the MATTi Bulk Data entries. See Remarks 6 and 7.
LOAD	The selected temperature set will be used to determine an equivalent static load and to update material properties in a nonlinear analysis. See Remarks 6 and 7.
BOTH	Both MATERIAL and LOAD will use the same temperature set.
n	Set identification number of TEMP, TEMPD, TEMPP1, TEMPRB, or TEMPEX bulk entries. (Integer>0)

Describer	Meaning
INITIAL	The selected temperature set will be used to determine initial temperature distribution. See Remarks 6 and 7.

REMARKS:

1. In a linear, non-superelement analysis, only one temperature-dependent material request can be made, and should be specified above the subcase level. A fatal error will occur if multiple temperature-dependent material requests are made. See also Remarks 6 and 7.

   In a superelement analysis, each superelement subcase can have a different TEMPERATURE(MATERIAL), but consecutive load cases must have consistent TEMPERATURE(MATERIAL).

2. The total load applied will be the sum of the external (LOAD command), thermal (TEMP(LOAD) command), element deformation (DEFORM command) and constrained displacement (SPC command) loads.

3. Static, thermal, and element deformation loads should have unique set identification numbers.

4. INITIAL is used in steady state heat transfer analysis for conduction material properties and provides starting values for iteration.

5. In superelement data recovery restarts, TEMPERATURE(LOAD) requests must be respecified in the Case Control Section.

6. TEMPERATURE(MATERIAL) and TEMPERATURE(INITIAL) cannot be specified simultaneously in the same run.

7. TEMPERATURE(BOTH) cannot be specified simultaneously with TEMPERATURE(LOAD) or TEMPERATURE(MATERIAL).

8. The TEMPERATURE case control inputs for thermal strain and temperature dependent material properties in linear solutions are described as follows.

   Thermal strain in linear solutions is calculated from

   $\varepsilon = \alpha(T_2 - T_1)$

   where T_2=TEMPERATURE(LOAD) and T_1=TEMPERATURE(INITIAL) if it is defined, otherwise T_1=TREF. TREF is defined on material bulk entries (MAT1, MAT2, ..., MATi) or the PCOMP and PCOMPG bulk entries and defaults to "0.0".

Linear Solution Input Combinations

- If TEMPERATURE(LOAD) and TEMPERATURE(INITIAL) are defined, T_1=TEMPERATURE(INITIAL), and MATTi temperature dependent material properties are evaluated at TEMPERATURE(INITIAL).

  $\varepsilon = \alpha_{init}(T_{load} - T_{init})$

- If TEMPERATURE(LOAD) and TEMPERATURE(MATERIAL) are defined, T_1=TREF, and MATTi temperature dependent material properties are evaluated at TEMPERATURE(MATERIAL).

  $\varepsilon = \alpha_{mat}(T_{load} - T_{ref})$

- If TEMPERATURE(BOTH) is defined, T_1=TREF, and MATTi temperature dependent material properties are evaluated at TEMPERATURE(BOTH).

  $\varepsilon = \alpha_{mat}(T_{load} - T_{ref})$

- If TEMPERATURE(LOAD) is defined and neither of TEMPERATURE(INITIAL) or TEMPERATURE(MATERIAL) are defined, T_1=TREF, and MATTi temperature dependent material properties are ignored. Material properties are obtained from the MATi entries.

  $\varepsilon = \alpha(T_{load} - T_{ref})$
  where α on the MATi entry is used.

- If TEMPERATURE(LOAD) is not defined, material properties are evaluated at TEMPERATURE(MATERIAL) or TEMPERATURE(INITIAL) if either is defined, and no thermal strains result for that subcase.

- If none of TEMPERATURE(LOAD), TEMPERATURE(INITIAL) or TEMPERATURE(MATERIAL) are defined, no thermal strains result for that subcase, and MATTi temperature dependent material properties are ignored. Material properties are obtained from the MATi entries.

9. The TEMPERATURE case control inputs for thermal strain and temperature dependent material properties in solutions 106, 401, 601 and 701 are described as follows.

   Thermal strain in the nonlinear solutions is calculated from

   $\varepsilon = \alpha_{load}(T_{load} - T_{ref}) - \alpha_{init}(T_{init} - T_{ref})$

   where α_{load} is evaluated at TEMPERATURE(LOAD) and α_{init} is evaluated at TEMPERATURE(INIT).

Input Combinations for Nonlinear Solutions 106, 401, 601 and 701

- For SOL 106, TEMPERATURE(INITIAL) is required when TEMPERATURE(LOAD) is defined, and should be above the subcase level. See SOLs 601 and 701 Remark 5.
- For solutions 106, 401, 601 and 701, the specification of TEMPERATURE(MATERIAL) or TEMPERATURE(BOTH) will cause a fatal error.
- For solutions 106, 401, 601 and 701, TEMPERATURE(LOAD) is used to update temperature-dependent material properties. Temperature-dependent material properties are specified with MATi, MATTi, MATS1, and/or TABLEST bulk entries.
- For solutions 106, 401, 601 and 701, if TEMPERATURE(INITIAL) is defined but TEMPERATURE(LOAD) is not defined, MATTi temperature dependent material properties are evaluated at TEMPERATURE(INITIAL), and no thermal strains result for that subcase.
- For solutions 106 and 401, if neither TEMPERATURE(LOAD) or TEMPERATURE(INITIAL) are defined, MATTi temperature dependent material properties are ignored.
- For solutions 601 and 701, if neither TEMPERATURE(LOAD) or TEMPERATURE(INITIAL) are defined, MATTi temperature dependent material properties are evaluated at "0.0".
- For solution 401, the TVAR parameter on the NLCNTL bulk entry controls if the temperature loads are ramped, or applied as a constant for each subcase.
- For solution 401, the thermal strain computation can be turned off by defining the parameter setting THRMST=NO (default=YES) on the NLCNTL bulk entry. This is useful for temperature dependent material evaluation without thermal loading.

10. For solutions 103, 110, 111 and 112, TEMPERATURE(LOAD) should not be used. If TEMPERATURE(LOAD) is used in these solutions, thermal strain is incorrectly subtracted from total strain. This results in the calculation of incorrect modal stresses.

11. The TEMPEX bulk entry can only be selected with the TEMP (LOAD) and TEMP (INIT) in SOL 401.

12. For all solutions except solution 401, if multiple TEMPERATURE(INITIAL) case control commands are present, NX Nastran applies the TEMPERATURE(INITIAL) *used* in the last subcase to all of the subcases. The following examples demonstrate.

- In the following example, since TEMP(INIT)=400 is used in the last subcase (subcase 3), the software applies TEMP(INIT)=400 to all three subcases.

```
.....
   TEMP(INIT)=100
Subcase 1
   TEMP(INIT)=200
Subcase 2
   TEMP(INIT)=300
Subcase 3
   TEMP(INIT)=400
.....
```

- In the following example, since TEMP(INIT) is undefined in subcase 3, subcase 3 uses the globally defined TEMP(INIT)=100. As a result, the software applies TEMP(INIT)=100 to all three subcases.

```
.....
   TEMP(INIT)=100
Subcase 1
   TEMP(INIT)=200
Subcase 2
   TEMP(INIT)=300
Subcase 3
.....
```

- In the following example, since TEMP(INIT) is undefined in subcase 3 and TEMP(INIT) is not defined globally, subcase 3 has no initial temperature to use. As a result, the software applies no initial temperature to any of the subcases.

```
.....
Subcase 1
   TEMP(INIT)=200
Subcase 2
   TEMP(INIT)=300
Subcase 3
.....
```


REMARKS RELATED TO SOLS 601 AND 701:

1. See Remark 9 above.
2. n should reference the set identification number of the TEMP or TEMPD Bulk Data entry only.
3. Temperature loads can be used in both static and transient analysis.

4. TEMPERATURE(LOAD) is used to select a constant temperature load. For a time-dependent temperature load, the DTEMP case control command is used instead. A model can have temperature loading defined either by TEMPERATURE(LOAD) or DTEMP, but not both.

5. If TEMPERATURE(INITIAL) is not specified, then the initial temperatures are assumed to be “0.0”.

6. If TEMPERATURE(INITIAL) is defined but neither TEMPERATURE(LOAD) or a time-dependent temperature load using DLOAD are defined, TEMPERATURE(LOAD) will default to TEMPERATURE(INITIAL). As a result, thermal strain is zero, and the automatic time stepping scheme (ATS) does not increase the time step size. See the parameter ATSMXDT on the NXSTRAT bulk entry.

TFL

Transfer Function Set Selection

Selects the transfer function set(s) to be added to the direct input matrices.

FORMATS:

TFL={n_1, [n_2, n_3 THRU n_4, EXCEPT n_5, n_6, n_7]}

EXAMPLES:

```
TFL=77
TFL=1, 25, 77
TFL=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79
TFL=1 THRU 10
```

DESCRIBERS:

Describer	Meaning
n_1, n_2, etc.	Identification numbers of TF Bulk Data entries. (Integer>0)
n_3 THRU n_4	Identification numbers of TF bulk data entries (n_3>n_4). (Integer>0)
EXCEPT	Identification numbers following EXCEPT will be deleted from the list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU.

REMARKS:

1. Transfer functions will not be used unless selected in the Case Control Section.
2. Transfer functions are supported in dynamics problems only.
3. Transfer functions are described in the *NX Nastran Advanced Dynamic Analysis User's Guide*.

4. It is recommended that PARAM,AUTOSPC,NO be specified when using transfer functions. See the *NX Nastran User's Guide*.

5. The transfer functions are additive if multiple TF values are referenced on the TFL command.

6. Identification numbers following EXCEPT within the range of the THRU must be in ascending order.

7. In the third example in the Examples section, the numbers 77, 78, 79 are included because they are outside the prior THRU range.

8. The TFL command is supported across subcases. A TFL command selecting different TF bulk data entries can be defined for each subcase.

THERMAL

Temperature Output Request

Requests the form and type of temperature output.

FORMAT:

$$\text{THERMAL}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
THERMAL=5
THER(PRINT,PUNCH)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output is presented as a tabular listing of point temperatures for each load or time step.
SORT2	Output is presented as a tabular listing of loads or time steps for each point temperature.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Compute temperatures but do not print.
ALL	Temperatures for all points will be output.
NONE	Temperatures for no points will be output.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only temperatures of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. The THERMAL output request is designed for use with the heat transfer option. The printed output will have temperature headings. The PUNCH option produces TEMP Bulk Data entries, and the SID on the entries will be the subcase number (=1 if no SUBCASES are specified).

2. SORT1 is the default in steady state heat transfer analysis. SORT2 is the default in transient heat transfer analysis.

3. In a transient heat transfer analysis, the SID on the punched TEMP Bulk Data entries, equals the time step number.

REMARKS RELATED TO SOL 601:

1. THERMAL output request is supported for heat transfer analysis in SOL 601,153 and SOL 601,159.

THSTRN

Requests thermal strain output at grid points on elements for SOL 401. Also requests thermal strain output at grid points on composite shell elements for SOLs 101 and 106.

Requests thermal strain at grid points on elements.

FORMAT:

$$\text{THSTRN}\left[\left(\left[\begin{matrix}\text{STRCUR}\\ \text{FIBER}\end{matrix}\right],\left[\begin{matrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{matrix}\right],\ \text{PRINT, PUNCH, PLOT}\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

EXAMPLES:

```
THSTRN=ALL
THSTRN(PRINT,PUNCH)=17
The following example is only supported in SOLs 101 and 106:
THSTRN(STRCUR,CORNER,PRINT) = ALL
```

DESCRIBERS FOR SOL 401:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.

DESCRIBERS FOR SOLS 101 AND 106:

Describer	Meaning
PRINT	Compute and write output to the print file (f06). (Default)
PUNCH	Compute and write output to the punch file (pch).
PLOT	Compute output.
ALL	Requests output for all grid points.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0)
NONE	Output is not computed.
STRCUR	Elastic strain at the reference plane and curvatures is output for plate elements.
FIBER	Elastic strain at locations Z1, Z2 is computed for plate elements.
CENTER	Outputs elastic strains at the center of the element only. See Remark 2 for SOLs 101 and 106.
CORNER or BILIN	Outputs elastic strains at the center and corner grid points only using extrapolation. See Remark 3 for SOLs 101 and 106.
SGAGE	Outputs elastic strains at the center and corner grid points using strain gage approach. See Remark 4 for SOLs 101 and 106.
CUBIC	Outputs elastic strains at the center and corner grid points using cubic bending correction. See Remark 5 for SOLs 101 and 106.

REMARKS FOR SOL 401:

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

REMARKS FOR SOLS 101 AND 106:

1. The STRCUR option is default for shell elements that reference a PCOMP or PCOMPG entry. Any FIBER option on STRAIN or ELSTRN or THSTRN will override STRCUR option for all STAIN and ELSTRN and THSTRN.
2. For the CENTER option, thermal strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element.
3. For the CORNER (or BILIN) option, thermal strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry.
4. For the SGAGE option, thermal strains are output at the center and grid points for CQUAD4 elements that reference a PCOMP or PCOMPG entry.
5. For the CUBIC option, thermal strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PCOMP or PCOMPG entry.
6. The CENTER/CORNER(BILIN)/SGAGE/CUBIC option on STRAIN card overrides the corresponding option on ELSTRN and THSTRN card.

TITLE

Output Title

Defines a character string that will appear on the first heading line of each page of NX Nastran printer output.

FORMAT:

TITLE=title

EXAMPLES:

```
TITLE=RIGHT WING, LOAD CASE 3.
```

DESCRIBERS:

Describer	Meaning
title	Any character string.

REMARKS:

1. If this command appears under a SUBCASE command, then the title appears in the output for that subcase only.
2. If this command appears before all SUBCASE commands, then the title is used in all subcases without a TITLE command.
3. If no TITLE command is present, then the title line will contain data and page numbers only.
4. The title also appears on plotter output.

TRIM

Aerodynamic Trim Variable Constraint Selection

Selects trim variable constraints in static aeroelastic response.

FORMAT:

TRIM=n

EXAMPLES:

```
TRIM=1
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a TRIM Bulk Data entry. (Integer>0)

REMARKS:

Aerodynamic extra points (trim variables) that are not constrained by a TRIM Bulk Data entry will be free during the static aeroelastic response solution.

TSTEP

Transient Time Step Set Selection

Selects integration and output time steps for linear or nonlinear transient analysis.

For SOL 401: Selects time stepping for static analysis.

For SOLs 601 and 701: Selects time stepping for advanced nonlinear analysis.

FORMAT:

TSTEP=n

EXAMPLES:

```
TSTEP=731
```

DESCRIBERS:

Describer	Meaning
n	Set identification number of a TSTEP or TSTEPNL Bulk Data entry. (Integer>0)
n (For SOL 401)	Set identification number of a TSTEP1 Bulk Data entry. (Integer>0)
n (For SOLs 601 and 701)	Set identification number of a TSTEP Bulk Data entry. (Integer>0)

REMARKS:

1. A TSTEP entry must be selected to execute a linear transient analysis (SOLs 109 or 112) and TSTEPNL for a nonlinear transient analysis (SOLs 129 and 159).
2. A TSTEPNL entry must be selected in each subcase to execute a nonlinear transient problem.

3. For the application of time-dependent loads in modal frequency response analysis (SOLs 111 and 146), a TSTEP entry must be selected by the TSTEP command. The time-dependent loads will be recomputed in frequency domain by a Fourier Transform.

REMARKS RELATED TO SOLS 601 AND 701:

1. In SOL 601, TSTEP selects the solution time steps for a static or transient analysis.

2. In SOL 701, the actual time step size may be based on the critical time step size for stability calculated by the program. However, the total solution time is considered. For example, if the TSTEP entry specifies 5 time steps of step size 0.001, the program will execute up to solution time 0.005 using an appropriate time step size. To request that the program use the time step size specified in TSTEP Bulk Data entry, specify XSTEP=1 in the NXSTRAT entry. See the *Advanced Nonlinear Theory and Modeling Guide*on the critical time step size and when results are output.

TSTEPNL

Transient Time Step Set Selection for Nonlinear Analysis

See the description for TSTEP.

TSTRU

Temperature Set ID for a Structures Run

Defines a temperature set ID for a structures run based on a heat transfer subcase.

FORMAT:

TSTRU=n

EXAMPLES:

```
TSTRU=999
```

DESCRIBERS:

Describer	Meaning
n	Set identification for use on TEMP(LOAD)=n or TEMP(INIT)=n

REMARKS:

1. TSTRU should be placed in the heat transfer subcase.
2. If TSTRU does not explicitly appear in the heat transfer subcase, it is defaulted to TSTRU=Heat Transfer Subcase ID.
3. In a structures run, a temperature set generated from a heat transfer run will override an existing temperature set with an identical set ID defined with TEMP, TEMPD, TEMPP1, TEMPRB or any combination.
4. TSTRU may be placed in the first subcase of a PARAM,HEATSTAT,YES run.
5. TSTRUs may be placed in each subcase of an APPHEAT run. The associated structures run then requires the following:

```
ASSIGN heat_run='heat transfer job name.MASTER'
DBLOC DATABLK=(UG,EST,BGPDTS,CASECCR/CASEHEAT) LOGICAL=heat_run
```

6. Heat transfer runs and structure runs must have the same mesh.

7. For nonlinear heat transfer SOL 106 or SOL 153 the INOUT field on the NLPARM Bulk Data entry must be blank or NO if the results of the run are to be transferred to a linear structures run.

   PARAM,NLHTLS,-1 should be placed in the nonlinear heat run. This will place UG heat transfer on the database.

VECTOR

Displacement Output Request

Requests the form and type of displacement vector output.

See the description for DISPLACEMENT.

VELOCITY

Velocity Output Request

Requests the form and type of velocity vector output.

FORMAT:

$$\text{VELOCITY}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{ABS}\\ \text{REL}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \begin{bmatrix}\text{RPUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

EXAMPLES:

```
VELOCITY=5
VELOCITY(SORT2,PHASE,PUNCH)=ALL
```

DESCRIBERS:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.

Describer	Meaning
PLOT	Generates, but does not print, velocities.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ABS	For enforced motion dynamic analysis, velocity results will be output as absolute velocity.
REL	For enforced motion dynamic analysis, velocity results will be output relative to the enforced motion input.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RMS	Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RALL	Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark 7.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 7.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 7.

Describer	Meaning
RPUNCH	Writes random analysis results to the punch file. See Remark 7.
ALL	Velocity for all solution points will be output.
NONE	Velocity for no solution points will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. Both PRINT and PUNCH may be requested.

2. Velocity output is only available for transient and frequency response problems.

3. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

4. VELOCITY=NONE overrides an overall output request.

5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer request is present for magnitude/phase representation.

6. Velocity results are printed and/or punched in the global coordinate system (see field CD on the GRID bulk data entry). The coordinate system for plotted velocity output depends on the PARAM,POST setting. See the parameter POST.

7. The following applies to random solutions:

   - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

   - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

   - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

   - Autocorrelation (ATOC) calculations require the RANDT1 bulk entry.

8. When doing enforced motion dynamic analysis and relative output is requested (using the REL describer), the output will be relative to the input as described by the equation:

$$u_f = y_f - K_{ff}^{-1} K_{fs} u_s$$

where u_f = absolute displacement

y_f = relative displacement

u_s = enforced motion.

REMARKS RELATED TO SOLS 601 AND 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC, RMS, and RALL are ignored.

2. Displacements, velocities and accelerations must be output for the same set of grid points if requested. Output requested for set n in this command will be combined with the sets requested in the DISPLACEMENT and ACCELERATION commands, and velocities will be output at the grid points of the combined set.

VOLUME

Volume Definition

Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

FORMAT:

$$\text{VOLUME id SET sid, [PRINCIPAL, DIRECT STRESS], } \left[\text{SYSTEM} \left\{ \begin{array}{c} \text{ELEMENT} \\ \text{CORD cid} \\ \text{BASIC} \end{array} \right\} \right]$$

EXAMPLES:

```
VOLUME  21 SET 2
```

DESCRIBERS:

Describer	Meaning
id	Volume identification number.
sid	Set identification number of a SET command that defines the elements in the volume. Either form of the SET command may be used. The default is all elements.
PRINCIPAL	Requests principal stresses or strains, direction cosines, mean pressure, and von Mises equivalent stresses or strains to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
DIRECT	Requests direct stress or strains, mean pressure stress and von Mises equivalent stress to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
SYSTEM	Used to specify the reference coordinate system to be used to define the output stress orientation coordinate system.

Describer	Meaning
ELEMENT	Specifies the element coordinate system.
CORD cid	Specifies the coordinate system specified on a CORDij entry.
BASIC	Specifies the basic coordinate system.

REMARKS:

1. VOLUME commands must be specified after OUTPUT(POST).
2. The volume identification number must be referenced on a SET command, specifically the SET command which comes after OUTPUT(POST). This SET identification number may then be referenced on the GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands.
3. If ELEMENT is specified, element stresses or strains are not transformed.
4. In the example given above, for all elements in SET 2:
   - Both PRINCIPAL and DIRECT stress are output.
   - The BASIC output system is used.

WEIGHTCHECK

Rigid Body Mass Reduction Check

At each stage of the mass matrix reduction, compute the rigid body mass and compare with the rigid body mass of the g-set.

FORMAT:

$$\text{WEIGHTCHECK}\left[\begin{array}{l}\left(\begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix}, \text{SET}=\left(\begin{Bmatrix}\text{G, N, N+AUTOSPC, F, A,V}\\ \text{ALL}\end{Bmatrix}\right)\right.\\ \left.\text{GRID} = \text{gid, CGI} = \begin{bmatrix}\text{YES}\\ \text{NO}\end{bmatrix}, \begin{bmatrix}\text{WEIGHT}\\ \text{MASS}\end{bmatrix}\right)\end{array}\right]=\begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

EXAMPLES:

```
WEIGHTCHECK=YES
WEIGHTCHECK(GRID=12,SET=(G,N,A),MASS)=YES
```

DESCRIBERS:

Describer	Meaning
PRINT	Writes output to the print file (default).
NOPRINT	Does not write output to the print file.
SET	Selects degree-of-freedom set(s). (Default SET=G)
gid	Reference grid point for the calculation of rigid body motion. The default is the origin of the basic coordinate system.
CGI	For SET = N, N+AUTOSPC, F, A, or V, CGI = YES requests output of center of gravity and mass moments of inertia. This output is always printed for SET = G. (Default: CGI = NO).
WEIGHT/MASS	Selects output in units of weight or mass. (Default=WEIGHT)

REMARKS:

1. WEIGHTCHECK must be specified above the subcase level.

2. For SET=N, N+AUTOSPC, F, or A, the WEIGHTCHECK command also outputs a percentage loss or gain in the reduced rigid body mass matrix (e.g., MAA) as compared to the g-set rigid body mass matrix (e.g., MGG). G must also be requested to obtain this comparison; e.g., WEIGHTCHECK(SET=(G,A))=YES.

3. SET=N+AUTOSPC uses the mass matrix for the n-set with the rows corresponding to degrees-of-freedom constrained by the PARAM, AUTOSPC operation zeroed out. If AUTOSPC was not performed then this check is redundant with respect to SET=N.

4. WEIGHTCHECK is available in all SOLs. However, in SOLs 101, 105, 114, and 116, because no mass reduction is performed, only WEIGHTCHECK(SET=G) is available.

4.5 Case Control Applicability Tables

The following tables describe the applicability of Case Control commands to Solution Sequences:

Table 4-7	SOLs 101 through 115
Table 4-8	SOLs 116 through 701

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
A2GG	X	X	X	X	X	X	X	X	X	X	X	X
ACCELERATION		X				X	X		X	X		
ADAMSMNF		X										
ADAPTERR												
ADMRECVR		X										
AECONFIG												
AEROF												
AESYMXY												
AESYMXZ												
ANALYSIS												
APRESSURE												
AUXCASE												
AUXMODEL												
B2GG	X	X	X	X	X	X	X	X	X	X	X	X
B2PP					X	X	X	X	X	X		
BC		X										
BCRESULTS	X	X	X									
BCSET	X	X							X	X		
BEGIN BULK	X	X	X	X	X	X	X	X	X	X	X	X
BGRESULTS	X	X	X			X						
BGSET	X	X	X	X	X	X	X	X	X	X	X	X
BOLTLD	X	X	X		X	X	X	X	X	X		
BOUTPUT				X								

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
CKGAP												
CLOAD				X								
CMETHOD					X			X				
CSSCHD												
CZRESULTS												
DEFORM	X		X								X	
DESGLB												
DESOBJ												
DESSUB												
DISPLACEMENT	X	X	X	X	X	X	X	X	X	X	X	X
DIVERG												
DLOAD					X	X	X	X	X	X		
DRSPAN												
DSAPRT												
DSYM											X	X
EBDSET												
ECHO	X	X	X	X	X	X	X	X	X	X	X	X
ECHOOFF	X	X	X	X	X	X	X	X	X	X	X	X
ECHOON	X	X	X	X	X	X	X	X	X	X	X	X
EDE		X				X			X	X		X
EFLOAD	X	X				X						
ELSTRN												
EKE		X				X			X	X		X

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
ELSDCON	X		X									
ELSUM	X	X	X	X	X	X	X	X	X	X	X	X
ENTHALPY												
ERP						X			X			
ESE	X	X	X			X			X	X	X	X
EXTSEOUT	X	X			X	X	X	X	X	X		
FLSFSEL						X			X			
FLSPOUT						X			X			
FLSTCNT						X			X			
FLUX	X										X	
FMETHOD												
FORCE	X	X	X	X	X	X	X	X	X	X	X	X
FREQUENC						X			X			
GELSTRN												
GKRESULTS												
GPFORCE	X	X	X	X		X	X		X	X	X	X
GPKE		X							X*			X
GPRSORT	X	X	X	X	X	X	X	X	X	X	X	X
GPSDCON	X		X								X	
GPSTRAIN	X	X		X			X			X	X	X
GPSTRESS	X	X		X			X			X	X	X
GROUNDCHECK	X	X	X	X	X	X	X	X	X	X	X	X
GSTRAIN												

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
GSTRESS												
GTHSTRN												
GUST												
HARMONICS	X	X	X		X	X	X	X	X	X	X	X
HDOT												
HOUTPUT											X	X
IC							X					
INCLUDE	X	X	X	X	X	X	X	X	X	X	X	X
JINTEG												
K2GG	X	X	X	X	X	X	X	X	X	X	X	X
K2PP					X	X	X	X	X	X		
K42GG	X	X	X	X	X	X	X	X	X	X	X	X
LABEL	X	X	X	X	X	X	X	X	X	X	X	X
LINE	X	X	X	X	X	X	X	X	X	X	X	X
LOAD	X		X	X		X	X		X	X	X	
LOADSET	X		X	X		X	X		X	X	X	
M2GG	X	X	X	X	X	X	X	X	X	X	X	X
M2PP					X	X	X	X	X	X		
MASTER	X	X	X	X	X	X	X	X	X	X	X	X
MAXLINES	X	X	X	X	X	X	X	X	X	X	X	X
MAXMIN	X	X	X				X				X	X
MBDEXPORT		X										X
MBDRECVR		X										X

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
MEFFMASS		X						X	X	X		
METHOD		X		X	X	X	X	X	X	X		X
MFLUID		X			X	X	X	X	X	X		X
MODALE									X			
MODCON								X	X	X		
MODES		X										X
MODSEL		X	X	X				X	X	X		X
MODTRAK												
MPC	X	X	X	X	X	X	X	X	X	X	X	X
MPCFORCES	X	X	X		X	X	X	X	X	X	X	X
MPRES		X			X	X	X	X	X	X		X
NLCNTL												
NLLOAD							X			X		
NLPARM				X								
NLSTRESS				X								
NONLINEAR							X			X		
NOUTPUT											X	X
NSM	X	X	X	X	X	X	X	X	X	X	X	X
OFREQUENCY						X			X			
OLOAD	X		X	X		X	X		X	X	X	
OMODES		X	X					X*	X*	X*		X
OTEMP												
OTIME							X			X	X	

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X	X	X
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X	X	X
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X	X	X
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X	X	X
P2G	X		X	X		X	X		X	X	X	
PAGE	X	X	X	X	X	X	X	X	X	X	X	X
PANCON						X			X			
PARAM	X	X	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X	X	X
PFRESULTS												
PLOTID	X	X	X	X	X	X	X	X	X	X	X	X
PRESSURE		X			X	X	X					
RANDOM						X			X			
RCROSS						X			X			
REPCASE	X	X										
RESVEC		X		X				X	X	X		
RIGID	X	X	X	X	X	X	X	X	X	X		
RMAXMIN	X						X			X		
RMETHOD	X				X	X	X	X	X	X		
RSMETHOD		X			X	X	X	X	X	X		
SACCELERATION						X	X		X	X		
SDAMPING								X	X	X		

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
SDISPLACEMENT		X	X		X	X	X		X	X		X
SEALL	X	X	X	X	X	X	X	X	X	X	X	X
SEDR	X	X	X	X	X	X	X	X	X	X	X	X
SEDV												
SEEXCLUDE	X	X	X	X	X	X	X	X	X	X	X	X
SEFINAL	X	X	X	X	X	X	X	X	X	X	X	X
SEKREDUCE	X	X	X	X	X	X	X	X	X	X	X	X
SELGENERATE	X		X	X		X	X		X	X	X	
SELREDUCE	X		X	X		X	X		X	X	X	
SEMGENERATE	X	X	X	X	X	X	X	X	X	X	X	X
SEMREDUCE		X	X		X	X	X	X	X	X		X
SEQDEP												
SERESP												
SET	X	X	X	X	X	X	X	X	X	X	X	X
SETMC								X	X	X		
SHELLTHK												
SKIP	X	X	X	X	X	X	X	X	X	X	X	X
SMETHOD	X					X						
SPC	X	X	X	X	X	X	X	X	X	X	X	X
SPCFORCES	X	X	X	X	X	X	X	X	X	X	X	X
STATSUB		X	X		X	X	X	X	X	X		X
STRAIN	X	X	X	X	X	X	X	X	X	X	X	X
STRESS	X	X	X	X	X	X	X	X	X	X	X	X

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
STRFIELD	X	X	X				X			X	X	X
SUBCASE	X	X	X	X	X	X	X	X	X	X	X	X
SUBCOM	X											
SUBSEQ	X											
SUBTITLE	X	X	X	X	X	X	X	X	X	X	X	X
SUPER	X	X	X	X	X	X	X	X	X	X	X	X
SUPORT1	X	X	X	X	X	X	X	X	X	X	X	X
SURFACE		X	X					X	X	X	X	X
SVECTOR	X	X		X			X			X		X
SVELOCITY						X	X		X	X		
SYM	X											
SYMCOM	X											
SYMSEQ	X											
TEMP(INIT)	X	X	X	X	X	X	X	X	X	X	X	X
TEMP(LOAD)	X		X	X		X	X		X	X	X	
TEMP(MATE)	X	X	X		X	X	X	X	X	X	X	X
TFL					X	X	X	X	X	X		
THERMAL	X										X	
THSTRN												
TITLE	X	X	X	X	X	X	X	X	X	X	X	X
TRIM												
TSTEP							X			X		
TSTRU	X											

Table 4-7. Case Control Command Applicability – Solutions 101 through 115

Command Name	Solution Number (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
VECTOR	X	X	X	X	X	X	X	X	X	X	X	X
VELOCITY						X	X		X	X		
VOLUME	X	X		X			X			X	X	X
WEIGHTCHECK	X	X	X	X	X	X	X	X	X	X	X	X

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
A2GG	X	X	X	X	X	X	X	X	X	X		
ACCELERATION		X	X			X			X	X		X
ADAMSMNF												
ADAPTERR											X	
ADMRECVR												
AECONFIG				X	X					X		
AEROF				X		X				X		
AESYMXY				X	X	X				X		
AESYMXZ				X	X	X				X		
ANALYSIS							X			X	X	
APRESS				X						X		
AUXCASE										X		
AUXMODEL										X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
B2GG	X	X	X	X	X	X	X	X	X	X		
B2PP		X	X		X	X		X		X		
BC									X			
BCRESULTS											X	X
BCSET											X	X
BEGIN BULK	X	X	X	X	X	X	X	X	X	X	X	
BGRESULTS			X				X	X			X	
BGSET	X	X	X				X	X	X	X	X	601
BOLTLD											X	601
BOUTPUT			X				X	X				
CKGAP											X	
CLOAD							X					
CMETHOD					X							
CRSTRN											X	
CSSCHD				X								
CZRESULTS											X	
DEFORM	X			X						X		
DESGLB										X		
DESOBJ										X		
DESSUB										X		
DISPLACEMENT	X	X	X	X	X	X	X	X	X	X	X	X
DIVERG				X						X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
DLOAD		X	X		X	X		X		X	X	X
DRSPAN										X		
DSAPRT										X		
DSYM	X	X										
EBDSET												X
ECHO	X	X	X	X	X	X	X	X	X	X	X	
ECHOOFF	X	X	X	X	X	X	X	X	X	X		
ECHOON	X	X	X	X	X	X	X	X	X	X		
EDE		X							X	X		
EFLOAD												
EKE		X							X	X	X	
ELSDCON				X						X		
ELSTRN											X	
ELSUM	X	X	X	X	X	X			X	X	X	X
ENTHALPY								X				
ERP										X		
ESE	X	X		X					X	X	X	
EXTSEOUT			X	X	X	X		X	X			
FLSFSEL										X		
FLSPOUT												
FLSTCNT												
FLUX							X	X		X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
FMETHOD					X					X		
FORCE	X	X	X	X	X	X	X	X	X	X	X	
FREQUENC		X				X				X		
GCRSTRN											X	
GELSTRN											X	
GKRESULTS												601
GPFORCE	X	X		X		X			X	X	X	601
GPKE									X	X	X	
GPLSTRN											X	
GPRSORT	X	X	X	X	X	X	X	X	X	X		
GPSDCON				X			X			X		
GPSTRAIN			X	X					X			
GPSTRESS			X	X					X			
GROUNDCHECK	X	X	X	X	X	X			X	X	X	
GSTRAIN											X	
GSTRESS											X	
GTHSTRN											X	
GUST						X	X					
HARMONICS	X	X							X	X		
HDOT								X				
HOUTPUT	X	X										
IC								X				X

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
INCLUDE	X	X	X	X	X	X	X	X	X	X	X	X
JINTEG											X	
K2GG	X	X	X	X	X	X	X	X	X	X		
K2PP		X	X		X	X		X		X		
K42GG	X	X	X	X	X	X	X	X	X	X		
LABEL	X	X	X	X	X	X	X	X	X	X	X	
LINE	X	X	X	X	X	X	X	X	X	X	X	
LOAD	X	X	X	X		X	X			X		X
LOADSET	X	X	X	X		X	X	X		X		
M2GG	X	X	X	X	X	X	X	X	X	X		
M2PP		X	X		X	X		X		X		
MASTER	X	X	X	X	X	X	X	X	X	X		
MAXLINES	X	X	X	X	X	X	X	X	X	X	X	
MAXMIN			X	X	X	X			X	X		
MBDEXPORT												
MBDRECVR												
MEFFMASS					X	X			X	X	X	
METHOD	X	X	X	X	X	X	X	X	X	X	X	
MFLUID	X	X	X	X	X	X			X	X		
MODALE												
MODCON						X						
MODES	X									X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
MODSEL	X				X	X			X			
MODTRAK												
MPC	X	X	X	X	X	X	X	X	X	X	X	X
MPCFORCES	X	X		X	X	X			X	X	X	
MPRES		X		X	X	X			X	X		
NLCNTL											X	
NLLOAD								X				
NLPARM							X					
NLSTRESS												
NONLINEAR			X					X				
NOUTPUT	X	X										
NSM	X	X	X	X	X	X	X	X	X	X	X	X
OFREQUENCY		X				X				X		
OLOAD	X	X	X	X		X	X	X		X	X	
OMODES									X	X	X	
OPRESS											X	
OTEMP											X	
OTIME	X	X	X	X		X	X	X	X	X		
OUTPUT (blank)	X	X	X	X	X	X	X	X	X	X		
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X		
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X		
P2G	X	X	X	X		X	X			X		
PAGE	X	X	X	X	X	X	X	X	X	X		
PANCON												
PARAM	X	X	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X		
PFRESULTS											X	
PLSTRN											X	
PLOTID	X	X	X	X	X	X	X	X	X	X		
PRESSURE									X	X		
RANDOM		X				X				X		
RCROSS		X				X						
REPCASE									X	X		
RESVEC						X			X	X		
RIGID											X	
RMAXMIN												
RMETHOD												
RSMETHOD		X	X		X	X		X	X	X		
SACCELERATION		X	X			X				X		
SDAMPING					X	X				X		
SDISPLACEMENT		X	X		X	X		X	X	X		
SEALL	X	X	X	X	X	X	X	X	X	X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
SEDR	X	X	X	X	X	X	X	X	X	X		
SEDV										X		
SEEXCLUDE	X	X	X	X	X	X	X	X	X	X		
SEFINAL	X	X	X	X	X	X	X	X	X	X		
SEKREDUCE	X	X	X	X	X	X	X	X	X	X		
SELGENERATE	X	X	X	X		X	X	X		X		
SELREDUCE	X	X	X	X		X	X	X		X		
SEMGENERATE	X	X	X	X	X	X	X	X	X	X		
SEMREDUCE	X	X	X	X	X	X		X	X	X		
SEQDEP											X	
SERESP										X		
SETMC						X						
SET	X	X	X	X	X	X	X	X	X	X	X	X
SHELLTHK												X
SKIP	X	X	X	X	X	X	X	X	X	X		
SMETHOD											X	
SPC	X	X	X	X	X	X	X	X	X	X	X	X
SPCFORCES	X	X	X	X	X	X	X	X	X	X	X	X
STATSUB	X								X	X		
STRAIN	X	X	X	X	X	X	X	X	X	X	X	
STRESS	X	X	X	X	X	X	X	X	X	X	X	X
STRFIELD	X			X					X	X		

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
SUBCASE	X	X	X	X	X	X	X	X	X	X	X	X
SUBCOM										X		
SUBSEQ										X		
SUBTITLE	X	X	X	X	X	X	X	X	X	X	X	
SUPER	X	X	X	X	X	X	X	X	X	X		
SUPORT1	X	X	X	X	X	X	X	X	X	X		
SURFACE	X								X	X		
SVECTOR	X	X			X		X		X	X		
SVELOCITY		X	X			X		X		X		
SYM										X		
SYMCOM										X		
SYMSEQ										X		
TEMP(INIT)	X	X	X	X	X	X	X	X	X	X	X	X
TEMP(LOAD)		X		X		X				X	X	X
TEMP(MATE)	X	X	X	X	X	X	X	X	X	X		
TFL		X	X		X	X		X		X		
THERMAL							X	X	X	X		
THSTRN											X	
TITLE	X	X	X	X	X	X	X	X	X	X	X	X
TRIM				X						X		
TSTEP			X			X		X		X	X	X
TSTRU												

Table 4-8. Case Control Command Applicability – Solutions 116 through 701

Command Name	Solution Number (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
VECTOR	X	X	X	X	X	X	X	X	X	X		
VELOCITY		X	X			X				X		X
VOLUME	X								X	X		
WEIGHTCHECK	X	X	X	X	X	X			X	X	X	

* For modal portion of solution only.

Chapter 5: Case Control X-Y PLOT Commands

5.1 Case Control X-Y PLOT Commands Overview

The X-Y output request packet of the Case Control Section includes all commands between either OUTPUT(XYPLOT) or OUTPUT(XYOUT) and either BEGIN BULK or OUTPUT(PLOT). The remainder of this section describes the X-Y output commands.

A single set of plotted X-Y pairs is known as a curve. Curves are the entities that the user requests to be plotted. The surface (paper, microfilm frame, etc.) on which one or more curves is plotted is known as a frame. Curves may be plotted on a whole frame, an upper-half frame, or a lower-half frame. Grid lines, tic marks, axes, axis labeling and other graphic control options may be chosen by the user. The program will select defaults for parameters not selected by the user.

Only two commands are required for an X-Y output request.

1. Only one of OUTPUT(XYPLOT) or OUTPUT(XYOUT) at the beginning of the X-Y output command packet.

2. At least one of the commands XYPLOT, XYPEAK, XYPRINT, XYPUNCH, or XYPAPLOT.

The commands OUTPUT(XYPLOT) and OUTPUT(XYOUT) are equivalent. If the X-Y output is to be printed and/or punched, a PLOTTER command is not required.

If only the required commands are used, the graphic control options will all assume default values. Curves using all default parameters have the following general characteristics:

1. Tic marks are drawn on all edges of the frame. Five spaces are provided on each edge of the frame.

2. All tic marks are labeled with their values.

3. Linear scales are used.

4. Scales are selected such that all points fall within the frame.

5. The plotter points are connected with straight lines.

6. The plotted points are not identified with symbols.

The above characteristics may be modified by inserting any of the parameter definition commands in the next section, ahead of the XY____ command(s). The use of a parameter definition command sets the value of that parameter for all following command

operation commands unless the CLEAR command is inserted. If grid lines are requested, they will be drawn at the locations of all tic marks that result from defaults or user requests. The locations of tic marks (or grid lines) for logarithmic scales cannot be selected by the user. Values for logarithmic spacing are selected by the program. The values for the number of tic marks (or grid lines) per cycle depend on the number of logarithmic cycles required for the range of the plotted values.

The definition and rules for the X-Y output commands follow. The form of X-Y output commands differs in many instances from that of similar commands used in the OUTPUT(PLOT) section.

5.2 X-Y Output Command Summary

Commands Applied To All Curves

PLOTTER	Selects format of plot file for interpretation by plotter post-processor.
CAMERA	Selects plotter media.
PENSIZE	Selects pen number.
DENSITY	Selects the line density for microfilm plotters only.
XPAPER	Defines the size of the paper in x-direction.
YPAPER	Defines the size of the paper in y-direction.
XMIN	Specifies the minimum value on the x-axis.
XMAX	Specifies the maximum value on the x-axis.
XLOG	Selects logarithmic or linear x-axis.
YAXIS	Controls the plotting of the y-axis on all curves.
XINTERCEPT	Specifies the location of the x-axis on the y-axis.
UPPER TICS	Specifies how to draw tic marks on the upper edge.
LOWER TICS	Specifies how to draw tic marks on the lower edge.
CURVELINE	Selects lines and/or symbols to be drawn through the x-y points.

XDIVISIONS	Specifies spacing of tic marks on the x-axis for all curves.
XVALUE SKIP	Specifies how often to print the x-values alongside the x-axis tic marks.
CLEAR	Resets X-Y PLOT commands to their default value.
XTITLE	Defines a character string that will appear along the x-axis.
TCURVE	Defines a character string that will appear at the top of the plot frame.
LONG	Controls amount of curve's summary printout.
CSCALE	Defines the scale factor for characters in the plot frame.

Commands Applied to Whole Frame Curves Only

YMIN	Specifies the minimum value on the y-axis.
YMAX	Specifies the maximum value on the y-axis.
XAXIS	Controls the plotting of the x-axis.
YINTERCEPT	Specifies the location of the y-axis on the x-axis.
YLOG	Selects the logarithmic or linear y-axis.
LEFT TICS	Specifies how to draw tic marks on the left edge of the frame.
RIGHT TICS	Specifies how to draw tic marks on the right edge of the frame.
ALLEDGE TICS	Specifies how to draw tic marks on all edges of the frame.
YDIVISIONS	Specifies spacing of tic marks on the y-axis.
YVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
XGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.

YTITLE	Defines a character string that will appear along the y-axis.

Commands Applied to Upper Half Frame Curves Only

YTMIN	Specifies the minimum value on the y-axis.
YTMAX	Specifies the maximum value on the y-axis.
YTAXIS	Controls the plotting of the y-axis.
YTINTERCEPT	Specifies the location of the y-axis on the x-axis.
YTLOG	Selects the logarithmic or linear y-axis.
TLEFT TICS	Specifies how to draw tic marks on the left edge.
TRIGHT TICS	Specifies how to draw tic marks on the right edge.
TALL EDGE TIC	Specifies how to draw tic marks on all edges.
YTDIVISIONS	Specifies spacing of tic marks on the y-axis.
YTVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
XTGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YTGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YTTITLE	Defines a character string that will appear along the y-axis.

Commands Applied to Lower Half Frame Curves Only

YBMIN	Specifies the minimum value on the y-axis.
YBMAX	Specifies the maximum value on the y-axis.
XBAXIS	Controls the plotting of the x-axis.
YBINTERCEPT	Specifies the location of the y-axis on the x-axis.
YBLOG	Selects the logarithmic or linear y-axis.

BLEFT TICS	Specifies how to draw tic marks on left edge.
BRIGHT TICS	Specifies how to draw tic marks on right edge.
BALL EDGE TIC	Specifies how to draw tic marks on all edges.
YBDIVISIONS	Specifies spacing of tic marks on the y-axis.
YBVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
XBGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YBGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YBTITLE	Defines a character string that will appear along the y-axis.

X-Y Plot Generation Commands

XYPAPLOT	Generates X-Y plots for a printer.
XYPEAK	Prints only the summary for all curves.
XYPLOT	Generates X-Y plots for a plotter.
XYPRINT	Generates a table of X-Y pairs for a printer.
XYPUNCH	Generates a table of X-Y pairs for the PUNCH file.

ALLEDGE TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the frame.

FORMAT:

ALLEDGE TICS tic

EXAMPLES:

```
ALLEDGE -1
```


DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 0) -1 Draw tic marks only. 0 Do not draw tic marks or associated values. (Default) 1 Draw tic marks and associated values.

REMARKS:

1. ALLEDGE TICS applies to whole frame curves only.

2. To determine if on any given edge (a) tic marks will be drawn without values, (b) no tic marks or values will be drawn, or (c) tic marks with values will be drawn, the following sum must be computed by the user. Add the tic values of the edge in question to its associated ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS tic values. If the resulting value is less than zero, tic marks will be drawn without values. If the resulting value is zero, no tic marks or values will be drawn. If the resulting value is greater than zero, tic marks with values will be drawn. The user should be careful in his use of the ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS commands. For example, the use of only the ALLEDGE TICS

= -1 command will result in no tic marks or values being drawn since the default values for individual edges is +1. Tic values input may only be -1, 0, or 1.

BALL EDGE TICS

Controls Drawing of Tic Marks on Lower Half

Specifies how to draw tic marks on lower half of frame.

FORMAT:

BALL EDGE TICS tic

EXAMPLES:

```
BALL EDGE TICS -1
```


DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 0) -1 Draw tic marks only. 0 Do not draw tic marks or associated values. (Default) 1 Draw tic marks and associated values.

REMARKS:

1. BALL EDGE TICS applies to lower frame curves only.
2. See ALLEDGE TICS.

BLEFT TICS

Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on the left edge of the lower half of the frame.

FORMAT:

BLEFT TICS tic

EXAMPLES:

```
BLEFT TICS -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1) -1 Draw tic marks only. 0 Do not draw tic marks or associated values. 1 Draw tic marks and associated values. (Default)

REMARKS:

1. BLEFT TICS applies to lower frame curves only.
2. See ALLEDGE TICS.
3. See related command BRIGHT TICS.

BRIGHT TICS

Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on the right edge of the lower half of the frame.

FORMAT:

BRIGHT TICS tic

EXAMPLES:

```
BRIGHT TICS -1
```


DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1) -1 Draw tic marks only. 0 Do not draw tic marks or associated values. 1 Draw tic marks and associated values. (Default)

REMARKS:

1. BRIGHT TICS applies to lower frame curves only.
2. See ALLEDGE TICS.

CAMERA

Plotter Media Selection

Selects plotter media.

FORMAT:

CAMERA ctype

EXAMPLES:

```
CAMERA 1
```

DESCRIBERS:

Describer	Meaning
ctype	Camera type. (Integer 1, 2, or 3; Default = 2) 1 Film 2 Paper (Default) 3 Both

REMARKS:

If the CAMERA command is not specified then CAMERA 2 is assumed.

CLEAR

Resets X-Y PLOT Commands

Resets X-Y PLOT commands to their default value.

FORMAT

CLEAR

REMARKS:

All commands except XTITLE, YTITLE, YTTITLE, YBTITLE, and TCURVE will revert to their default values.

CSCALE

Character Scale Factor

Defines scale factor for characters in the plot frame.

FORMAT:

CSCALE cs

EXAMPLES:

```
CSCA 2.0
```

Describer	Meaning
cs	Scale factor applied to characters in the plot frame.

REMARKS:

1. CSCALE is used to control the spacing of characters when plots are made with the NASTRAN plotter and they are postprocessed with the NX/NASPLOT routine. For example, if the SCALE FACTOR on the NASPLOT data command is 2.0, a value for cs of 0.5 will result in characters of default size (.07 inches) at the regular spacing. A value of 1.8 produces good spacing when using the post-processing plotter programs NASTPLT, TEKPLT, and NEUPS. On the other hand, if the user wishes to double the size of both the plot and the characters, the SCALE FACTOR and the CSCALE FACTOR on the NASPLOT data command should both be set equal to 2.0.

2. The CSCALE command must immediately precede the PLOTTER command. If a second CSCALE command is specified then a second PLOTTER command must also be specified.

CURVELINESYMBOL

Curve, Line and Symbol Selection

Selects lines and/or symbols to be drawn through the x-y points.

FORMAT:

CURVELINESYMBOL symtype

EXAMPLES:

```
CURV 4
```


DESCRIBERS:

Describer	Meaning
symtype	Specifies the symbol drawn at the x-y points. If symtype is 0 then only lines will be drawn through the points with no symbol. If symtype is less than zero then only the symbol and not the lines will be drawn. If symtype is greater than zero then both the symbol and the lines will be drawn. (-9 ≤ Integer ≤ 9, Default = 0)

symtype	Symbol
0	none
1	X
2	*
3	+
4	—
5	.
6	×
7	[ ]
8	< >
9	/ \

REMARKS:

If more than one curve is plotted per frame then the symbol number is incremented by 1 for each curve.

DENSITY

Microfilm Plotter Line Density

Selects the line density for microfilm plotters only.

FORMAT

DENSITY d

EXAMPLE

```
DENS 3
```


DESCRIBERS:

Describer	Meaning
d	Specifies line density scale factor for microfilm plotters. A line density of d is d times heavier than a line density of 1. (Integer ≥ 0, Default = 1)

LEFT TICS

Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on the left edge of whole frame curves.

FORMAT:

LEFT TICS tic

EXAMPLES:

```
LEFT -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1) -1 Draw tic marks only. 0 Do not draw tic marks or associated values. 1 Draw tic marks and associated values. (Default)

REMARKS:

1. LEFT TICS applies to whole frame curves only.
2. See ALLEDGE TICS.
3. See related command RIGHT TICS.

LONG

Summary Print Control

Controls amount of curve's summary printout.

FORMAT:

$$\text{LONG}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	One page for each curve's summary. (Default)
NO	Condensed curve summary.

REMARKS:

If LONG is not specified then LONG=NO is assumed.

LOWER TICS

Controls Drawing of Tic Marks on Lower Edge

Specifies how to draw tic marks on lower edge.

FORMAT:

LOWER TICS tic

EXAMPLES:

```
LOWER -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

REMARKS:

1. LOWER TICS applies to all curves.
2. See ALLEDGE TICS.

PENSIZE

Pen Selection

Selects pen number.

FORMAT:

PENSIZE p

EXAMPLES:

```
PENS  3
```


DESCRIBERS:

Describer	Meaning
p	Specifies pen number that is used to generate the plot. (Integer > 0, Default = 1)

PLOTTER

X-Y Plot File Format

Selects format of plot file for interpretation by plotter post-processor.

FORMAT:

$$\text{PLOTTER}\left\{\begin{array}{c}\text{NAST}\\ \text{SC}\end{array}\right\}$$

EXAMPLES:

PLOTTER NAST

DESCRIBERS:

Describer	Meaning
NAST	Specifies format suitable for postscript plotters. (Default)
SC	Specifies Stromberg-Carlson microfilm plotter format.

RIGHT TICS

Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on the right edge of the frame.

FORMAT:

RIGHT TICS tic

EXAMPLES:

```
RIGHT -1
```


DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

REMARKS:

1. RIGHT TICS applies to whole frame curves only.
2. See ALLEDGE TICS.
3. See related command LEFT TICS.

TALL EDGE TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the upper half of the frame.

FORMAT:

TALL EDGE TICS tic

EXAMPLES:

```
TALL -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 0)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values. (Default)
1	Draw tic marks and associated values.

REMARKS:

1. TALL EDGE TICS applies to upper half frame curves only.
2. See ALLEDGE TICS.

TCURVE

Curve Title

Defines a character string that will appear at the top of the plot frame.

FORMAT:

TCURVE ctitle

EXAMPLES:

```
TCUR  RIGHT WING -- LOAD CASE 3
```


DESCRIBERS:

Describer	Meaning
ctitle	Any character string. (Character, Default = blank)

REMARKS:

TCURVE may not be continued to the next command line.

TLEFT TICS

Controls Drawing of Tic Marks on the Left-Edge

Specifies how to draw tic marks on the left edge of the upper half of the frame.

FORMAT:

TLEFT TICS tic

EXAMPLES:

```
TLEFT -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

REMARKS:

1. TLEFT TICS applies to upper half frame curves only.
2. See ALLEDGE TICS.
3. See related command TRIGHT TICS.

TRIGHT TICS

Controls Drawing of Tic Marks on the Right Edge

Specifies how to draw tic marks on the right-edge of the upper half of the frame.

FORMAT:

TRIGHT TICS tic

EXAMPLES:

```
TRIGHT -1
```


DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

REMARKS:

1. TRIGHT TICS applies to upper half frame curves only.
2. See ALLEDGE TICS.
3. See related command TLEFT TICS.

UPPER TICS

Controls Drawing Of Tic Marks On Upper Edge

Specifies how to draw tic marks on the upper edge.

FORMAT:

UPPER TICS tic

EXAMPLES:

```
UPPER -1
```

DESCRIBERS:

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

REMARKS:

1. UPPER TICS applies to all curves.
2. See ALLEDGE TICS.
3. See related command LOWER TICS.

XAXIS

X-Axis Plot Control

Controls the plotting of the x-axis on whole frame curves only.

FORMAT:

$$\text{XAXIS}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

REMARKS:

1. XAXIS applies to whole frame curves only.
2. See related command YAXIS.

XBAXIS

X-Axis Plot Control

Controls the plotting of the x-axis on lower half frame curves only.

FORMAT:

$$\text{XBAXIS}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

REMARKS:

XBAXIS applies to lower half frame curves only.

XBGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on lower half frame curves only.

FORMAT:

$$\text{XBGRID LINES} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

REMARKS:

1. XBGRID applies to lower half frame curves only.
2. See the related command YBGRID LINES.

XDIVISIONS

Tic Spacing on X-Axis

Specifies spacing of tic marks on the x-axis for all curves.

FORMAT:

XDIVISIONS xd

EXAMPLES:

```
XDIV 10
```

DESCRIBERS:

Describer	Meaning
xd	Number of spaces between tic marks on x-axis. (Integer > 0, Default = 5)

REMARKS:

1. XDIVISIONS applies to all curves and to the commands: UPPER TICS, LOWER TICS, and YINTERCEPT.
2. XDIVISIONS is ignored for logarithmic x-axes.

XGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on whole frame curves only.

FORMAT:

$$\text{XGRID LINES} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

REMARKS:

1. XGRID applies to whole frame curves only.
2. See the related command YGRID LINES .

XINTERCEPT

Location of X-Axis on Y-Axis

Specifies the location of the x-axis on the y-axis.

FORMAT:

XINTERCEPT xi

EXAMPLES:

```
XINT 50.
```

DESCRIBERS:

Describer	Meaning
xi	Location of x-axis on the y-axis. (Real, Default = 0.0)

XLOG

Logarithmic or Linear X-Axis

Selects logarithmic or linear x-axis.

FORMAT:

$$\text{XLOG}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot a logarithmic x-axis.
NO	Plot a linear x-axis. (Default)

REMARKS:

1. XLOG applies to all curves.
2. The default value for tic division interval depends on the number of log cycles. The default values for tic divisions are given below but will range over whole cycles

Number of Cycles	Intermediate Values
1, 2	2., 3., 4., 5., 6., 7., 8., 9.
3	2., 3., 5., 7., 9.,
4	2., 4., 6., 8.,
5	2., 5., 8.
6, 7	3., 6.
8, 9, 10	3.

XMAX

Maximum X-Axis Value

Specifies the maximum value on the x-axis.

FORMAT:

XMAX xmax

EXAMPLES:

```
XMAX 100.
```

DESCRIBERS:

Describer	Meaning
xmax	Maximum value on the x-axis. (Real)

REMARKS:

1. If XMAX is not specified then the maximum value is set to the highest value of x.
2. See related commands XMIN, YMIN, and YMAX.

XMIN

Minimum X-Axis Value

Specifies the minimum value on the x-axis.

FORMAT:

XMIN xmin

EXAMPLES:

```
XMIN 100.
```


DESCRIBERS:

Describer	Meaning
xmin	Minimum value on the x-axis. (Real)

REMARKS:

1. XMIN applies to all curves.
2. If XMIN is not specified then the minimum value is set to the lowest value of x.
3. See related commands XMAX, YMIN, and YMAX.

XPAPER

Paper Size in the X-Direction

Defines the size of the paper in the x-direction.

FORMAT:

XPAPER xsize

EXAMPLES:

```
XPAP 10.
```

DESCRIBERS:

Describer	Meaning
xsize	Size of paper in the x-direction and in inches. (Real, Default = 20.0)

REMARKS:

1. The default paper size is 20 by 20 inches.
2. See related command YPAPER.

XTGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on upper half frame curves only.

FORMAT:

$$\text{XTGRID LINE} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

REMARKS:

XTGRID applies to upper half frame curves only.

XTITLE

X-Axis Title

Defines a character string that will appear along the x-axis.

FORMAT:

XTITLE xtit

EXAMPLES:

```
XTIT   RIGHT WING CASE 3 - TIME
```

DESCRIBERS:

Describer	Meaning
xtit	Any character string. (Character, Default = blank)

REMARKS:

1. XTITLE may not be continued to the next command line.
2. XTITLE applies to all curves.

XYPAPLOT

Generate X-Y Plots for a Printer

Generates X-Y plots for a printer.

See XYPLOT for Format, Describers, and additional Remarks.

REMARKS:

1. The x-axis moves vertically along the page and the y-axis moves horizontally along the page.
2. An asterisk (*) identifies the points associated with the first curve of a frame, then for successive curves on a frame, the points are designated by symbols O, A, B, C, D, E, F, G and H.

XYPEAK

Print Curve Summary

Prints only the summary for all curves. The summary output is titled:

“X Y - O U T P U T S U M M A R Y”

and is also printed under XYPLOT, XYPUNCH, XYPRINT, and XYPAPLOT. This output contains the maximum and minimum values of y for the range of x.

See XYPLOT for Format, Describers, and additional Remarks.

XYPLOT

Generates X-Y Plots for a Plotter

Generates X-Y Plots for a Plotter.

FORMAT

```
XYPLOT    yvtype ptype [ i1, i2, i3,... ] /
          id11 (itemu11 [, iteml11]) , id12 (itemu12 [, iteml12]) , ... /
          id21 (itemu21 [, iteml21]) , id22 (itemu22 [, iteml22]) , ... / ....
```


DESCRIBERS:

Describer	Meaning	
yvtype	Type of y-value to be plotted: (Character)	
	ACCE	Acceleration in the physical set
	BOUT	Slideline contact output
	DISP	Displacement in the physical set
	ELFORCE	Element force
	ENTHALPY	Enthalpy in the physical set
	FLUX	Element Heat flux
	HDOT	Rate of change of enthalpy in the physical set
	NONLINEAR	Nonlinear applied load
	OLOAD	Applied Load
	PRESSURE	Pressure of fluid-structure body
	SACCE	Acceleration in the solution set

Describer	Meaning	
	SDISP	Displacement in the solution set
	SPCF	Single-point force of constraint
	STEMP	Temperature in the solution set
	STRESS	Element stress
	SVELO	Velocity in the solution set
	TEMP	Temperature in the physical set
	VECTOR	Displacement in the physical set
	VELO	Velocity in the physical set
	VG	Flutter analysis
ptype	Plot type defining the meaning of i1, i2, ... etc., idi, itemui and itemli. (Character, Default = “RESPONSE”)	
	RESPONSE	Time or frequency in SORT2 format or grid identification numbers in SORT1 format. (Default)
	AUTO	Autocorrelation function on whole frame curves only.
	PSDF	Power spectral density function on whole frame curves only.
	SPECTRAL	Response spectrum on whole frame curves only.
i1, i2,...	Subcase identification numbers for ptype = RESPONSE. The list must be specified in ascending order. For ptype = SPECTRAL, the subcase refers to the RECNO in the DTI,SPSEL Bulk Data entry. The list is ignored for ptype = AUTO and PSDF. (Integer ≥ 0, Default is all subcases)	
idij	Element, grid, scalar, or extra point identification number for y-value for frame i. For yvtype = VG, idij refers to the loop count of a flutter analysis. (Integer > 0)	

Describer	Meaning
itemuij,	Item code for y-value. itemuij is for the upper half or whole itemlij curves on frame i and itemlij is for lower half curves only on frame i. If itemlij is not specified, then whole frame curves will be plotted with itemuij. itemlij is ignored for ptype = “AUTO”, “PSDF”, and “SPECTRAL”. (Character or Integer > 0) For elements, the code represents a component of the element stress, strain, or force and is described in the *Item Codes* topic in this guide. For ptype = “AUTO” and “PSDF”, the complex stress or strain item codes need to be used. As the output quantities are real, you can use either the real or the imaginary item code. Both will give the same result. For grid points and ptype = “RESPONSE”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3, T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, or R3IP, where Ti stands for the i-th translational component, Ri stands for the i-th rotational component, RM means real or magnitude, and IP means imaginary or phase. For scalar or extra points or heat transfer analysis, use T1, T1RM, or T1IP. For grid points and ptype = “AUTO” or “PSDF”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3. For scalar or extra points, use T1. For yvtype = VG, itemui and/or itemli can be “F” for frequency or “G” for damping.

REMARKS:

1. Multiple XYPLOT, XYPUNCH, XYPRINT, XYPEAK and/or XYPAPLOT commands may be specified in the OUTPUT(XYPLOT) section.

2. Solution set requests are more efficient, because the time consuming recovery of the dependent displacements can be avoided.

3. The item codes appear in printed summaries under the “CURVE ID” column for grid points as well as element data. For grid points, the component number in this listing will have the values 3 through 14, corresponding to T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, R3IP, respectively. For

scalar points, the component value is always 3. The grid point output component numbers are listed in the following table.

Input	Output Grid Component Number
T1RM	3
T2RM	4
T3RM	5
R1RM	6
R2RM	7
R3RM	8
T1IP	9
T2IP	10
T3IP	11
R1IP	12
R2IP	13
R3IP	14

4. The information after each slash (/) specifies the curve(s) that are to be plotted on the same frame. The describer idij identifies the grid point j or element j associated with the frame number i. All plot requests on one command are sorted on idij to improve the efficiency of the plotting process. Symbols are assigned in order by idij.

5. If any of the item codes, itemlij or itemuij, are not specified; e.g. (8,) or (,5), the corresponding half frame curve is not plotted. If both the comma (,) and itemlij not specified; e.g., (8), then whole frame curves will be plotted. Also, for any single frame, the specifications of “(itemuij,itemlij)” must be consistently half frame (upper and/or lower) or whole frame. For example on half frame curves, if iteml11 and the comma is not specified then either iteml12 or itemu12 must not be specified and on whole frame curves, the commas, iteml11, and iteml12 must not be specified. In other words, the curves on each plot frame must be all whole or half (upper and/or lower).

6. The XYPLOT command may be continued on the next line as long as “XYPLOT yvtype ptype [i1, i2, i3,...] /” is specified on the first line.

7. Specifying a nonexistent grid point may cause the program to exit in the XYTRAN module and missing plots to occur.

XYPRINT

Generate Table of X-Y Pairs for a Printer

Generates tabular printer output of the X-Y pairs.

See XYPLOT for Format, Describers, and additional Remarks.

XYPUNCH

Generate Table of X-Y Pairs for the PUNCH File

Generates tabular punch output of the X-Y pairs. This is the same as XYPRINT, except that the output is written to the PUNCH file.

See XYPLOT for Format, Describers, and additional Remarks.

XVALUE PRINT SKIP

Print Values on X-Axis Tic Marks

Specifies how often to print the x-values alongside the x-axis tic marks.

FORMAT:

XVALUE PRINT SKIP xvps

EXAMPLES:

```
XVAL 5
```


DESCRIBERS:

Describer	Meaning
xvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values. (Integer ≥ 0)

REMARKS:

XVALUE applies to all curves.

YAXIS

Y-Axis Plot Control

Controls the plotting of the y-axis on all curves.

FORMAT:

$$\text{YAXIS}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the y-axis.
NO	Do not plot the y-axis. (Default)

YBDIVISIONS

Tic Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for lower half frame curves only.

FORMAT:

YBDIVISIONS ybd

EXAMPLES:

```
YBDI 10
```


DESCRIBERS:

Describer	Meaning
ybd	Number of spaces between tic marks on y-axis. (Integer > 0, Default = 5)

REMARKS:

1. YBDIVISIONS applies to lower half frame curves only.
2. YBDIVISIONS is ignored for logarithmic y-axes.

YBINTERCEPT

Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for lower half frame curves only.

FORMAT:

YBINTERCEPT ybi

EXAMPLES:

```
YBINT 50
```

DESCRIBERS:

Describer	Meaning
ybi	Location of y-axis on the x-axis. (Real, Default = 0.0)

REMARKS:

YBINTERCEPT applies to lower half frame curves only.

YBGRID LINES

Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on lower half frame curves only.

FORMAT:

$$\text{YBGRID LINES} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

REMARKS:

YBGRID applies to lower half frame curves only.

YBLOG

Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for lower half frame curves only.

FORMAT:

$$\text{YBLOG}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

REMARKS:

1. YBLOG applies to lower half frame curves only.
2. See XLOG.

YBMAX

Maximum Y-Axis Value

Specifies the maximum value on the y-axis for lower half frame curves only.

FORMAT:

YBMAX ymax

EXAMPLES:

```
YBMAX 100
```


DESCRIBERS:

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

REMARKS:

1. YBMAX applies to lower half frame curves only.
2. If YBMAX is not specified, then the maximum value is set to the highest value of y.
3. See the related command YBMIN.

YBMIN

Minimum Y-Axis Value

Specifies the minimum value on the y-axis for lower half frame curves only.

FORMAT:

YBMIN ymin

EXAMPLES:

```
YBMIN 100
```

DESCRIBERS:

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

REMARKS:

1. YBMIN applies to lower half frame curves only.
2. If YBMIN is not specified, then the minimum value is set to the lowest value of y.
3. See the related command YBMAX.

YBTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for lower half frame curves only.

FORMAT:

YBTITLE ytit

EXAMPLES:

```
YBTIT   RIGHT WING LOADS - CASE 3
```


DESCRIBERS:

Describer	Meaning
ytit	Any character string. (Character, Default = blank)

REMARKS:

1. YBTITLE may not be continued to the next command line.
2. YBTITLE applies to lower half frame curves only.

YBVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks on lower half frame curves only.

FORMAT:

YBVALUE PRINT SKIP yvps

EXAMPLES:

```
YBVAL 5
```

DESCRIBERS:

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values. (Integer ≥ 0)

REMARKS:

YBVALUE applies to lower half frame curves only.

YDIVISIONS

Tic Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for whole frame curves only.

FORMAT:

YDIVISIONS yd

EXAMPLES:

```
YDIV 10
```


DESCRIBERS:

Describer	Meaning
yd	Number of spaces between tic marks on the y-axis. (Integer > 0, Default = 5)

REMARKS:

1. YDIVISIONS applies to whole frame curves only and to the commands: LEFT TICS, RIGHT TICS, and XINTERCEPT.
2. YDIVISIONS is ignored for logarithmic y-axes.

YINTERCEPT

Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for whole frame curves only.

FORMAT:

YINTERCEPT yi

EXAMPLES:

```
YINT  50
```

DESCRIBERS:

Describer	Meaning
yi	Location of y-axis on the x-axis. (Real, Default = 0.0)

REMARKS:

YINTERCEPT applies to lower half frame curves only.

YGRID LINES

Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on whole frame curves only.

FORMAT:

$$\text{YGRID LINES} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

REMARKS:

YGRID applies to whole frame curves only.

YLOG

Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for whole frame curves only.

FORMAT:

$$\text{YLOG}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

REMARKS:

1. YLOG applies to whole frame curves only.
2. See XLOG.

YMAX

Maximum Y-Axis Value

Specifies the maximum value on the y-axis.

FORMAT:

YMAX ymax

EXAMPLES:

```
YMAX 100
```


DESCRIBERS:

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

REMARKS:

1. If YMAX is not specified, then the maximum value is set to the highest value of y.
2. See the related command YMIN .

YMIN

Minimum Y-Axis Value

Specifies the minimum value on the y-axis.

FORMAT:

YMIN ymin

EXAMPLES:

```
YMIN 100
```

DESCRIBERS:

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

REMARKS:

1. YMIN applies to all curves.
2. If YMIN is not specified, then the minimum value is set to the lowest value of y.
3. See the related command YMAX.

YPAPER

Paper Size in Y-Direction

Defines the size of the paper in the y-direction.

FORMAT:

YPAPER ysize

EXAMPLES:

```
YPAP 10
```


DESCRIBERS:

Describer	Meaning
ysize	Size of the paper in the y-direction and in inches. (Real, Default = 20.0)

REMARKS:

1. The default paper size is 20 by 20 inches.
2. See the related command XPAPER.

YTAXIS

Y-Axis Plot Control

Controls the plotting of the y-axis on upper half frame curves only.

FORMAT:

$$\text{YTAXIS}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

REMARKS:

YTAXIS applies to upper half frame curves only.

YTDIVISIONS

The Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for upper half frame curves only.

FORMAT:

YTDIVISIONS ytd

EXAMPLES:

```
YTDI 10
```


DESCRIBERS:

Describer	Meaning
ytd	Number of spaces between tic marks on y-axis. (Integer > 0, Default = 5)

REMARKS:

1. YTDIVISIONS applies to upper half frame curves only.
2. YTDIVISIONS is ignored for logarithmic y-axes.

YTGRID LINES

Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on upper half frame curves only.

FORMAT:

$$\text{TYGRID LINES} \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

REMARKS:

YTGRID applies to upper half frame curves only.

YTINTERCEPT

Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for upper half frame curves only.

FORMAT:

YTINTERCEPT yti

EXAMPLES:

```
YTINT  50
```


DESCRIBERS:

Describer	Meaning
yti	Location of y-axis on the x-axis. (Real, Default = 0.0)

REMARKS:

YTINTERCEPT applies to upper half frame curves only.

YTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for whole frame curves only.

FORMAT:

YTITLE ytit

EXAMPLES:

```
YTIT   RIGHT WING LOADS - CASE 3
```

DESCRIBERS:

Describer	Meaning
ytit	Any character string. (Character, Default = blank)

REMARKS:

1. YTITLE may not be continued to the next command line.
2. YTITLE applies to whole frame curves only.

YTLOG

Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for upper half frame curves only.

FORMAT:

$$\text{YT(LOG)}\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

DESCRIBERS:

Describer	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

REMARKS:

1. YTLOG applies to upper half frame curves only.
2. See XLOG.

YTMAX

Maximum Y-Axis Value

Specifies the maximum value on the y-axis for upper half frame curves only.

FORMAT:

YTMAX ymax

EXAMPLES:

```
YTMAX 100
```

DESCRIBERS:

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

REMARKS:

1. YTMAX applies to upper half frame curves only.
2. If YTMAX is not specified, then the maximum value is set to the highest value of y.
3. See the related command YTMIN.

YTMIN

Minimum Y-Axis Value

Specifies the minimum value on the y-axis for upper half frame curves only.

FORMAT:

YTMIN ymin

EXAMPLES:

```
YTMIN 100
```


DESCRIBERS:

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

REMARKS:

1. YTMIN applies to upper half frame curves only.
2. If YTMIN is not specified, then the minimum value is set to the lowest value of y.
3. See the related command YTMAX.

YTTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for upper half frame curves only.

FORMAT:

YTTITLE ytit

EXAMPLES:

```
YTTIT   RIGHT WING LOADS - CASE 3
```

DESCRIBERS:

Describer	Meaning
ytit	Any character string. (Character, Default = blank)

REMARKS:

1. YTTITLE may not be continued to the next command line.
2. YTTITLE applies to upper half frame curves only.

YTVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks on upper half frame curves only.

FORMAT:

YTVALUE PRINT SKIP yvps

EXAMPLES:

```
YTVAL 5
```


DESCRIBERS:

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values. (Integer ≥ 0)

REMARKS:

YTVALUE applies to upper half frame curves only.

YVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks on whole frame curves only.

FORMAT:

YVALUE PRINT SKIP yvps

EXAMPLES:

```
YVAL 5
```

DESCRIBERS:

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values. (Integer ≥ 0)

REMARKS:

YVALUE applies to whole frame curves only.

YVALUE PRINT SKIP

Chapter 6: Parameters and Parameter Applicability Tables

6.1 Parameter Descriptions

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. For more information on the PARAM Bulk Data entry, see PARAM. For more information on the PARAM Case Control command, see PARAM. A complete list of PARAMeter names and their solution support is given in the Parameter Applicability Tables.

NX Nastran allows the specification of PARAM statements as follows:

1. Inside of the input file.

2. In the nast*r*rc (Linux) and nast*r*.rcf (WINDOWS).

3. Parameters can be assigned a user defined keyword. The new keyword can then be used to specify a value for the parameter on the command line or in the nastran resource file. The keywords can be defined in the “nastran.params” file in the architecture directory.

See the section “Parameter Specification” in chapter 3 of the *NX Nastran Installation and Operations Guide* for details.

Parameters and Superelements

The following should be considered when external superelements or part superelements are included in the Bulk Data section of your input file.

- Parameters specified in the Bulk Data, but not in a BEGIN SUPER portion of the Bulk Data, apply only to the residual and not to external or part superelements.

- Parameters specified in a BEGIN SUPER portion of the Bulk Data apply only to that external or part superelement.

- To ensure that parameters apply to the residual and all superelements, you should specify them in the Case Control section above the subcases. For example, specifying the PARAM,POST setting above the subcases will request a consistent output format for the residual and the superelements.

A – Parameters

ACOUT Default = PEAK

ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis (see Performing a Coupled Acoustic-Structural Analysis in the *NX Nastran Acoustics User's Guide*). ACOUT=RMS requests root-mean-square output.

To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

$$\mathrm{dB} = 20 \cdot \log\left(\frac{P}{\mathrm{PREFDB}}\right)$$

ACSYM Default = YES

By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM,ACSYM,NO requests the pre-MSC.Nastran Version 69 formulation which involves no symmetrization and will require more CPU time. See the Formulation of Dynamic Equations in SubDMAP GMA in the *NX Nastran User's Guide*.

If the iterative solver is selected (see the ITER=YES keyword on the NASTRAN statement) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

ADPCON Default = 1.0

Initial penalty values used in contact analysis are calculated automatically by the program and are given by k · SFAC · |ADPCON| where k is a number selected for each slave node based on the diagonal stiffness matrix coefficients that are in the contact region, and SFAC is the value specified by the user in the SFAC field of the BCONP Bulk Data entry. The ADPCON value applies to all the contact regions in the model. During the analysis, if convergence problems are encountered, penalty values are automatically reduced. Still there may be some problems where convergence cannot be achieved with default values. In such cases, analysis may be restarted with a lower value of ADPCON.

In some cases the default penalty values may be low. In such situations, analysis may be restarted with a higher value of ADPCON.

Generally, penalty values are recalculated every time there is a change in stiffness. However, if ADPCON is negative, penalty values are calculated only at the beginning of each subcase, and penalty values are not adjusted during analysis. This is useful if the contact between two elastic bodies is being analyzed.

ADSTAT

Default = YES (SOL 109 and 112 only)

Used in transient analysis data recovery with differential stiffness (see the Case Control Command, STATSUB) to request whether the static solution (displacements, SPC forces, and MPC forces) is to be included (YES) or excluded (NO) in the transient results.

AESDISC

Default = 1.E-8

Tolerance for discarding generalized coordinates in the RITZ method (see AESMETH) which are not linearly independent.

AESMAXIT

Default = 15

Maximum number of iterations for the ITER method (see AESMETH).

AESMETH

Default = SELECT

Solution method for static aeroelastic analysis.

SELECT	selects the DIRECT method on models with less than 5000 DOF in the solution set; otherwise selects AUTO.
AUTO	selects the reduced basis method for an approximate solution, which is used as starting vectors for an ITER solution.
DIRECT	selects the direct solution.
RITZ	selects the reduced basis approximate solution.
ITER	selects the iterative solution.

AESRNDM

Default = 2

Number of random vectors to use as generalized functions in the RITZ method (see AESMETH).

AESTOL Default = 1.E-10

Convergence criteria for the iterative solver.

AFNORM Default = 1.0

Scale factor used in AF (amplitude-frequency) normalization. For additional information, see Mode Shape Normalization in the *Basic Dynamic Analysis User's Guide*.

AFZERO Default = 1.0 hertz

Frequency threshold in units of hertz used by the SOL 401 cyclic and Fourier subcase types to determine if a mode is a rigid body mode when AF normalization is requested. The software considers a mode to be a rigid body mode if its frequency is below the value of AFZERO. For the modes considered as rigid body modes, the software sets ω = 1 when computing the AF normalization factor. See "Subcase Types" in the *Multi-Step Nonlinear User's Guide*.

AGGPCH Default = NO

Requests the output of the fluid-structure coupling matrix AGG to the punch file if YES. If NO, the fluid-structure coupling matrix AGG is not written.

ALPHA1, ALPHA2 Default = (0.0,0.0)

In frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to complex zero, then Rayleigh's damping is added to the viscous damping. ALPHA1 is the complex scale factor applied to the mass matrix and ALPHA2 to the stiffness matrix. The scale factors are applied to the d-set or h-set matrices. See Formulation of Dynamic Equations in SubDMAP GMA in the *NX Nastran User's Guide*.

$[B'] = [B] + \text{ALPHA1} \cdot [M] + \text{ALPHA2} \cdot [k]$

ALTRED Default = NO

ALTRED=YES requests the alternate stiffness and load reduction technique for superelement analysis in SOLs 101 and 114. This technique is described in Static Condensation in SubDMAPs SEKR and SEMR2 and Data Recovery Operations in SubDMAP SEDISP in the *NX Nastran User's Guide*.

AMPCZ Default = 1.0e-6

Zero tolerance for pivot elements in the AUTOMPC process. You can optionally increase the tolerance if AUTOMPC has problems resolving redundant constraint equations.

ASCOUP Default = YES

In coupled fluid-structure analysis, if PARAM,ASCOUP,YES is specified in or omitted from the input file, and the A2GG case control command is also omitted from the input file, coupling for the stiffness and mass is computed. If PARAM,ASCOUP,NO is specified and A2GG is omitted, the coupling is not computed.

When ASCOUP and A2GG are both specified, direct input acoustic/fluid-structure coupling matrices can be added to the computed coupling matrix or the computed coupling matrix can be ignored altogether in favor of direct input matrices. In either case, the coupling matrices can be scaled by:

- Specifying one or both of the CA1 and CA2 parameters.
- Specifying in-line scale factors using the A2GG case control command.
- Specifying both in-line scale factors using the A2GG case control command and one or both of the CA1 and CA2 parameters.

If PARAM,ASCOUP,YES and A2GG are both specified, the total acoustic/fluid-structure coupling matrix is:

$$\left[A_{jj}\right]=CA1\cdot\left[A_{jj}^{x}\right]+CA2\cdot\left[A_{jj}^{2}\right]$$

where [A^x_{jj}] is the computed acoustic/fluid-structure coupling matrix and [A^2_{jj}] is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

If PARAM,ASCOUP,NO and A2GG are both specified, the computed coupling matrix is ignored and the total acoustic/fluid coupling matrix is:

$$\left[A_{jj}\right]=CA2\cdot\left[A_{jj}^{2}\right]$$

where [A^2_{jj}] is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

For additional information, see Performing a Coupled Acoustic-Structural Analysis in the *NX Nastran Acoustics User's Guide*

ASING Default = 0

ASING specifies the action to take when singularities (null rows and columns) exist in the [K_{ll}] in statics). If ASING=-1, then a User Fatal Message will result.

If ASING=0 (the default), singularities are removed by appropriate techniques depending on the type of solution being performed. The procedures used are described in Formulation of Dynamic Equations in SubDMAP GMA and Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS in the *NX Nastran User's Guide*.

AUNITS Default = 1.0

AUNITS is used in SOL 144 to convert accelerations specified in units of gravity on the TRIM Bulk Data entry to units of distance per time squared.

AUTOADJ Default = YES

In SOL 200, a value of YES will automatically select the direct or the adjoint method for sensitivity analysis based on the performance criteria. AUTOADJ=NO requests that the adjoint method not be selected for sensitivity analysis and the direct method is enforced. The default should be preferred in all cases. However, the presence of this parameter allows investigation of the alternative of using direct methods in place of adjoint methods in the sensitivity calculations.

AUTOMPC Default = NO

AUTOMPC=YES specifies that the software automatically selects the m-set dofs rather than use the m-set values as specified on MPC or RIGID element cards (RBE*, RBAR, RROD, etc.) in the bulk data definition except as noted below.

This option relieves the need to carefully define rigid elements (or MPCs) so as to not have a conflict of the m-sets between elements. In addition, any redundant constraints will be eliminated.

This option is not available in some circumstances and will be automatically set to NO for a design optimization solution (SOL=200) with DVGRID data.

The AUTOMPC option is not recommended for use in models with RSSCON elements connected to CPENTA elements.

If an RBE3 element contains UM information on the m-set data, those dofs will be used in the m-set. SPOINTS in MPC equations will always be made part of the m-set.

When using this option, it is possible to specify that selected dofs must not be made part of the m-set. This is accomplished by defining these dofs on USET/USET1 bulk data cards making them part of the U5 set. If specified, the UM information on RBE3 elements will be ignored.

An option also exists to specify that selected dof must be made part of the m-set. Again, using USET/USET1 bulk data cards, but making them part of the U4 set.

AUTOSEEL Default = NO

Requests that elements connected totally to external (boundary) grids (b-set and c-set) of super elements be included in the super element rather than the residual structure. AUTOSEEL = NO (default) means such elements are included in the residual structure.AUTOSEEL = YES means they are included in the super element.

AUTOSPC Default = YES (In all SOLs, except 106, 129, 153, and 159)

Default = NO (In SOLs 106, 129, 153, and 159)

AUTOSPC specifies the action to take when singularities exist in the stiffness matrix [K_{gg}]. AUTOSPC=YES means that singularities will be constrained automatically. AUTOSPC=NO means that singularities will not be constrained. If AUTOSPC=NO is set, then the user should take extra caution analyzing the results of the grid point singularity table and epsilon.

See Constraint and Mechanism Problem Identification in SubDMAP SEKR in the *NX Nastran User's Guide* for details of singularity identification and constraint. Singularity ratios smaller than PARAM,EPPRT (default = 1.E-8) are listed as potentially singular. If PARAM,AUTOSPC has the value YES, identified singularities with a ratio smaller than PARAM,EPZERO (default = 1.E-8) will be automatically constrained with single-point constraints. If PARAM,EPPRT has the same value as PARAM,EPZERO (the default case), all singularities are listed. If PARAM,EPPRT is larger than PARAM,EPZERO, the printout of singularity ratios equal to exactly zero is suppressed. If PARAM,PRGPST is set to NO (default is YES), the printout of singularities is suppressed, except when singularities are not going to be removed. If PARAM,SPCGEN is set to 1 (default = 0), the automatically generated SPCs are placed in SPCi Bulk Data entry format on the PUNCH file.

AUTOSPC provides the correct action for Superelements in all contexts. It is ignored for the residual structure in SOLs 106, 129, 153 and 159 when the solution is either material nonlinear or geometric nonlinear. PARAM, AUTOSPCR, not AUTOSPC, is used for the o-set (omitted set) in the residual structure in SOLs 106 and 153.

AUTOSPCR Default = NO (SOLs 106 and 153 only)

AUTOSPCR specifies the action to take when singularities exist in the linear stiffness matrix of the residual structure after multipoint constraints are processed. AUTOSPCR=YES means that singularities will be constrained and AUTOSPCR=NO means they will not be constrained. It is recommended that all degrees-of-freedom attached to nonlinear elements be specified on ASETi entries. Parameters EPPRT, EPZERO, PRGPST, and SPCGEN may be used with AUTOSPCR.

AUTOSPRT Default = YES

By default, models that are not fully constrained and involve residual vectors, constraint modes, or attachment modes will be automatically supported as long as F1 ≤ 0.0 on the EIGR bulk data entry or V1 ≤ 0.0 on the EIGRL bulk data entry (for SOLs 103 and 110, this only applies if residual vectors are requested). The auto-SUPORT capability may be deactivated by specifying PARAM,AUTOSPRT,NO or F1 / V1 > 0.0. The parameter FZERO is the maximum frequency (Hz) assumed for a rigid body mode. FZERO is used by the auto-SUPORT capability to extract rigid body frequencies. For direct solutions with SOLs 108 and 109, massless mechanisms will be sought and supported if there are no SUPORT entries defined.

For a SOL 103, 107, 108, 109, 110, 111, or 112 run with a model that is not fully constrained, the model will be supported by default in a static subcase that computes differential stiffness. To deactivate this functionality, specify PARAM,AUTOSPRT,NO.

B – Parameters

BAILOUT

Default = 0

In a normal modes analysis, Sturm sequence number techniques are used to check the agreement with the number of modes actually computed in the range. If a disagreement occurs, the computed modes are likely incorrect and the run is terminated with a fatal message by default. PARAM, BAILOUT, -1 may be used to allow the solution to proceed after the Sturm sequence check error.

See also MAXRATIO.

BETA

Default = (0.333333,0.0)

BETA specifies the transient integration control factor for the Newmark method in SOLs 129 and 159. The stability limits are $0.25 \le \beta < 0.5$.

BIGER, BIGER1, BIGER2

Default = 0.0

See S1.

BOLTFACT

Real ≥ 0.0, default = 1.0E7

Factor used to reduce the bolt stiffness during the first phase of a bolt preload analysis. If BOLTFACT = 0.0, the axial stiffness of the bolts is zero during the first phase of a bolt preload analysis. Generally, BOLTFACT = 0.0 produces better results. However, a singular stiffness matrix will occur if eliminating the axial stiffness of the bolts cause the structure to be unstable.

BSHDAMP

Default = DIFF

Determines if the PBUSH/PBUSHT bulk entry fields GE2-GE6/TGEID2-TGEID6 are used.

SAME: The fields GE2-GE6/TGEID2-TGEID6 are ignored, and only fields GE1/TGEID1 are considered.

DIFF: The fields GE2-GE6/TGEID2-TGEID6 are used. If any of the fields GE2-GE6 and TGEID2-TGEID6 are defined, then all PBUSH and PBUSHT bulk entries are assumed to have all GE1-GE6 and TGEID1-TGEID6 fields defined, and any blank fields are considered a zero value.

BUCKLE Default = -1

BUCKLE=1 requests a nonlinear buckling analysis in a restart run of SOLs 106 or 153. See the *NX Nastran Handbook for Nonlinear Analysis*.

BUCKLE=2 requests buckling in a SOL 106 cold start run.

C – Parameters

CA1, CA2 Default =(1.0, 0.0)

CA1 and CA2 specify factors for scaling the total acoustic/fluid-structure coupling matrix. The total acoustic/fluid-structure coupling matrix is:

$$[A_{jj}] = CA1 \cdot [A^{x}_{jj}] + CA2 \cdot [A^{2}_{jj}]$$

where $[A^{x}_{jj}]$ is the computed acoustic/fluid-structure coupling matrix and $[A^{2}_{jj}]$ is the direct input acoustic/fluid-structure coupling matrix specified using the A2GG case control command. CA1 and CA2 are only effective if A2GG is specified in the case control section.

CB1, CB2 Default =(1.0, 0.0)

CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is:

$$[B_{jj}] = \mathrm{CB1} \cdot [B^{x}_{jj}] + \mathrm{CB2} \cdot [B^{2}_{jj}]$$

where B^{2}_{jj} is selected using the B2GG case control command and B^{x}_{jj} comes from CDAMPi or CVISC element bulk entries. These parameters are effective only if B2GG is selected in the case control section.

CDIF Default = YES for shape optimization with or without property optimization.

Default = NO for property optimization only.

CDIF may be used to override the default finite difference scheme used in the calculation of pseudoloads in SOL 200. PARAM,CDIF,YES forces the selection of the central difference scheme used in the semianalytic approach regardless of the type of optimization requested. PARAM,CDIF,NO forces the selection of the forward difference scheme.

CDITER Default = 0

See CDPRT.

CDPCH Default = NO

See CDPRT.

CDPRT Default = YES

If CDITER>0, perform constrained displacement iterations in SOL 101. The value is the maximum number of iterations. If CDPRT=YES, print those negative displacements and tension forces which do not satisfy constraints. If CDPCH=YES, punch DMIG CDSHUT entries for final state; by default all gaps are closed. These can be used for initial conditions for a restart. Potential contact points must be specified on the SUPORTi entries. The SUPORTi points must be in the residual structure. Optional DMIG entries to define the initial shut vector may be specified. Degrees-of-freedom that are specified on the SUPORT entry and have a value of 1.0 defined on the DMIG,CDSHUT entry will be considered closed initially. If the DMIG,CDSHUT entry is not supplied, then all degrees-of-freedom specified on the SUPORT entries will be considered shut initially. A fatal message will be issued if this parameter is used and PARAM,INREL is specified.

CHECKOUT Default = NO (structured solutions only)

CHECKOUT=YES requests a model checkout in SOLs 101 through 200. See “Geometry Processing in SubDMAP PHASEO” in the *NX Nastran User’s Guide*. The run will terminate prior to phase 1 of superelement analysis. The PARAM,POST options are also available with PARAM,CHECKOUT,YES. The following options and their user parameters are also available with PARAM,CHECKOUT,YES,

1. PARAM,PRTGPL,YES

   Prints a list of external grid and scalar point numbers in internal sort. It also lists external grid and scalar point numbers along with the corresponding sequence numbers in internal sort. The sequence numbers are defined as (1000*external number) and will reflect any user-requested resequencing.

2. PARAM,PRTEQXIN,YES

   Prints a list of external and internal grid and scalar numbers in external sort. It also lists external grid and scalar numbers with

the corresponding coded SIL number in external sort. The coded SIL numbers are defined as:

$$10 \cdot \text{SIL} + \begin{cases} 1 \text{ for grid point} \\ 2 \text{ for scalar point} \end{cases}$$

The SIL numbers correspond to degrees-of-freedom, i.e., one SIL number for scalar point and six SIL numbers for a grid point.

3. PARAM,PRTGPDT,YES

   Prints, for each grid and scalar point, the following information in the internal sort:

   - Coordinate system ID in which grid point geometry is defined (ID=-1 for scalar points).
   - Spatial location of grid points in the “CP” coordinate system. For scalar points, all entries are zero.
   - Coordinate system ID for grid point displacements, forces, and constraints (ID=0 for scalar points).
   - Permanent single-point constraints defined on GRID Bulk Data entries. A zero is entered for scalar points.

4. PARAM,PRTCSTM,YES

   Prints for each coordinate system type the transformation matrix from the global to the basic coordinate system, and the origin of the indicated coordinate system in the basic coordinate system. Coordinate system types are: 1 = rectangular; 2 = cylindrical; 3 = spherical.

5. PARAM,PRTBGPDT,YES

   Prints all grid and scalar points listed in internal sort with their x, y, and z coordinates in the basic coordinate system. In addition, the coordinate system ID for grid point displacements, forces, and constraints is indicated for each grid point (ID=-1 for scalar points). The x, y, and z coordinates of scalar points are zero.

6. PARAM,PRTGPTT,YES

   Prints, for each temperature load set, information on element and grid point temperatures.

7. PARAM,PRTMGG,YES

   Prints the g-size mass matrix labeled by grid point/degree-of-freedom.

8. PARAM,PRTPG,YES

   Prints the g-size load vectors labeled by grid point/degree-of-freedom.

9. The summation of forces and moments of applied loads in the basic coordinate system is automatically output for each loading condition requested in the Case Control Section. Related parameters are GPECT, PROUT, and EST.

CK1, CK2 Default =(1.0, 0.0)

CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K_{jj}^{x}] = \mathrm{CK1} \cdot [K_{jj}^{z}] + \mathrm{CK2} \cdot [K_{jj}^{2}]$$

where $[K^2_{jj}]$ is selected via the Case Control command K2GG and $[K^z_{jj}]$ is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in Case Control. A related parameter is CK3. Note that stresses and element forces are not factored by CK1, and must be adjusted manually.

CK3 Default = (1.0, 0.0)

CK3 specifies a factor for the stiffness derived from GENEL Bulk Data entries. The total stiffness matrix is

$$[K_{jj}] = [K_{jj}^{x}] + \mathrm{CK3} \cdot [K_{jj}^{y}]$$

where $[K^y_{jj}]$ comes from the GENEL Bulk Data entries and $[K^x_{jj}]$ is derived using PARAMs CK1 and CK2. CK3 is effective only if GENEL entries are defined. Related parameters include CK1 and CK2.

CK41, CK42 Default =(1.0, 0.0)

CK41 and CK42 specify factors for the total structural damping matrix. The total structural damping matrix is

$$\left[K_{jj}^{4,x}\right] = CK41 \cdot \left[K_{jj}^{4,z}\right] + CK42 \cdot \left[K_{jj}^{4,2}\right]$$

where $[K^{4,2}_{jj}]$ is selected via the Case Control command K42GG and $[K^{4,z}_{jj}]$ is generated from the stiffness of structural element (e.g., CBAR) entries in the Bulk Data times the structural damping coefficient GE on material entries (e.g. MAT1). These are effective only if K42GG is selected in Case Control. Note that stresses and element forces are not factored by CK41, and must be adjusted manually.

CLOSE Default = 1.0

See SCRSPEC.

CM1, CM2 Default = (1.0, 0.0)

CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] = CM1 \cdot \left[M_{jj}^{x}\right] + CM2 \cdot \left[M_{jj}^{2}\right]$$

where $[M^2_{jj}]$ is selected via the Case Control command M2GG and $[M^x_{jj}]$ is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

CNSTRT No longer required.

CNTASET Default = NO

When contact conditions are defined in SOL 101, CNTASET determines if a static condensation (A-set reduction) occurs with the contact degrees-of-freedom.

When CNTASET=YES, a static condensation is performed to reduce the full K_{gg} matrix to the K_{aa} matrix containing only the contact degrees-of-freedom. The contact iterations are then performed using this reduced matrix.

When CNTASET=NO, the static condensation of contact degrees-of-freedom does not occur.

COLPHEXA Default=NO

Specify YES to allow collapsed CHEXA elements for crack propagation analysis in SOL 401. If PARAM,COLPHEXA,YES is not specified, collapsed CHEXA elements are disallowed by the input file checks. Note that the collapsed CHEXA is not supported in a glue or contact region.

COMPMATT Default=NO

During SOL 106 solutions, composite materials compute temperature-dependent properties for the plies only at the reference temperature given on the PCOMP or PCOMPG bulk entries. The resulting ply properties are smeared and used for all load steps, regardless of whether the temperature is changing through application of thermal loads.If the parameter equals YES, the temperature-dependent properties for the plies are updated and smeared at the current temperature for each load step.This parameter only applies to SOL 106, and CQUAD4/CTRIA3 elements only.

CONFAC Default = 1.E-5

In superelement analysis, CONFAC specifies the tolerance factor used in checking the congruence of the location and displacement coordinate systems of the boundary points between image superelements and their primaries (see the Bulk Data entry, CSUPER). Specification of this parameter is recommended instead of DIAG 37 (DIAG 37 ignores User Fatal Messages 4277 and 4278).

CORROPT Default = NO

This parameter is valid only for SOL 103 when used with the Simcenter FE Model Update product. The documentation for this product is included in the Help.

NO: Output for the Simcenter FE Model Update product is deactivated.

YES: Output for the Simcenter FE Model Update product is activated.

COUPMASS Default = -1

COUPMASS>0 requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability, as listed in “Element Summary – Small Strain Elements” in the *NX Nastran Element Library*. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CONROD, CHEXA, CPENTA, CPYRAM, CQUAD4, CQUAD8, CQUADR, CQUADX4, CQUADX8, CRAC2D, CRAC3D, CROD, CTETRA, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, CTUBE. A negative value (the default) causes the generation of lumped mass matrices (translational components only) for all of the above elements.

CBEND elements are always generated with coupled mass and are not affected by COUPMASS.

CP1, CP2 Default = (1.0, 0.0)

The load vectors are generated from the equation

$$\{P_j\} = \mathrm{CP1} \cdot \left\{ P_j^x \right\} + \mathrm{CP2} \cdot \left\{ P_j^2 \right\}$$

where

$\{P^2_j\}$ is selected via the Case Control command P2G, and $\{P^x_j\}$ comes from Bulk Data static load entries.

CURV Default = -1

PARAM,CURV,1 requests that the CTRIA3 and CQUAD4 element stress and/or strain output be computed in a material coordinate system (normal output is in the element or basic coordinate system) and/or to interpolate it to grid points. (CQUAD4 element corner stress output is not supported.)

The integer parameter OG controls the calculation of stress and/or strain data at grid points. If OG is set to -1, the calculation for stresses and/or strain data at grid points is not performed. The default value of zero provides the calculation of these quantities at those grid points to which the selected elements connect.

User parameters S1G, S1M, S1AG, and S1AM, set to 1, request the printout of stresses at grid points, stresses in the material coordinate system, strains at grid points and strains in the material coordinate system, respectively.

The integer parameter OUTOPT may be set in accordance with the below options to select print, punch, and/or plotter output for stress and/or strain data that are computed in user-defined material coordinate systems.

OUTOPT Value	Description
0	Default-standard NX Nastran device codes are used.
1	Print only
2	Plot only
4	Punch only

The above values may be combined additively to select two or more forms of output. For example, OUTOPT=6 requests both plot and punch output. Related parameters include BIGER, CURVPLOT, DOPT, NUMOUT, NINTPTS, S1G, S1M.

For stress and/or strain/curvature output in a user-defined material coordinate system MCSID must be defined on MAT1 and MAT2 Bulk Data entries. The values of MCSID reference CORDiR, CORDiC, and CORDiS Bulk Data entries. A value of zero for MCSID does not imply the basic coordinate and will eliminate all elements which reference the MATi from the subject calculations.

1. If these data are requested at the element centers, the program will compute the unit vector $\vec{i}_m$ along the T1 or x-axis of the material coordinate system, and compare $\left|\vec{n} \cdot \vec{i}_m\right|$ for each element that references the material coordinate system, where n is the normal to the surface of the element. If $\left|\vec{n} \cdot \vec{i}_m\right|^2 \geq 4$ the projection of the y-axis on the surface of the element is taken as the reference axis. Otherwise, the projection of the x-axis on the surface of the element is taken as the reference axis. The angle between the x-axis of the element coordinate system and the projection of the selected reference axis of the material coordinate system is used to transform the stress and/or strain data into the material coordinate system at the element centers.

2. If, on the other hand, the user requests these data at the grid points to which the elements connect, the program will interpolate the results from (a) to the grid points to which the elements connect. The parameter NINTPTS = N, the stress and/or strain data at the N closest element centers to the grid point in question will be used in the interpolation. The program may include more

that N points in the interpolation if the distance of other element centers is not more than 10% greater than the closest N element centers.

The following specifies the output headings for stresses and/or strains in the material coordinate system.

Element stresses (PARAM,S1M,1) are:

1. Available in CQUAD4 and CTRIA3 elements.
2. In page headings:

   STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)

   STRESSES IN TRIANGULAR ELEMENTS (CTRIA3)
3. Under the column FIBER DISTANCE:

   Z1 is replaced by MCSID.

   Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point stresses (PARAM,S1G,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements.
2. In page heading:

   STRESSES AT GRID POINTS
3. Under the $\left\{\begin{matrix} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{matrix}\right\}$ column:

   Z1 is replaced by MCSID:

   Z2 = A+10*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

Element strains (PARAM,S1AM,1) are:

1. Available for CQUAD4 and CTRIA3 elements.
2. In page headings:

STRAINS IN QUADRILATERAL ELEMENTS (CQUAD4)

STRAINS IN TRIANGULAR ELEMENTS (CTRIA3)

3. Under the column FIBER DISTANCE:

   Z1 is replaced by MCSID.

   Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point strains (PARAM,S1AG,1 and PARAM,OG,1):

1. Available for CQUAD4 and CTRIA3 elements.

2. In page heading:

   STRAINS AND CURVATURES AT GRID POINTS

3. Under the $\left\{ \begin{array}{c} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{array} \right\}$ column:

   Z1 is replaced by MCSID.

   Z2 = A+10*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

CURVPLOT Default=-1

PARAM,CURVPLOT,1 requests x-y (or curve) plots whose abscissas are a sequence of grid points and whose ordinates may be displacements, loads, SPC forces, or grid point stresses. To obtain stress plots, set the CURV parameter to +1. The default for DOPT is the length between grid points

Specify the XYOUTPUT Case Control command in the usual manner, replacing the point ID with the SID of SET1 Bulk Data entries.

The SET1 Bulk Data entries must contain unique SIDs for each set of grid points to be plotted.

User requests for xy-plots of output quantities appear in the Case Control Section in the standard form. For example,

```
.
.
.
OUTPUT(XYOUT)
.
.
XYPLOT DISP 1/4(T3)
.
.
XYPLOT SPCF 2/5(T1)
.
.
BEGIN BULK
```

The first XYPLOT command will produce an xy-plot from the displacement output of subcase 1. The abscissa of the curve will reflect the grid point IDs listed on the SET1 entry with a SID of 4, and the ordinate will reflect the T3 component of displacement at these grid points. The second XYPLOT command will produce an xy-plot whose ordinates are the T1 components of the forces of constraint in subcase 2 at the grid points listed on the SET1 entry with a SID of 5.

The user has some degree of control over the scaling of the abscissas on these xy-plots. This control is exercised through the parameter DOPT on a PARAM Bulk Data entry. The legal values of this parameter provide the following scaling options for the abscissas.

Value of DOPT	Scaling for Abscissa
0 (default)	$\|\|g_j - g_i\|\|$
1	$\|x_j - x_i\|$
2	$\|y_j - y_i\|$
3	$\|z_j - z_i\|$
4	1.

Thus, the default value of DOPT will place the first grid point listed on the referenced SET1 card at the origin, and subsequent grid points will be located along the abscissa at intervals proportional to the distance between that grid point and its predecessor. Values of DOPT equal to 1, 2, or 3 will scale the abscissa so that the interval between adjacent grid points is proportional to the difference in the X, the Y, and the Z components of the subject grid points respectively. DOPT = 4 will space the grid points equally along the abscissa.

D – Parameters

DBALL Default = DBAL

By default, all data to be stored on the database for restart purposes will be located on the DBALL database set DBset. These parameters permit the storage of some data blocks on DBsets other than DBALL, which are defined by the user and specified on the INIT File Management statement. Any or all of these parameters may be set to SCRATCH in order to reduce overall disk space usage; e.g., PARAM,DBUP,SCRATCH or PARAM,DBALL,SCRATCH. However, automatic restarts will be less efficient because data normally assigned to a permanent DBset will have to be recomputed.

A unique value for each superelement may be specified in the Case Control Section for the parameters DBALL, DBDN, DBRCV, and DBUP. Certain DBsets may be taken offline depending on which phase of superelement analysis is being performed (see “Summary of Solution Sequence Operations” in the *NX Nastran User’s Guide*). PARAM,DBALL specifies the default value for parameters DBDN, DBUP, and DBRCV.

The DBDN DBset contains data blocks necessary for “downstream” processing. For example, the stiffness, mass, damping, and static loads matrices that have been reduced to the boundary of the superelement are stored in this DBset.

The DBRCV DBset contains data blocks that must be online during the first pass through data recovery (Phase 3). These data blocks are used to recover the total displacement vector u_g of the superelement. This operation is performed by the SSG3 and SDR1 modules. On subsequent data recovery restarts, this DBset may be taken offline. Its default is determined from the value of DBUP.

The DBUP DBset contains data blocks necessary for “upstream” processing. For example, the geometry and property tables along with the stiffness, mass, damping, and static loads matrices related to the interior grid points of the superelement are stored in this DBset. These matrices and tables must be online during the reduction (Phase 1) and data recovery (Phase 3) of the superelement.

The IFP DBset contains data blocks that are required for all phases of the analysis. These data blocks are related to the entire model; examples are Bulk Data, superelement map, IFP module outputs, and resequenced grid points. This DBset must be online for all runs.

DBCCONV Default = XL

See PARAM, POST, 0.

DBCDIAG Default = 0

See PARAM, POST, 0.

DBCOVWRT Default = YES

See PARAM, POST.

DBDICT Default = -1

Controls the printout of the database directory at the beginning and end of the run. See the DBDICT file management statement description. If DBDICT = 0, then the database directory will be printed at the start of the run. If DBDICT=1, then the directory will be printed at the end of the run. If DBDICT ≥ 2, then it will be printed at the beginning and end of the run. If DBDICT = -1 (the default), the directory is not printed at the beginning or end of the run.

If multiple versions and/or projects exist on the database, then the parameters DBDRPRJ and DBDRVER allow the user to select the desired project and version, respectively. The appropriate values may be found in the Project/Version Table that is printed upon restart. If DBDRVER = 0 (or DBDRPRJ=0), then the current version (or project) is selected. If DBDRPRJ=-1 (or DBDRVER = -1), then all projects (or versions) are selected.

DBDN Default = value of PARAM,DBALL.

See DBALL.

DBDRPRJ Default = 0

Specifies the desired project-identification number. See DBDICT.

DBDRVER Default = 0

Specifies the desired version-identification number. See DBDICT.

DBRCV Default=value of PARAM,DBUP.

See DBALL.

DBUP

Default=value of PARAM,DBALL.

See DBALL.

DDRMM

Default = 0

By default, the matrix method of data recovery is used in the modal transient and frequency response solutions. DDRMM=-1 will force calculation of complete g-set solution vectors by the mode displacement method. See ”Dynamic Data Recovery in Modal Frequency Response Analysis” in the *Basic Dynamic Analysis User's Guide*.

DESPCH

Default = 0

DESPCH specifies in SOL 200 when the optimized (updated) Bulk Data entries are written to the PUNCH file. Currently, all the property entries, material entries, and connectivity entries that can be designed and DESVAR, DRESP1, and GRID entries can be written.

DESPCH < 0 never.

DESPCH = 0 at the last design cycle only. (Default)

DESPCH > 0 at every design cycle that is a multiple of DESPCH and the last design cycle. For example, if n=2 and the maximum number of design cycles is 5 (DESMAX=5 on the DOPTPRM entry), then, DESVAR, DRESP1 (when modified due to mode tracking: see DESPCH1 below), and GRID entries at design cycle 2, 4, and 5 are written in the punch file.

DESPCH1

Default=6

DESPCH1 specifies in SOL 200 the amount of data to be written to the.pch file. A positive DESPCH1 value requests large field formats while a negative value requests small field formats. Any updated EIGN or FREQ type DRESP1 written to the punch file due to mode tracking will be in small field format regardless of the sign of DESPCH1. No DRESP1 data will be written with the DESPCH1 request unless mode tracking has resulted in a change in such data. Note that the ECHO command provides more options for bulk data output into the punch file. For a shape optimization job, if DESPCH1<>0, the updated GRID entries of the whole model will be written in the.pch file.

Descriptions of various DESPCH1 values are given below:

DESPCH1 = 0, writes no data.

DESPCH1 = ± 1, writes the property entries that are designed.

DESPCH1 = ± 2, writes all the property entries of a given type when one property of that type is designed.

DESPCH1 = ± 4, writes DESVAR and DRESP1 entries.

DESPCH1 = ± n, writes combine quantities by summing the DESPCH1 values. For example, n=1+4=5 requests writing all the designed property entries, DESVAR andDRESP1 entries to the.pch file.

DFREQ

Default = 10^{-5}

DFREQ specifies the threshold for the elimination of duplicate frequencies on all FREQi Bulk Data entries. Two frequencies, f_1 and f_2, are considered duplicated if:

$$\left|f_1 - f_2\right| < \mathrm{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|$$

where f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

DIGITS

Default = 15

DIGITS controls the number of digits of precision for matrix data written to an OP4 file for the EXTSEOUT and MBDEXPORT case control commands. DIGITS refers to the number of digits after the decimal point. Thus, for the default value of 15, there are actually 16 digits of precision because of the one digit that precedes the decimal point.

DOF123

Default = 0

Valid only when PARAM,CORROPT,YES exists.

0: Nodes in Display Set will have all DOFs (123456) available in back expansion matrix (GOA).

1: Nodes in Display Set will have translational DOFs (123) available in back expansion matrix (GOA). If SPOINTs are present in the Display Set, DOF123 is forced to 0 so that all DOFs are made available.

DOPT

Default = 0

See CURVPLOT.

DPEPS Default = 1.0E-4

In SOL 200, if any difference between the property value on the property entries and the value calculated from the design variable values on the DESVAR entry (through DVPRELi relations) is greater than DPEPS, the design model values override the analysis values. If all the differences are less than DPEPS, analysis results from a previous run are accepted in a subsequent sensitivity/optimization task, thereby avoiding a reanalysis. The PTOL parameter on the DOPTPRM entry is a related parameter that checks the maximum difference.

DPREONLY Default = NO

DPREONLY is used to signal the SOL 187 analysis to stop execution before the Fortran NAVSHOK program is started. This option is used when you want to run the Fortran program outside the Nastran job.

If PARAM,DPREONLY,YES is specified, the DDAM solution will stop after the OUTPUT4 files are written, but just before the Fortran program is executed.

If PARAM,DPREONLY,NO is specified, the DDAM solution will start the Fortran NAVSHOK program once the OUTPUT4 files have been created.

DSNOKD Default = 0.0

DSNOKD specifies a scale factor to the differential stiffness matrix in buckling design sensitivity analysis. If DSNOKD>0.0, the effect of the differential stiffness matrix is included in buckling the design sensitivity analysis.

If PARAM,DSNOKD > 0 is specified in SOL 105, under the original Design Sensitivity Analysis (DSA), the differential stiffness sensitivity calculation is performed under the assumption that all the displacements are enforced; i.e., the change in the stiffness matrix due to the changes in the displacements are not computed. Therefore, PARAM,DSNOKD,0.0 is recommended in SOL 105. If PARAM,DSNOKD,1.0 is specified in SOL 200, the differential stiffness sensitivity calculation is performed more accurately; i.e., the change in the stiffness matrix due to the changes in the displacements are computed. However, the calculation is more expensive than with PARAM,DSNOKD,0.0.

Non-zero values of PARAM,DSNOKD cannot be used in SOL 200 with multiple buckling design subcases unless each subcase contains the same STATSUB command.

DSZERO Default = 0.0

DSZERO specifies the minimum absolute value for the printout of design sensitivities.

DYNSPCF Default = NEW (structured solutions only)

PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPCForce calculations for the linear dynamic solutions: SOLs 103, 107 through 112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPCForce results obtained in versions prior to MSC.Nastran Version 68.

E – Parameters

ELITASPC Default = NO

Set to YES to perform the autospc in the element iterative solver. Normally the element iterative solver does not perform an autospc function as it is usually not necessary. For solid elements, the rotational dofs are eliminated directly. If K6ROT is specified for linear shell elements, there is no issue either. But for CQUAD8 and CTRIA6 elements and possibly other special cases, the autospc function is required. The drawback of this option is that it requires the assembly of the KGG matrix which is used in the autospc and this can have a significant impact on performance. This parameter will also generate the rigid body mass properties.

If MPCFORCES output is requested, this parameter will automatically be set to YES since a partition of the assembled stiffness matrix is needed for the calculation of the mpc forces.

EPPRT Default = 1.E-8

Specifies the maximum value of singularities that will be printed. See AUTOSPC.

EPZERO Default = 1.E-8

Specifies the minimum value that indicates a singularity. See AUTOSPC.

ERROR Default = -1

If ERROR is set to 0, errors in superelement generation or assembly (e.g., a singular matrix, a mechanism, a nonpositive definite matrix, no stiffness, or all upstream superelements not available) will not terminate the run. The remaining computations for that superelement will be skipped, and additional superelements will be attempted. The default value causes a User Fatal Message to terminate the run. A related parameter is MAXRATIO.

EST Default = 2

EST = +1 requests that the measure (length, area or volume) and volume be computed and printed for each element in the model, or in the case of superelement analysis, for the SEMG-selected superelements. The default value of 2 causes this data to be computed but not printed, making it available for element strain energy density calculations. If EST is set to -1, the calculation is skipped, and the density data will not be calculated.

EXTBEMI Default = 0

Specify PARAM,EXTBEMI,1 to accept and convert information from boundary element method software.

Specify PARAM,EXTBEMI,0 (default) to not accept and convert information from boundary element method software.

EXTBEMO Default = 0

Specify PARAM,EXTBEMO,1 to output information to boundary element method software.

Specify PARAM,EXTBEMO,0 (default) to not output information to boundary element method software.

EXTDR Default = NO. See EXTOUT

EXTDROUT Default = NO. See EXTOUT

EXTDRUNT Default = 31. See EXTOUT

EXTOUT Default = NO

When inputting the matrices for a reduced external superelement (SEBULK, CSUPER), there are four options that can be selected using the parameter EXTOUT. EXTOUT must be placed in the Case Control Section above any subcase or in the main Bulk Data Section. The options for Step 1 (see the table at the end of this discussion) are as follows:

If EXTOUT is set to MATRIXDB, the reduced structural matrices and loading are stored on the database.

If EXTOUT is set to DMIGDB, the reduced structural matrices and loading are stored on the database in a format which allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model.

If EXTOUT is set to DMIGOP2, the reduced structural matrices and loading are written in OUTPUT2 format to a tape unit specified by the parameter EXTUNIT (default=30). The storage format is the same as the DMIGDB option and allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model. The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section.

If EXTOUT is set to DMIGPCH, the reduced structural matrices and loading are output on the punch file (.pch) in DMIG format.

The procedure for accessing the external superelement information depends on the option used to output the external superelement in Step 1. The methods are as follows:

1. If EXTOUT was MATRIXDB or DMIGDB in Step 1, use the following commands in the File Management Section:

```
ASSIGN SEXXX='step1.MASTER'
DBLOCATE DATABLK=(EXTDB)CONVERT(SEID=xxx),
LOGICAL=SEXXX
```

   where

   step1.MASTER is the database from Step 1.

and

xxx is the superelement identification number given to the partitioned Bulk Data Section for the external superelement.

2. If EXTOUT was DMIGOP2 in Step 1, then use the following commands in the File Management Section:

```
ASSIGN INPUTT2='step1_output2_file',UNIT=extunit
```

where

step1_output2_file is the OUTPUT2 file from Step 1.

and

extunit is the unit number specified by the parameter EXTUNIT (default = 30).

3. If EXTOUT was DMIGPCH in Step 1, then include the punch file from Step 1 in the partitioned Bulk Data Section. In addition, add the following Case Control commands in the subcase for the external superelement:

```
K2GG=KAAX
P2G=PAX
```

The SEBULK entry defining the superelement as an external superelement and the EXTRN entry in the partitioned Bulk Data Section should not be specified.

If data recovery is desired for the external component in SOLs 101, 103, and 107 through 112, there are three methods of transmitting the displacements of the reduced model to the external full model. The method is selected by the parameter EXTDROUT in the partitioned Bulk Data Section. The options are as follows:

1. EXTDROUT set to MATRIXDB. The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model.

2. EXTDROUT set to DMIGDB. The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.

3. EXTDROUT set to DMIGOP2. The same as EXTDROUT set to DMIGDB except that the displacements of the reduced model are written in OUTPUT2 format to a tape unit specified by parameter EXTDRUNT (default = 31). The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.

Data recovery for the external component is limited to SOLs 101 and 103 and 107 through 112. Data recovery is accomplished using a restart procedure from the data base created in Step 1 and setting parameter EXTDR to YES. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The input methods are as follows:

1. If EXTDROUT was MATRIXDB or DMIGDB in Step 2, then add the following commands in the File Management Section:

```
ASSIGN SEXX='step1.MASTER'
RESTART LOGICAL=SEXX
ASSIGN SEYYY='step2.MASTER'
DBLOCATE DATABLK=(EXTDB) WHERE(SEID=yyy),
LOGICAL=SEYYY
```

   where:

   step1.MASTER is the database from the Step 1.

   step2.MASTER is the database from the Step 2.

   and

   yyy is the superelement identification number given to the partitioned Bulk Data Section for the external superelement in Step 2.

2. If EXTDROUT was DMIGOP2 in Step 2, then add the following commands in the File Management Section:

```
ASSIGN INPUTT2='step2_output2_file',UNIT=extdrunt
```

   where

   step2_output2_file is the OUTPUT2 file from Step 2.

   and

   extdrunt is the unit number specified by the parameter EXTDRUNT (default = 31).

For SOL 101, the Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in Step 1 to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.

For SOL 103 and 107 through 112, the Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, FREQ/DLOAD or TSTEP/DLOAD subcases.

Step 1 Create External SE	Step 2 Perform Analysis	Step 3 Data Recovery for External SE
PARAM, EXTOUT, MATRIXDB	1. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB), CONVERT (SEID=xxx), LOGICAL=SEXX 2. PARAM,EXDROUT,MATRIXDB	1. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY 2. PARAM,EXTDR,YES
PARAM, EXTOUT, DMIGDB	1. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB), CONVERT (SEID=xxx), LOGICAL=SEXX 2. PARAM,EXDROUT,MATRIXDB or PARAM,EXDROUT,DMIGDB or PARAM,EXDROUT,DMIGOP2	1. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY 2. ASSIGN INPUTT2='step2_output_file', UNIT=Extdrunt 3. PARAM,EXTDR,YES

PARAM, EXTOUT, DMIGOP2	1. ASSIGN INPUTT2='step1_output2_file', Unit = extunit 2. PARAM,EXDROUT,MATRIXDB or PARAM,EXDROUT,DMIGDB or PARAM,EXDROUT,DMIGOP2	1. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY 2. ASSIGN INPUTT2='step2_output_file', UNIT=Extdrunt 3. PARAM,EXTDR,YES
	Include the.PCH file in partitioned Bulk Data Section and for external SE subcase 1. K2GG=KAAX P2G=PAX	

EXTRCV Default = 0

EXTRCV > 0 indicates that data recovery is to be performed on an external superelement. In this type of run, the database for the external superelement must be attached as the primary database (See "Database Concepts" in the *NX Nastran User's Guide*), and the database that contains the solution vectors, superelement map, and external/internal grid point equivalence table for its downstream superelement must be attached via the DBLOCATE statements. The value of EXTRCV must also be specified in the CONVERT clause of the DBLOCATE statement for the EMAP data block.

The following example shows the DBLOCATE statements for external superelement data recovery in SOL 101.

```
ASSIGN DOWNSE=...  $ DOWNSTREAM SUPERELEMENT DATABASE
DBLOC DB=EMAP CONVERT (EXTRCV=500) LOGICAL=DOWNSE
DBLOC DB=UL WHERE (SEID=0) LOGICAL=DOWNSE
DBLOC DB=UG WHERE (SEID=0) LOGICAL=DOWNSE
DBLOC DB=BGPDTS WHERE (PEID=0) LOGICAL=DOWNSE
SOL 101
CEND
SEDR=...  $ EXTERNAL SUPERELEMENT ID
.
```

```
.
.
BEGIN BULK
PARAM,EXTRCV,500
```

EXTUNIT Default = 30.

F – Parameters

F56 Default = NO

Directs the software to write an output file that contains warning messages, error messages, and convergence information only.

When F56 = YES, an .f56 file is created that contains all of the warning messages, error messages, and convergence information associated with a job.

When F56 = NO, an .f56 file is not created.

Optionally use system cell 552 to change the FORTRAN unit number from the default value of 56 to some other value.

FACTOR Default = 10000

See OLDSEQ.

FCTCi, FCTMi, FCTZi Default = 1.0

By default, coupling terms are included in the equations of motion for a rotor dynamic analysis. Use FCTCi, FCTMi, and FCTZi to selectively exclude coupling terms from the equations of motion.

FCTC1 = 0.0 excludes sin(ωt), cos(ωt), sin(2ωt), and cos(2ωt) terms from the Coriolis matrix components.
FCTC2 = 0.0 excludes sin(ωt) terms from the Coriolis matrix components.
FCTC3 = 0.0 excludes cos(ωt) terms from the Coriolis matrix components.
FCTC4 = 0.0 excludes sin(2ωt) terms from the Coriolis matrix components.
FCTC5 = 0.0 excludes cos(2ωt) terms from the Coriolis matrix components.

FCTM1 = 0.0 excludes sin(ωt), cos(ωt), sin(2ωt), and cos(2ωt) terms from the mass matrix components.

FCTM2 = 0.0 excludes sin(ωt) terms from the mass matrix components.
FCTM3 = 0.0 excludes cos(ωt) terms from the mass matrix components.
FCTM4 = 0.0 excludes sin(2ωt) terms from the mass matrix components.
FCTM5 = 0.0 excludes cos(2ωt) terms from the mass matrix components.

FCTZ1 = 0.0 excludes sin(ωt), cos(ωt), sin(2ωt), and cos(2ωt) terms from the centrifugal matrix components.
FCTZ2 = 0.0 excludes sin(ωt) terms from the centrifugal matrix components.
FCTZ3 = 0.0 excludes cos(ωt) terms from the centrifugal matrix components.
FCTZ4 = 0.0 excludes sin(2ωt) terms from the centrifugal matrix components.
FCTZ5 = 0.0 excludes cos(2ωt) terms from the centrifugal matrix components.

FIXEDB

Default = 0

FIXEDB is used to reduce the cost of superelement checkout.

FIXEDB = -2 (SOL 101 only) is used on the initial runs when the user suspects that the superelement may contain errors and that only operations necessary for fixed-boundary solutions need be performed. In particular, the generation of the $[G_{oa}]$ matrix is branched over in the SEKR operation and $[P_a]$ is not generated in the SELR operation. These operations typically result in 50% of the reduction cost and are not needed in the fixed-boundary data recovery operations described in the next paragraph. After this operation has been completed, the keyword SELANG will appear in the database dictionary, indicating that the $[P_a]$ stored there is incomplete, and should not be summed into the downstream superelement, because System Fatal Message 4252 will be issued.

FIXEDB ≤ -1 (SOLs 101 and 103 only) allows uncoupled solutions for any superelement with conventional output requests. This output may be obtained as soon as the superelement is successfully generated and reduced and does not require that the entire model be assembled. In superelement statics, the solution is the component due to the $\{u^o_o\}$ vector, i.e., a fixed-boundary solution. In superelement modes, the solution is the uncoupled eigenvectors of the component. If PARAM,FIXEDB,-1 is specified in the Bulk Data or in the residual structure subcase, the modes of the residual structure will not be computed. For a printout or plotting of the component mode eigenvectors it is recommended that PARAM,FIXEDB,-1 be

specified in the Bulk Data Section or above the subcase level in Case Control. If the modes of the residual structure are desired, then PARAM,FIXEDB,0 should be specified in the residual structure subcase. Exterior degrees-of-freedom listed on SECSETi and SESUP entries are free, and those on SEBSETi degrees-of-freedom are fixed. Data recovery for the residual structure should not be requested for this option.

FIXEDB = +1 (SOL 101 only) is used after the superelement has been proven valid. In the SEKR and SELR operations, it provides a branch over all operations already completed in the SEKR and SELR phases and completes the generation of the $[G_{oa}]$ matrix and the boundary stiffness and load matrices. It is also a method to recover the factor of the $[K_{oo}]$ matrix if the run aborted while computing $[G_{oa}]$.

FKSYMFAC Default = 1.0

FKSYMFAC controls the symmetrization of the follower force stiffness in SOL 106. By default, the follower force stiffness K_f is symmetrized as:

$$K_{fs} = \frac{1}{2}(K_f + K_f^T)$$

and the symmetric part K_{fs} is used for efficiency. If FKSYMFAC= 0. is specified, the original follower force stiffness K_f is used. If a value of 0. < FKSYMFAC < 1. is specified, the non-symmetric part of the follower force stiffness is calculated as:

$$K_{fn} = K_f - K_{fs}$$

and the ratio of unsymmetry:

$$r = \frac{\|K_{fn}\|}{\|K_f\|}$$

is compared with the user specified value of FKSYMFAC. The norm $\|\cdot\|$ is the absolute maximum number of the matrix.

If r < FKSYMFAC, the symmetric stiffness K_{fs} is used.

If r > FKSYMFAC, the original unsymmetrical stiffness K_f is used.

For most cases, the symmetrized follower force stiffness (default) will give sufficiently accurate answers. If the influence of the

non-symmetric part of the follower force stiffness is important, a value of FKSYMFAC=1.e-9 is recommended.

The parameter FKSYMFAC is applicable to SOL 106 only. All other solution sequences symmetrize the follower force stiffness. See parameter FOLLOWK for a list of solution sequences which calculate the follower force stiffness.

FLEXINCR Default = NO

In SOL 144, a value of YES will cause the TRIM subcases to be ignored. Instead, the TRIM Bulk Data will be used to obtain the set of Mach, Dynamic pressure and symmetry values for Unit Solutions (Flexible Increments). These data can be archived in the aeroelastic database for subsequent reuse. (Flexible Increments are always computed. This param merely avoids the TRIM subcase if these increments are all that is required.)

FLUIDMP Default = 0

In SOLs 111 and 200, under ANALYSIS=MFREQ with fluid-structure models (see Performing a Coupled Acoustic-Structural Analysis in the *NX Nastran Acoustics User's Guide*), fluid modal participation factors are computed and output for the first n modes of the fluid defined by user parameters LFREQFL, HFREQFL, and LMODESFL, where n is specified by PARAM,FLUIDMP,n. Structural, load, and panel factors are computed and output for the first n modes of the structure defined by user parameters LFREQ, HFREQ, and LMODES, where n is specified by PARAM,STRUCTMP,n. By default, panel factors are computed for all modes specified by PARAM,STRUCTMP. PARAM,PANELMP,-1 suppresses the calculation and printout of panel participation factors. If n = -1 for STRUCTMP or FLUIDMP, then factors will be output for all modes.

By default, factors are computed for all forcing frequencies at all fluid grid points in the residual structure. If this is not desired, then the OFREQ Case Control command may be used to specify the desired forcing frequencies, and the PARTN Case Control command may be used to specify the desired fluid grid points.

FLUIDSE Default = 0

PARAM,FLUIDSE,seidf specifies a special superelement reserved for fluid elements. Frequency dependent fluid elements must still be in the residual. The newer partitioned superelements are not supported.

FOLLOWK Default = YES

In SOLs 103, 105, 106, 107, 108, 109, 110, 111, 112, 115, and 116, FOLLOWK=YES (default) requests the inclusion of follower force stiffness in the differential stiffness. FOLLOWK=NO requests that the follower force stiffness not be included. For FOLLOWK=YES in SOLs 103, 107, 108, 109, 110, 111, 112, 115, and 116, a separate static subcase is required and the STATSUB command is also required in the eigenvalue subcase. In nonlinear analysis (SOL 106), the follower force is included if PARAM,LGDISP,1 is specified. FOLLOWK is ignored in SOL 106 if LGDISP is not specified.

FRQDEPO Default = NO

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

FRRU Default = YES

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

When SYSTEM(462) = 0, the software to turn off the automatic selection and request that the software use the original FRRD1 method, you can define the new parameter setting PARAM,FRRU,NO.

FRUMIN Default = 500

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

FZERO Default = 1.0E-3

Tolerance for zero frequency rigid body modes. Frequencies (in Hz) below this value are considered zero. See the AUTOSPRT and WMODAL parameters.

G – Parameters

G,GFL

Default = 0.0

G and GFL specify the uniform structural and fluid-damping coefficient in the formulation of dynamics problems. In coupled fluid-structure analysis, G is applied to the structural portion of the model and GFL to the fluid portion of the model. To obtain the value for the parameter G or GFL, multiply the critical damping ratio, C/C_o, by 2.0. PARAM,G and GFL are not recommended for use in hydroelastic or heat-transfer problems. If PARAM,G (or GFL) is used in transient analysis, PARAM,W3 (or W3FL) must be greater than zero or PARAM,G (or GFL) will be ignored. See “Formulation of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide*.

G*x*
(*x* is an integer)

Real, Default = (0.0,0.0)

G*x* is used by NX to retain node and element groups during the exporting and importing of bulk data files and .op2 results files. It is not used by NX Nastran, and it has no impact on the NX Nastran solution sequence.

For example, in the parameter setting

PARAM,G5,6.0,8.0

5 is the original NX group ID, 6 is a SET ID containing nodes, and 8 is a SET ID containing elements.

GDAMPF

Character, Default = YES

Determines if the global structural damping defined with PARAM,G is included in the stiffness when computing force and stress in frequency and transient response solutions for CELAS1, CELAS2, CELAS3, CELAS4, and CBUSH elements.

=NO PARAM,G is not included when computing force and stress.

=YES PARAM,G is included when computing force and stress.

GEOMU

Default = 40

See PARAM, POST, 0.

GPECT Default = -1

GPECT controls the printout of all elements connected to each grid point. GPECT=+1 requests the printout. In superelement analysis, the list is printed if PARAM,CHECKOUT,YES is specified or the SEMG or SEALL Case Control command selects the superelement. GPECT=-1 suppresses the printout.

GRDPNT Default = -1

GRDPNT>-1 will cause the grid point weight generator to be executed. The default value (GRDPNT=-1) suppresses the computation and output of this data. GRDPNT specifies the identification number of the grid point to be used as a reference point. If GRDPNT=0 or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid-related masses and masses on scalar points are ignored.

The NX Nastran principal mass axes have a specific meaning and should not be confused with the principal axes discussed in text books. The principal mass axes in NX Nastran have no coupling terms between the translational masses, but most importantly, they are with respect to the basic coordinate system. See Overview of the Grid Point Weight Generator in the *NX Nastran Basic Dynamic Analysis User's Guide* for more information.

The following weight and balance information is automatically printed following the execution of the grid point weight generator.

- Reference point.
- Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
- Transformation matrix [S] from the basic coordinate system to principal mass axes.
- Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.).
- Inertia matrix I(S) about the center of gravity relative to the principal mass axes. Note: Change the signs of the off-diagonal terms to produce the “inertia tensor.”
- Principal inertias I(Q) about the center of gravity.

- Transformation matrix [Q] between S-axes and Q-axes. The columns of [Q] are the unit direction vectors for the corresponding principal inertias.

In superelement static or geometric nonlinear analysis, GRDPNT> -1 also specifies the grid point to be used in computing resultants, in the basic coordinate system, of external loads and single point constraint forces applied to each superelement. If GRDPNT is not a grid point (including the default value of -1), then the resultants are computed about the origin of the basic coordinate system. In superelement analysis, weights and resultants are computed for each superelement without the effects of its upstream superelements.

For the CQUDX4, CQUADX8, CTRAX3, and CTRAX6 elements, the masses and inertias are reported for the entire model of revolution but the center of gravity is reported for the axisymmetric cross section.

GUSTAERO Default = 1

If gust loads are to be computed, for example on restart, set GUSTAERO to -1. The default is recommended if no gust loads are to be computed after the flutter analysis.

H – Parameters

HEATSTAT Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

```
TEMP(LOAD) = Heat Transfer Subcase ID
```

If this default is not acceptable, then in the heat transfer subcase add the Case Control word TSTRU=SID and in the structures subcase here

```
TEMP(LOAD) = SID
```

See the Case Control command, TSTRU. PARAM,NESET is no longer used.

HFREQ, HFREQFL

Default = 1.+30

The parameters LFREQ, HFREQ, LFREQFL, and HFREQFL specify the frequency range in cycles per unit time of the modes to be used in the modal formulations. (LFREQ and LFREQFL are the lower limits and HFREQ and HFREQFL are the upper limits.) In coupled fluid-structure analysis, HFREQ and LFREQ are applied to the structural portion of the model and HFREQFL and LFREQFL are applied to the fluid portion of the model. Note that the default for HFREQ and HFREQFL will usually include all vectors computed. Related parameters are LMODES and LMODESFL.

I – Parameters

IFP

Default = value of PARAM,DBALL

See DBALL

IFTM

Default = 0

IFTM specifies the method for Inverse Fourier Transformation in SOLs 111 and 146. See the *NX Nastran Aeroelastic Analysis User's Guide* for further discussion. Permissible values are 0, 1, and 2. The default value is recommended.

0: constant (default)

1: piecewise linear

2: cubic spline

INEMETH

Default = 1

Controls the coupling of mass and mass moments of inertia terms in the rotor dynamics solutions (107, 108 and 109) when using the coupled method between the rotating and fixed part (selected by PARAM, ROTCOUP). In the 6x6 mass matrix, the upper left 3x3 are the mass terms (M_{ij}), and the lower right 3x3 are the mass moments of inertia terms (I_{ij}).

1: Both the mass terms and the mass moments of inertia terms are coupled.

0: Only the mass terms are coupled.

INP4FMT Default = 32

Selects the type of OP4 file you are inputting when using the ILP-64 executable.

If INP4FMT=32 (default), a 32-bit OP4 file is the input, and the software automatically converts the 32-bit data to 64-bit.

If INP4FMT=64, a 64-bit OP4 file is the input. No conversion is required, and the 64-bit data is read directly.

INREL Default = 0

INREL controls the calculation of inertia relief or enforced acceleration in SOLs 101, 105, and 200. INREL = -1 requests that inertia relief or enforced acceleration be performed. INREL=-2 (SOL 101 only) requests automatic inertia relief, in which case no SUPORT entries are necessary.

SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. The total number of degrees-of-freedom specified on SUPORT and SUPORT1 entries must be less than or equal to six.

In SOL 105, SUPORT1, not SUPORT, Bulk Data entries must be used to define the supported degrees-of-freedom and the SUPORT1 Case Control command may only be specified in a separate static subcase.

Loads due to unit rigid body accelerations at the point referenced by PARAM,GRDPNT are computed and then appended to the external loads. If PARAM,GRDPNT is specified in superelement analysis, then the point must be interior to the residual structure and exterior to all superelements.

Enforced accelerations, if desired, are input on the DMIG,UACCEL Bulk Data entry (not supported for the INREL=-2 option).

INREL=-1 and INREL=-2 are both supported with models which include linear contact when the sparse solver is used.

INREL=-1 and INREL=-2 are both supported in models which include glue conditions when the sparse solver is used.

Bolt preload (BOLTLD case control command) is not supported with inertia relief.

If INREL=0, a static analysis is performed.

See also:

"Data Recovery Operations in SubDMAP SEDISP" in the *NX Nastran User's Guide*

"Buckling Analysis in SubDMAP MODERS" in the *NX Nastran User's Guide*

"Automatic Inertia Relief" in the *NX Nastran User's Guide*

INRLM Default = 0

INRLM=-1 requests that component (superelement) inertia relief shapes be appended to the generalized dynamic modes as described in the NX Nastran User's Guide, Solution Sequences chapter, Superelement Analysis section. If this feature is used, the component must have at least six independent rigid body mass degrees-of-freedom; otherwise, duplicate modes will be generated, causing a singular matrix.

IRES Default = -1

IRES=1 requests that the residual load vectors RULV and RUOV be output in all solution sequences. In superelement analysis, the parameters PRPA and PRPJ may also be used to request output of the partial load vectors $\{P_a\}$ and $\{P_j\}$, respectively. In geometric nonlinear analysis, PARAM,IRES,1 will cause the printing of the residual vector as follows:

$$\{\Delta P_f\} = [K_{ff}]\left\{u_f^{n+1} - u_f\right\} + \{F_f\} - \{P_f\}$$

ITAPE Default = -1

ITAPE specifies the output status of the DSCMR matrix in SOLs 101, 103, and 105; and the DSCMCOL table and the DSCM2 matrix in SOL 200. (See the OUTPUT2 and OUTPUT4 module descriptions in the *NX Nastran DMAP Programmer's Guide*.)

IUNIT Default = 11

IUNIT specifies the FORTRAN unit number on which the DSCMR matrix in Design Sensitivity SOLs 101, 103, and 105 and the DSCMCOL table and the DSCM2 matrix in SOL 200 will be written. (See the OUTPUT2 and OUTPUT4 module descriptions in the *NX Nastran DMAP Programmer's Guide*.)

K – Parameters

K6ROT

Default = 100.0

K6ROT specifies the stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. This is an alternate method to suppress the grid point singularities, and is intended primarily for geometric nonlinear analysis. A value between 1.0 and 100.0 is recommended to suppress singularities. A large value may be required in nonlinear and eigenvalue analyses. This parameter is ignored for CQUADR, CTRIAR, CQUAD8, and CTRIA6 elements. K6ROT is forced to 0 when only membrane elements exist.

KDAMP, KDAMPFL

Default = 1

If KDAMP (or KDAMPFL) = 1, values entered on TABDMP1 entry are used to compute modal viscous damping.

If KDAMP (or KDAMPFL) = -1, values entered on TABDMP1 entry are used to compute modal structural damping.

In coupled fluid-structure analysis, KDAMP is applied to the structural portion of the model and KDAMPFL to the fluid portion of the model.

For more information, see “Formulation of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide*.

KDIAG

Default = -1.0 (SOLs 106 and 153 only)

In nonlinear static analysis, KDIAG may be used to eliminate spurious mechanisms and singularities in the nonlinear stiffness matrix. The absolute value of KDIAG will be added to some or all of the diagonal terms in the nonlinear stiffness matrix as follows:

If KDIAG < 0.0, then add the absolute value of KDIAG to the null diagonal terms only. (Default)

If KDIAG = 0.0, then no action is taken.

If KDIAG > 0.0, then add the value of KDIAG to all diagonal terms.

KGGCPCH Default=0

Requests the final contact stiffness matrix in DMIG format from a SOL 101 linear contact solution. The solver writes the contact stiffness matrix from the final contact iteration (not necessarily converged) into a PUNCH file in the format required for DMIG (Direct Matrix Input at Grids). This option is only available with the default sparse solver. The iterative solvers do not support this option.

If KGGCPCH=0, contact stiffness matrix is not output.

If KGGCPCH=1, contact stiffness matrix is output.

KGGLPCH Default=0

Requests the glue stiffness matrix in DMIG format in any solution except 144-146, 401, 601 and 701. The solver writes the glue stiffness matrix into a PUNCH file in the format required for DMIG (Direct Matrix Input at Grids). This option is only available with the default sparse solver. The iterative solvers do not support this option.

If KGGLPCH=0, glue stiffness matrix is not output.

If KGGLPCH=1, glue stiffness matrix is output.

L – Parameters

LAMLIM Default = 0.001

LAMLIM is used by the NX Response Dynamics Capability to identify rigid body modes for computing the viscous damping ratio matrix which is written to the OP2 file when RSOPT =1. The frequencies below the LAMLIM value will be considered a rigid body mode. The column and rows of the viscous damping ratio matrix associated with the rigid body will be set to zero.

LANGLE Default = 1

LANGLE specifies the method for processing large rotations in nonlinear analysis. By default, large rotations are computed with the gimbal angle method in nonlinear analyses SOLs 106, 129, 153, and 159 with geometric nonlinearity (PARAM,LGDISP,1). If PARAM,LANGLE,2 is specified, then they are computed with the Rotation Vector method. The value of LANGLE cannot be changed in a subsequent restart.

LFREQ, LFREQFL

Default = 0.0

See HFREQ, HFREQFL.

LGDISP

Default = -1

If LGDISP = + 1, all the nonlinear element types that have a large displacement capability in SOLs 106, 129, 153, 159, and 401 (see “Element Summary – Small Strain Elements” in the *NX Nastran Element Library* under “Geometric Nonlinear”) will be assumed to have large displacement effects (updated element coordinates and follower forces). If LGDISP = -1, then no large displacement effects will be considered.

SOLs 401, 601, and 701 support LGDISP = +1 or -1 only.

Deformation dependent loading in SOLs 601 and 701 is controlled by the LOADOPT parameter in the NXSTRAT Bulk Data entry.

If LGDISP = 2, then follower force effects will be ignored but large displacement effects will be considered.

If LGDISP ≥ 0, then the differential stiffness is computed for the linear elements and added to the differential stiffness of the nonlinear elements.

LGSTRN

LGSTRN is supported by SOLs 601 and 701 only.

If LGSTRN = 0, small strains are assumed. (Default)

If LGSTRN = 1, large strains, displacements and rotations are assumed (i.e. LGDISP is automatically set to 1). Large strain formulation is only applicable to 2D axisymmetric, plane strain, 3D solid and single layer shell elements (except 8-node shell).

LMDYN

Integer input; LMDYN default = -1; LMSTAT default = 6

With Lagrange RBE3 elements, these parameters scale the artificial stiffness term c_k according to the following equations:

Dynamic solutions: $c_K = \frac{1.0}{10^{LMDYN}}$

Static solutions: $c_K = \frac{1.0}{10^{LMSTAT}}$

If either LMSTAT or LMDYN is defined as -1, c_K is set to 0.0, and the equations for c_K are not applied. See “Rigid element processing options” in the *Element Library Reference*.

LMFACT

Default = 1.0E+6

Scale factor used to adjust the stiffness matrix of Lagrange rigid elements (see the case control command “RIGID” for more information). If their stiffness is too small, the results will be inaccurate. If their stiffness is too large, numerical problems can occur (fatal errors during the solution may occur). Best results are obtained when their stiffness is similar to the rest of the model.

LMODES, LMODESFL

Default = 0

LMODES and LMODESFL are the number of lowest modes to use in a modal formulation. In coupled fluid-structure analysis, LMODES specifies the lowest modes of the structural portion of the model and LMODESFL the modes of the fluid portion of the model. If LMODES (or LMODESFL) = 0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

In SOL 103, LMODES may be used to reduce the number of eigenvectors to be processed in data recovery which may significantly reduce the CPU and storage costs.

LMSTAT

See the parameter LMDYN.

LOADU

Default = -1

See PARAM, POST, -1.

LOOPID

Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153, the restart proceeds from the last loop ID of the subcase defined by PARAM,SUBID.

LSTRN

Replaced by the STRAIN Case Control command.

M – Parameters

MACH Default = 0.0

Mach number. If more than one Mach number was used to compute aerodynamic matrices, the one closest to MACH will be used in dynamic aeroelastic response analysis. The default causes the matrices computed at the lowest MACH number to be used.

MATNL Default = –1

Specify PARAM,MATNL,1 to turn on all material nonlinear capabilities in SOL 401. Material nonlinear capabilities include plasticity and creep. Specify PARAM,MATNL,–1 (default) to turn off all material nonlinear capabilities in SOL 401.

MAXLP Default = 5

MAXLP specifies the maximum number of iterations for element relaxation and material point subincrement processes in SOLs 129 and 159. MAXLP is 10 in SOLs 106 and 153 and cannot be changed by the user.

MAXRATIO Default = 1.E7

Default = 1E10 if glue, linear contact or Lagrange rigid elements are present in the model.

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (nonpositive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAILOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAILOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

The value -1 of BAILOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101 through 200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence. (See “Superelement Analysis” in the *NX Nastran User’s Guide*.)

BAILOUT and MAXRATIO may be specified in the Case Control Section in order to provide different values between superelements but not between different boundary conditions.

MDOF

See the parameter MGRID.

MECHFIL

Default: 1.0E-6

Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constrained modes being retained. This parameter is associated with the MECHFIX parameter.

MECHFIX

Default = AUTO

Control for fixing the massless mechanism problem. This capability is provided automatically by default. It is executed only when the solution does not provide answers because of symptoms consistent with the presence of massless mechanisms. If MECHFIX is set to YES, the constrained modes are removed before attempting a solution. When set to NO, this capability is blocked, and the eigensolution portion uses the rule of three failed shifts and a fatal exit. See the parameter MMFIL.

MECHPRT

Default = NO

For SOL 103 only. If massless mechanisms are found, the constrained modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINED MODE, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model with the goal to remove all massless mechanisms before starting production analysis. The number of each “mode” matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes, the output file will be large. There is no method to plot these shapes.

MESH Default = NO

If MESH=YES is specified, then a summary of the shading status and the subelement mesh for each CHBDYi element referencing a VIEW Bulk Data entry is printed.

METHCMRS Default = 0

In dynamic analysis (SOLs 103, 107, 108, 109, 110, 111, 112, 145, 146, and 200), METHCMRS specifies the set identification number of an EIGR or EIGRL entry to be used in the computation of the normal modes on the o-set (fixed boundary modes) or v-set (mixed boundary modes) of the residual structure depending on the b-set and/or c-set entries.

These modes are calculated for a component mode reduction of the residual structure when q-set exists. If no q-set exists in the residual structure, a component modal reduction is not performed.

The case control entry RSMETHOD has the same effect as the METHCMRS parameter. If both of them exist, the RSMETHOD takes precedence. If neither exists, a warning message is issued since only Guyan reduction will be executed and the results are poor. In the latter case, the residual structure fixed boundary normal modes or mixed boundary normal modes will be computed based on the METHOD Case Control command and the appropriate b-set and c-set entries.

MGRID The parameters MGRID and MDOF only apply to the solution monitor when a direct frequency response (SOL 108) or a direct transient response (SOL 109) is used. MGRID selects the grid (node) ID, and MDOF selects the DOF (123456) for plotting the solution progress within the user interface. By default, the first dof in the L-set is used (See Displacement Vector Sets in the *NX Nastran Basic Dynamic Analysis User's Guide* for a definition of the L-set).

MMFIL Default = 1.0E-10

Filter value used to distinguish between massless mechanism modes and rigid body modes. The default value has been found to be effective; a smaller value may discard rigid body modes.

MODACC Default = -1

MODACC = 0 selects the mode acceleration method for data recovery in dynamic analysis. See “Formulation of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide* for further discussion.

MODACC = 1 is the same as MODACC = 0 except if SUPORTi entries are present then the displacements are corrected for rigid body motion and the local flexibility at the SUPORTi degrees-of-freedom.

MODACC ≥ 0 is not recommended for use in hydroelastic problems.

MODEL Default = 0

This parameter also allows several models to be stored in the same graphics database created by PARAM,POST,0.

MODTRK Default = 2

This parameter is used in a rotor dynamics solution to select the mode tracking method.

If MODTRK = 1, the pre-NX Nastran 7 method is used. Outer loop over rotor speed, inner loop over degrees of freedom. This method does not work well for the direct method (SOL 107) because there, new solutions can come in and old solutions can leave the solution space.

If MODTRK = 2, the method introduced in NX Nastran 7 is used. Outer loop over degrees of freedom, inner loop over rotor speed. Process is repeated until all solutions have been tracked.

If MODTRK = 3, the method introduced in NX Nastran 8.5 is used. Eigenvectors and eigenvalues are used to track the modes. This method is applicable to models having any combination of unsymmetric stiffness, unsymmetric viscous damping, and structural damping.

If MODTRK = 4, the method introduced in NX Nastran 11 is used. The initial reference rotor modes are used as the base vectors for complex eigenvalue calculations at subsequent reference rotor speeds. Mode tracking begins at the first nonzero reference rotor speed. Modes for the reference rotor speed of zero are excluded from the Campbell diagram results. Strain and kinetic energy filtering of modes can be used with this mode tracking option. For information on how to filter modes, see the ROTPARM bulk entry.

The following parameters apply if MODTRK = 2: MTREPSI, MTREPSR, MTRFCTD, MTRFCTV, MTRFMAX, MTRRMAX, and MTRSKIP.

The following parameters apply if MODTRK = 3: MTRFMAX and MTRRMAX.

If the tolerances MTREPSR and MTREPSI are chosen too small, the solution may be lost for some speed values. If the tolerance is too large, there may be lines crossing from one solution to another. Problems may occur for turbines with many elastic blades with equal frequencies. Then clusters of lines and crossings may occur.

MPCX

Default = 0

See OLDSEQ.

MPCZERO

Default = 1.0E–11

Filter value to remove small “noise” values from the multipoint constraint matrix. To turn this filtering off, set to a value less than or equal to 0.0.

MTREPSI

Default = 0.01

Applies when MODTRK = 2. Relative tolerance for the imaginary part (eigenfrequency). When a root has been found in the extrapolation, a check is made if the correct solution has been found. If the present value is outside of this tolerance, the solution is skipped.

MTREPSR

Default = 0.05

Applies when MODTRK = 2. Relative tolerance for the real part (eigenfrequency). When a root has been found in the extrapolation, a check is made if the correct solution has been found. If the present value is outside of this tolerance, the solution is skipped.

MTRFCTD

Default = 0.5

Applies when MODTRK = 2. Threshold value for damping. This is in order to disregard the real part for solutions with low damping.

MTRFCTV

Default = 0.0

Applies when MODTRK = 2. Reference value for converting aerodynamic speed to rotor speed for wind turbines. Used only for wind turbines in SOL 145.

MTRFMAX Default = 0.0

Applies when MODTRK = 2 or 3. Maximum frequency to consider. If zero, all frequencies are used. With this parameter high frequency solutions can be filtered out.

MTRRMAX Default = 0.0

Applies when MODTRK = 2 or 3. Maximum absolute value of real part of solution to consider. If zero, all real part values are used. With this parameter solutions with high real parts can be filtered out. It can be useful to disregard numerical solutions which are not physically relevant.

MTRSKIP Default = 5

Applies when MODTRK = 2. If roots are not found for a specific speed, this speed is skipped and the next speed is analyzed. If there are more than MTRSKIP values missing, the curve is not considered. The solution will be marked as unused and will be considered in the next loop.

N – Parameters

NASPRT Default = 0

NASPRT specifies how often data recovery is performed and output in SOL 200. By default, SOL 200, in addition to performing an analysis and optimization, will perform full data recovery operations at the initial design cycle (cycle 0) and upon completion of the last design cycle.

If NASPRT < 0, then no data recovery operations are performed.

If NASPRT > 0, then data recovery operations are performed as follows, based on the numbering of the cycles in the “Summary of Design Cycle History” table at the end of the f06 file (*):

– At the initial design cycle (design cycle 0).
– At every design cycle that is a multiple of NASPRT, but counting cycle 0 as well (see the examples below).
– At the last design cycle.

For example, if PARAM,NASPRT,2 and the job has run for 5 cycles, then data recovery is done at the design cycles numbered 0, 1, 3, and 5. Design cycle 1 data recovery is done since cycle 1 is the second cycle starting with cycle 0.

Similarly, if PARAM,NASPRT,3, and the job has run for 9 cycles, then data recovery is done at cycles 0, 2, 5, 8, and 9.

(*) Due to legacy reasons, cycle numbers are printed out earlier within the f06 file during the flow of the program and are ahead of the numbers in the “Summary of Design Cycle History” table by a count of one before analysis takes place. This behavior will be modified in future versions to match the design cycle numbers in the “Summary of Design Cycle History” table.

NDAMP Default = 0.01

In SOLs 129 and 159, defines numerical damping in order to achieve numerical stability. A value of zero requests no numerical damping. The recommended range of values is from 0.0 to 0.1.

NEWSEQ Renamed OLDSEQ.

NINTPTS Default = 10

NINTPTS requests interpolation over the NINTPTS elements closest to each grid point. NINTPTS=0 requests interpolation over all elements, which can be computationally intensive. Related parameters include BIGER, CURV, NUMOUT, OG, OUTOPT, S1G, S1M, S1AG and S1AM.

NLAYERS Default=5 (minimum=1, maximum=12) (SOLs 106, 129, 153, and 159)

NLAYERS is used in material nonlinear analysis only and is the number of layers used to integrate through the thickness of CQUAD4 and CTRIA3 elements with isotropic material properties. Set NLAYERS=1 for efficiency if no bending is selected (MID2=0 or -1 on all PSHELL data entries). Do not specify NLAYERS=1 if MID2 is greater than zero. A larger value of NLAYERS will give greater accuracy at the cost of computing time and storage requirements.

NLHTLS Default=0

See Remarks under Case Control command, TSTRU..

NLMAX Default = 60

The number of suspected massless mechanisms is determined from the number of high ratio messages. If this number exceeds NLMAX, the number of trial massless mechanisms is reduced to NLMAX. This is a means to avoid an expensive run when there may be thousands of massless mechanisms due to modeling errors. The value of this parameter may be increased on initial runs where it is possible that there are many high ratio DOFs in order to see them all at once.

NLMIN Default = 10

If there are only a few high ratio DOFs there may be more mechanism modes present. More trial mechanism mode vectors are used, and those that do not indicate true problems are discarded. A smaller value could be considered on a stable mode undergoing small modeling changes.

NLTOL Default = 2 (SOL 106 only)

Default = 0 (SOL 153, nonlinear heat transfer)

NLTOL establishes defaults for the CONV, EPSU, EPSP and EPSW fields of NLPARM Bulk Data entry for nonlinear static and heat transfer analysis according to the following table.

NLTOL	Level of Accuracy
0	Very high
1	High
2	Engineering Design
3	Preliminary Design

See the NLPARM entry for further details and corresponding default NLPARM field values.

NMLOOP Default = 0

In SOLs 106 and 153, nonlinear static analysis, normal modes will be computed with the updated nonlinear stiffness if PARAM,NMLOOP is not equal to zero. The nonlinear normal modes will be computed at the last iteration loop of each subcase in which a METHOD command appears.

NOCOMPS Default = +1

NOCOMPS controls if stress and strain is computed for the composite elements, the equivalent homogeneous elements, or both. Composite elements reference a PCOMP or PCOMPG entry. Equivalent homogeneous elements, which are typically created by the software, reference a PSHELL entry and are designated with a MID1 or MID2 ID greater than or equal to 10^8. The STRESS and/or STRAIN case control commands are also required for any of these options.

NOCOMPS	Composite Stress/Strain	Equivalent Element Stress/Strain
1	YES	NO
0	YES	YES
-1	NO	YES

Stresses on the equivalent homogeneous elements are based upon a smeared representation of the laminate's properties and in general will be significantly different than the more accurate lamina stresses available on the PCOMP-based elements. Any PSHELL entries which have a MID1 or MID2 ID greater than or equal to 10^8 are considered by the software to be for equivalent homogeneous elements and require NOCOMPS 0 or -1 for any stress or strain recovery.

NOELOF Default = -1

If NOELOF > 0, then the grid point forces (GPFORCE Case Control command) are computed along the edges of the two-dimensional elements. The default value will suppress this output.

NOELOP Default = -1

If NOELOP > 0, then the sum of the grid point forces (GPFORCE Case Control command) are computed parallel to the edges of adjacent elements. The effect of CBAR element offsets is not considered in the calculation of the forces. The default value will suppress this output.

NOFISR Default = 0

NOFISR controls the printout of the composite failure indices and strength ratios. If NOFISR > 0, the failure indices and strength ratios will not be printed.

NOGPF Default = 1

NOGPF controls the printout of grid point forces (GPFORCE Case Control command). NOGPF > 0 specifies that the grid point forces are to be printed. NOGPF < 0 suppresses the printing of the grid point forces.

NOMSGSTR Default = 0

If NOMSGSTR = -1, the MSGSTRES module will be skipped even though Bulk Data entries provided for it.

NONCUP Default = -1

In SOLs 111 and 112, NONCUP selects either a coupled or uncoupled solution algorithm in modal frequency and modal transient response analyses. See “Frequency Response and Random Analysis in SubDMAP FREQRS” and “Transient Response in SubDMAPs SEDTRAN and SEMTRAN” in the *NX Nastran User’s Guide*.

NONCUP = -1 requests the coupled algorithm; however, if the dynamic matrices KHH, MHH, and BHH are diagonal, the uncoupled algorithm is used.

NONCUP= -2 requests the uncoupled algorithm regardless of the existence of off-diagonal terms in the dynamic matrices.

User Information Message 5222 indicates which algorithm is used in the analysis.

When using the ADAMSMNF or MBDEXPORT case control commands to request an Adams MNF file, a standard or state-space MATLAB file, or a standard or state-space OP4 file, the full equivalent modal viscous damping matrix is written to the interface file by default. To write only the diagonal values of the equivalent modal viscous damping matrix to the interface file, specify NONCUP=–2 on the ADAMSMNF or MBDEXPORT case control command, or specify PARAM,NONCUP,–2 in the bulk data section of the input file.

If both the NONCUP describer and the NONCUP parameter are specified, the NONCUP describer specification takes precedence.

NQSET Default = 0

If NQSET > 0, and the PARAM entry is in Case Control, all part superelements that do not contain QSET entries, or are not referenced by SENQSET entries in the main Bulk Data Section, have NQSET generalized coordinates assigned to them. These QSET variables are functionally equivalent to those generated by SENQSET entries.

NUMOUT, NUMOUT1, NUMOUT2 See S1, S1G, S1M.

O – Parameters

OCMP Default = YES

See POST.

ODS Default = NO

Requests an efficient RDMODES restart method which reduces the amount of eigenvector restart data.

= NO The efficient RDMODES restart method is not requested.

= YES The efficient RDMODES restart method is used when you run SOL 103 or 111 with RDMODES. As a result, the size of the restart data is smaller and is organized for an RDMODES restart solution to begin efficiently.

OEE Default = YES

See POST.

OEF Default = YES

See POST.

OEFX Default = YES

See POST.

OEPT Default = YES

See POST.

OES Default = YES

See POST.

OESE Default = YES

See POST.

OESX Default = YES

See POST.

OG Default = 0

See CURV.

OGEOM Default = YES

See POST.

OGPF Default = YES

See POST.

OGPS Default = YES

See POST.

OIBULK Default = NO

See PARAM,POST,<0.

OLDSEQ Default = -1 for non-iterative distributed parallel solutions

Default = 5 for iterative solutions using parallel methods (NASTRAN ITER = 1 and system (231) > 0)

Default = 6 if Super = 2

OLDSEQ selects from the following options for resequencing:

–1: No resequencing is performed.

1: Use the active/passive option.

2: Use the band option.

3: For the active/passive and the band option select the option giving the lowest RMS value of the active columns for each group of grid points.

4: Use the wavefront (Levy) option.

5: Use the Gibbs-King option even if the CPU estimate is higher than for nonsequencing.

6: Use the automatic nested dissection option even if the CPU estimate is higher than for no resequencing. See the following SUPER=2 description.

8: Semiautomatic selection. The program will compute estimates for two options that are suitable for the decomposition method selected by the PARALLEL and SPARSE keywords on the NASTRAN statement and select the option with the lowest estimate. The following table shows the suitable options for each decomposition method.

Decomposition Method	Suitable Options
regular	1 and 4
parallel	2 and 5
sparse	6 and 7

9: The Extreme partitioning is selected to generate domains for DMP solutions.

10: METIS with supernodal or grid compression is selected to generate domains for DMP solutions (see restrictions below).

11: MLV with supernodal or grid compression is selected to generate domains for DMP solutions.

110: Metis with supernodal compression is selected to generate domains for DMP solutions.

111: MLV with supernodal compression is selected to generate domains for DMP solutions.

210: Metis with grid compression is selected to generate domains for DMP solutions.

211: MLV with grid compression is selected to generate domains for DMP solutions.

In the GPARTN and GPARTNS modules, if at least two processors are available and both supernodal and grid compression are requested (OLDSEQ=10 or 11), then GPARTN/GPARTNS selects the compression resulting in the smallest global boundary size. If only one processor is available, the default is supernodal compression.

The default OLDSEQ value is 11. Only the MLV is supported for multilevel RDMODES. System(294)=1 requests that diagnostic information be printed to the f06 file.

Note that the wavefront option does not support superelement resequencing or starting nodes. Also note that the automatic nested dissection option uses starting nodes only to establish the root of the initial connectivity tree.

If the value of OLDSEQ is changed in superelement analysis, a SEALL=ALL restart is required.

PARAM,FACTOR is used to generate the sequenced identification number (SEQID) on the SEQGP entry as follows:

```
SEQlD=FACTOR*GRP+SEQ
```

where

SEQ = generated sequence number

and

GRP = group sequence number

If GRP=0, use GRP(MAX)+1 where GRP(MAX) is the largest group sequence number in the database.

PARAM,MPCX controls whether the grid point connectivity created by the MPC and MPCADD entries and/or the rigid element entries (e.g., RBAR) is considered during resequencing:

-1: Do not consider the connectivity of the MPC, MPCADD, or rigid element entries.

0: Consider the connectivity of the rigid element entries only. (Default).

>0: Consider the connectivity of the rigid element entries and the MPC and MPCADD entries with the set identification number set to the value of this parameter.

PARAM,SEQOUT controls the output options as follows:

0: Do not generate any printed or punched output for the new sequence. (Default)

1: Print a table of the internal/external sequence in internal order.

2: Write the SEQGP entries to the PUNCH file.

3: Perform SEQOUT=1 and 2.

PARAM,START specifies the number of the grid points at the beginning of the input sequence. The input sequence will be the sorted order of the grid point numbers including the effect of any SEQGP entries input by the user. A single SEQGP entry can be input to select the starting point for the new sequence. Otherwise, the first point of lowest connectivity will be used as the starting point.

If PARAM,SUPER<0, all grid points from the connection table that are not part of the group currently being processed are deleted. This option provides for sequencing only the interior points of a superelement. If any superelements are present, the residual structure is not resequenced. If all the grid points are in the residual structure, they are resequenced.

If PARAM,SUPER=0 or 1, all grid points in the connection table are considered. This option provides for the recognition of passive columns.

If PARAM,SUPER=2, then all points that are connected to multipoint constraints (via MPC entries) or rigid elements (e.g., the RBAR entry) are placed in a special group at the end of the sequence. This option also forces OLDSEQ=6 and may not be selected with other values of OLDSEQ. This option is intended primarily for models that have many active columns due to MPCs or rigid elements; e.g., a model with disjoint structures connected only by MPCs or rigid elements.

OMAXR Default = 2*BUFFSIZE

OMAXR specifies the maximum record length of data written by the OUTPUT2 module. OUTPUT2 data is requested when PARAM,POST,<0 is defined. BUFFSIZE is a machine-dependent value defined in the *NX Nastran Installation and Operations Guide*. For further information, see the MAXR parameter description under the OUTPUT2 module description in *NX Nastran DMAP Programmer's Guide*.

OMID Default = NO

To print or punch the results in the material coordinate system, set the parameter OMID to yes. Applicable to forces, strains, and stresses for CTRIA3, CQUAD4, CTRIA6, CQUAD8, CQUADR and CTRIAR elements. Other elements and outputs are not supported. This capability is not supported by pre-processors (xdb and OP2 output do not change) and grid point stress output which assume output is in the element coordinate system.

See "Using the OMID Parameter to Output Shell Element Results" in the *NX Nastran Element Library*

OMPT Default = YES

See POST.

OP2FMT Default = 32

Determines if OUTPUT2 results files are written as 32-bit or 64-bit when using the ILP-64 executable.

=32 (default), the software writes the OP2 file as 32-bit.

=64, the software writes the OP2 file as 64-bit.

=0, the software automatically checks if a PARAM, POST,*n* entry exists. If *n* is less than 0, the OP2 file is written as 32-bit. If not, the OP2 file is written as 64-bit.

Note: When using the ILP-64 executable and SYSTEM(525)=1 is defined, the OP2FMT parameter is ignored, and the software writes the OP2 file as 64-bit.

OP4FMT Default = 32

Determines if OUTPUT4 results files are written as 32-bit or 64-bit when using the ILP-64 executable.

If OP4FMT=32 (default), the OP4 file is always written as 32-bit.

If OP4FMT=64, the OP4 file is always written as 64-bit.

OPG Default = YES

See POST.

OPGEOM Default = -1

OPGEOM > -1 prints the set definition for all degrees of freedom, including the aerodynamic degrees of freedom. OPGEOM is similar to the USETPRT parameter.

OPGTKG Default = -1

OPGTKG > -1 prints the matrix for the interpolation between the structural and aerodynamic degrees-of-freedom.

OPPHIB Default = -1

In the flutter (SOLs 145 and 200) and aeroelastic (SOLs 146 and 200) solution sequences, OPPHIB > -1 and a DISPLACEMENT request in the Case Control Section will output the real vibration modes with the structural displacement components transformed to the basic coordinate system.

OPPHIPA Default = -1

In the flutter (SOLs 145 and 200) and the dynamic aeroelastic (SOL 146) solution sequences, OPPHIPA > -1 and a DISPLACEMENT command in the Case Control Section will output the real vibration modes at all degrees-of-freedom, including the aerodynamic degrees-of-freedom.

OPTEXIT Default = 0

In SOL 200, especially during the checkout of the analysis model and the design optimization input data (design model), it may be desirable to exit the solution sequence at certain points before proceeding with full optimization. OPTEXIT may be set to values of 0 through 8 and -4. The DSAPRT Case Control command overrides the specification of PARAM,OPTEXIT,4, -4, or 7. The description of OPTEXIT values follow.

OPTEXIT Value	Description
0	Do not exit. Proceed with optimization.
1	Exit after the initialization of the analysis and design model but before finite element analysis begins.
2	Exit after finite element analysis and initial design response and shape basis vector processing.
3	Exit after design constraint evaluation and screening.
4	Exit after design sensitivity analysis and print the matrix of design sensitivity coefficients (DSCM2). This is equivalent to the DSAPRT (UNFORM,END=SENS) Case Control command.
-4	Exit after design sensitivity analysis and write the data blocks related to sensitivity coefficients (DSCM2 and DSCMCOL) to an external file using the OUTPUT2 and OUTPUT4 modules. This is equivalent to the DSAPRT (NOPRINT,EXPORT END=SENS) Case Control command. See related parameters ITAPE, IUNIT, and OMAXR.
5	Exit after the first approximate optimization of the design model.
6	Exit after the first update of the analysis model based on the first approximate optimization of the design model.
7	Compute and output design sensitivity coefficients at the end of normal program termination: hard convergence, soft convergence, or maximum design cycles. This is equivalent to the DSAPRT (UNFORM,START=LAST) Case Control command.
8	Exit after writing out the data blocks and matrices requested for the Simcenter FE Model Update product. The documentation for this product is included in the Help.

OPTION Default = ABS

See SCRSPEC.

OQG Default = YES

See POST.

OSWELM Default=-1

Defines the offset for identification numbers of internally generated m-set constraints for CWELD elements which use the ELEMID or GRIDID formats.
= -1 (default) The numbering starts with SYSTEM(182) + 1. SYSTEM(182) defaults to 100,001,001.

> 0 The numbering starts with OSWELM + 1.

For each CWELD element, two RWELD constraint elements are generated if MSET=ON is specified. See the MSET field on the PWELD bulk entry.

OSWPPT Default= -1

Defines the offset for identification numbers of the internally generated grids GA and GB for CWELD elements which use the ELEMID or GRIDID formats.

= -1 The numbering starts with SYSTEM(178) + 1. SYSTEM(178) defaults to 101,000,000.

> 0 The numbering starts with OSWPPT + 1.

For each CWELD element, the grid points GA and GB are generated internally if GA and GB are not specified. See the Bulk Data entry CWELD.

OUG Default = YES

See POST.

OUGCORD See POST.

OUMU Default = YES

See POST.

OUNIT1 Default = Value of OUNIT2

See POST.

OUNIT2 Default = 12

See POST.

OUTOPT Default = 0

See CURV.

P – Parameters

PANELMP Default = 0

See FLUIDMP.

PATVER Default = 3.0

See PARAM, POST,-1.

PBRPROP Default = NO

Specifies the method used to calculate cross sectional properties for PBARL and PBEAML entries.
If PBRPROP = YES, the mesh-based Pilkey method is used.
If PBRPROP = NO, hard-coded formulas are used.

For additional information, see the “1D Elements” section of the *Element Library Reference*.

PDRMSG Default = 1

PDRMSG controls the printout of messages associated with deformed plots, including error messages. PDRMSG = 0 suppresses the printout. Contour values will not be displayed unless the default value is used.

PHIBGN Default = 0.0

Beginning azimuth angle (in degrees)

PHIBGN, PHIDEL, and PHINUM define the range of azimuth angle over which the equations of motion with time-dependent coupling terms are solved during a rotor dynamic analysis.

PHIDEL Default = 0.0

Azimuth angle increment (in degrees)

See PHIBGN.

PHINUM Default = 1

Number of azimuth angle increments

See PHIBGN.

PKRSP Default = -1

If PKRSP=0, the magnitude of the output quantities at the time of peak acceleration of the modal variables is output. The SRSS technique that is used for response spectra is described in “Shock and Response Spectrum Analysis” in the *NX Nastran Advanced Dynamic Analysis User’s Guide*. This option is available only for modal transient analysis.

PLTMSG Default = 1

PARAM,PLTMSG,0 suppresses messages associated with undeformed plot requests, including error messages.

POST Default = 1

PARAM,POST,0, then the following parameters and discussion apply:

The data blocks often used for pre- and postprocessing will be stored in the database and also converted, by the DBC module (see the *NX Nastran DMAP Programmer's Guide*) to a format suitable for processing. These data blocks include input data related to geometry, connectivity, element and material properties, and static loads; they also include output data requested through the Case Control commands OLOAD, SPCF, DISP, VELO, ACCE, THERMAL, ELSTRESS, ELFORCE, FLUX, GPSTRESS, GPFORCE, ESE, GPSDCON, and ELSDCON.

The converted data is written to logical FORTRAN units, which may be assigned to physical files in the File Management Section. The FORTRAN unit numbers are specified by the parameters GEOMU, POSTU, and LOADU. By default, all data is written to the logical FORTRAN unit indicated by GEOMU. If LOADU > 0, static load data may be diverted to another unit indicated by LOADU. If POSTU > 0, then output data requested with the Case Control commands listed above will be diverted to the logical unit indicated by POSTU. See "Database Concepts" in the *NX Nastran User's Guide* for the procedure for assigning physical files.

By default, if converted data already exists in the files indicated by GEOMU, POSTU, and LOADU, then the DBC module will overwrite the old data. If this is not desirable, then PARAM,DBCOVWRT,NO must be entered. The parameters MODEL and SOLID may be used to store more than one model and solution in the graphics database. These parameters are not supported by MSC.Patran.

PARAM,DBCDIAG > 0 requests the printing of various diagnostic messages from the DBC module (see *NX Nastran DMAP Programmer's Guide*) during data conversion. By default, no messages are printed.

PARAM,POST,<0, then the following parameters and discussion apply:

PARAM,POST,-1 outputs the appropriate files for the MSC.Patran NASPAT program. PARAM,POST,-2 outputs the appropriate files for the NX and I-DEAS Dataloader program. PARAM,POST,-4 outputs the files indicated below along with OPHIG for the MSC_NF interface by LMS International. PARAM,POST,-5 outputs the files indicated in the table below along with LAMA and OPHG1 for the FemTools interface by Dynamic Design Solutions. PARAM, POST=-4 and -5 are intended for SOL 103 only.

An OUTPUT2 file for FORTRAN unit 12 in binary format is automatically created in the same directory and with the same name as the input file and with the extension ".op2". For example, if the input file is fender.dat, the OUTPUT2 file will be named fender.op2.

An ASSIGN statement is required in the FMS Section only if neutral file format is desired as follows:

ASSIGN OP2='filename of FORTRAN file' FORM

Geometry data blocks are output with PARAM,OGEOM,YES (except with PARAM,PATVER<3.0) and are written to a FORTRAN unit specified by PARAM,OUNIT1 (Default = OUNIT2) for PARAM, POST = -1, -2, and -4.

PARAM,OUNIT2K (default = 91) specifies the unit number for KELM and KDICT with PARAM,POST,-5. PARAM,OUNIT2M (default = 92) specifies the unit number for MELM and MDICT with PARAM,POST,-5. See the following table for the specific geometry data blocks written for different values for POST.

By default, the EPT and MPT data blocks are output to the .op2 file when either PARAM,POST,-1 or PARAM,POST,-2 or PARAM,POST,-4 is used. PARAM,OEPT,NO can be used to disable the output of the EPT data block to the .op2 file. PARAM,OMPT,NO can be used to disable the output of the MPT data block to the .op2 file.

For SOL 601,N or SOL 701, PARAM,POST,-2 must be specified to generate both geometry and results data blocks in the .op2 file. PARAM,POST,-1 must be specified to generate only results data blocks. Otherwise, no .op2 file is generated.

See also the PARAM,POSTEXT description for additional data blocks written to the .op2 file.

POST				Geometry Data Block	Description
-1	-2	-4	-5		
YES	NO	NO	NO	GEOM1S, GEOM1VU	Grid Point Definitions (Superelement)
NO	YES	YES	NO	CSTM	Coordinate SystemTransformations
NO	YES	YES	NO	GPL	Grid Point List
NO	YES	YES	NO	GPDT	Grid Point Definitions
YES	YES	YES	NO	EPT	Element Properties
YES	YES	YES	NO	MPT	Material Properties
YES	YES	YES	NO	GEOM2	Element Definitions
YES	YES	NO	NO	GEOM3	Load Definitions
YES	YES	NO	NO	GEOM4	Constraint Definitions
YES	NO	NO	NO	DIT	Dynamic Table Input
YES	NO	NO	NO	DYNAMICS	Dynamic Loads Definition
NO	NO	YES	YES	KDICT	Element Stiffness Dictionary
NO	NO	YES	YES	KELM	Element Stiffness Matrices
NO	NO	YES	YES	MDICT	Element Mass Dictionary

NO	NO	YES	YES	MELM	Element Mass Matrices
NO	NO	NO	NO	ECTS	Element Connections
YES	NO	NO	NO	VIEWTB	View Element Table
YES	YES	NO	NO	EDOM	Design Model Input
YES	NO	NO	NO	GEOM2S, GEOM2VU	Same as GEOM2 for superelements
YES	NO	NO	NO	CSTMS	Same as CSTM for superelements
YES	NO	NO	NO	EPTS	Same as EPT for superelements
YES	NO	NO	NO	MPTS	Same as MPT for superelements
YES	YES	NO	NO	CONTACT	Surface contact definition
YES	YES	NO	NO	EDT	Element deformation table
Note: *With PARAM,PATVER,v3.0 (default)					

PARAM,OMACHPR,NO (default) selects the pre-MSC.Nastran Version 69 format for AXIC, BGPDT, CSTM, GEOM1, and GPDT, or the pre-MSC.Nastran 2001 format for EPT and GEOM4. PARAM,OMACHPR,YES selects the current NX.Nastran format for AXIC, BGPDT, CSTM, EPT, GEOM1, GEOM4, and GPDT.

For PARAM,POST = -1 and -2, results data blocks are output to a FORTRAN unit specified by PARAM,OUNIT2 (Default = 12). This parameter is allowed to vary between superelements. In buckling solution sequences (SOL 105), a unique value of OUNIT2 should be specified for the buckling subcase. See also the related parameter OMAXR.

For PARAM,POST = -1 and -2, PARAM,OIBULK,YES can be defined to include the IBULK datablock. (Default = NO) The IBULK datablock is an unsorted copy of the original bulk data including comments. The CASECC datablock immediately follows IBULK in the OP2 file.

By default, under PARAM,PATVER, ≥ 3.0, the displacements are output in the global coordinate system. To output in the basic coordinate system, specify PARAM,OUGCORD,BASIC. Under PARAM,PATVER,<3.0, the opposite is true.

PARAM,POST,-1: Results Data Blocks for MSC.Patran

By default, the following data blocks are output under PARAM,POST,-1. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file. PARAM,PATVER selects the appropriate version of MSC.Patran (Default = 3.0)

PARAM, PATVER		Parameter Name	Case Control	Data Block Name	Description
<3.0	≥3.0				
YES	YES	OQG	SPCFORCE	OQG1	Forces of single-point constraint
YES	NO	OUG	DISP	OUGV1PAT	Displacements in the basic coordinate system
YES	YES	OUG	DISP	OUGV1	Displacements in the global coordinate system
YES	NO	OES	STRESS	OES OESVM	Element stresses (linear elements only)
YES	NO	OEF	FORCE	OEF1	Element forces or heat flux (linear elements only)
YES	YES	OEE	STRAIN	OSTR1	Element strains
YES	YES	OGPS	GPSTRESS	OGS1	Grid point stresses
YES	YES	OESE	ESE	ONRGY1	Element strain energy
YES	YES	OGPF	GPFORCE	OGPFB1	Grid point force balance table
NO	YES	OEFX	FORCE	OEF1X	Element forces with intermediate (CBAR and CBEAM) station forces and forces on nonlinear elements
NO	YES	OESX	STRESS	OES1X	Element stresses with intermediate (CBAR and CBEAM) station stresses and stresses on nonlinear elements
NO	YES	OPG	OLOAD	OPG1	Applied static loads
NO	YES	OCMP	STRESS	OES1C	Ply stresses
NO	YES	OCMP	STRAIN	OSTR1C	Ply strains
NO	YES	none	DISP SPCFORCE FORCE	OUPV1 OQP1 DOEF1	Scaled Response Spectra

			STRESS STRAIN	DOES1 DOSTR1	
			none	LAMA	Nonlinear Buckling
NO	YES	none	DISP OLOAD	OCRUG OCRPG	
NO	YES	none	NLSTRESS	OESNLXR	Nonlinear static stresses
NO	YES	none	BOUTPUT	OESNLBR	Slideline stresses
NO	YES	none	NLLOAD	OPNL1	Nonlinear loads
NO	YES	none	STRESS	OESNLXD	Nonlinear transient stresses
YES	YES	none	MODCON	OUGMC OEFMC OESMC OSTRMC OQGMC	Modal contributions
YES	YES	none	PANCON	OUGPC OUGGC OUGRC	Panel contributions

PARAM,OMACHPR,NO (default) selects the pre-NX Nastran 8.5 format for the MAT10 entry. PARAM,OMACHPR,YES selects the current NX Nastran format for the MAT10 entry.

PARAM,POST,-2: Results Data Blocks for NX and I-DEAS

By default, the displacements are output in the basic coordinate system. To output in the global coordinate system, specify PARAM,OUGCORD,GLOBAL (SOL 601 and 701 displacements are always output in the global coordinate system).

The following data blocks are output when PARAM,POST,-2 is defined. The parameters listed in the first column can be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file.

PARAMeter Name	Case Control	Results Data Block Name	Description
	BCRESULTS	OBC OSPDS	Contact Result Output Request
	GKRESULTS	OGK	Gasket element results

	SHELLTHK	OSHT	Shell thickness results
	RMAXMIN	OUGV1MX OEF1MX OES1MX	Displacement, force, and stress extreme values
	OLOAD	OPG1	Applied loads.
	THERMAL	TOUGV1	Temperature output
	NLSTRESS	OESNLXR	Nonlinear stress
OGPS	GPSTRESS	OGS1	Grid point stresses
OGPF	GPFORCE	OGPFB1	Grid point forces
OQG	SPCFORCE	OQG1	Forces of single-point constraint
OUG	DISPLACEMENT	BOUGV1	Displacements in the basic coordinate system
		BOPHIG	Eigenvectors in the basic coordinate system
		OUGV1	Displacements in the global coordinate system
		TOUGV1	Grid point temperatures
OES	STRESS	OES OESVM	Element stresses (linear elements only)
OEF	FORCE	OEF1	Element forces (linear elements only)
	FLUX	HOEF1	Element heat flux
OEE	STRAIN	OSTR1	Element strains
OESE	ESE	ONRGY1	Element strain energy
OCMP	STRESS	OEFIT	Failure indices
	STRESS	OES1C	Ply stresses
	STRAIN	OSTR1C	Ply strains
OUMU	ESE	ONRGY2	Element kinetic energy
OEFX	FORCE	OEF1X	Element forces (nonlinear elements only)
OESX	STRESS	OES1X	Element stresses (nonlinear elements only)

none	none	ODELBGPD	Shape optimization geometry changes
	MODCON	OUGMC OEFMC OESMC OSTRMC OQGMC	Modal contributions
	PANCON	OUGPC OUGGC OUGRC	Panel contributions
	ELSTRN	ELSTRN	Elastic strain at grid points on elements
	GELSTRN	GELSTRN	Elastic strain at Gauss points.
	GSTRAIN	GSTRAIN	Strain at Gauss points.
	GSTRESS	GSTRESS	Stress at Gauss points.
	GTHSTRN	GTHSTRN	Thermal strain at Gauss points.
	OPRESS	OPRESS	Temperature output on grid points.
	OTEMP	OTEMP	Temperature output on grid points.
	THSTRN	THSTRN	Thermal strain at grid points on elements.

PARAM,OMACHPR,NO (default) selects the pre-NX Nastran 8.5 format for the MAT10 entry. PARAM,OMACHPR,YES selects the current NX Nastran format for the MAT10 entry.

PARAM, POST, -4: Results Data blocks for LMS International/MSC_NF

By default, the following data blocks are output under PARAM,POST,-4. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OUG	DISPLAC	OPHIG	Eigenvectors in the global coordinate system

PARAM, POST, -5: Results Data blocks for Dynamic Design Solutions/FemTools

By default, the following data blocks are output under PARAM,POST,-5. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For

example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file. PARAM,OUNIT2O (default=51) specifies the unit number of the OUTPUT2 file.

PARAMeter name	Case Control	Data Block Name	Descriptions
OUG	DISPLAC	OUGV1	Eigenvectors in the global coordinate system
		LAMA	Eigenvalue summary

POSTEXT Default=YES

Under PARAM,POST,-1 and -2, the following data blocks are also written to the.OP2 file specified by PARAM,OUNIT2. If these data blocks are not required, then specify PARAM,POSTEXT,NO.

Data Block Name	Description
FRL	Frequency response list
BHH	Modal damping matrix
KHH	Modal stiffness matrix
BGPDT*	Basic grid point definition table
PVT0	User parameter value table
EQEXIN(S)	Equivalence external to internal grid ID table
CLAMA	Complex eigenvalue table
OEDE1	Element energy loss output table
OEKE1	Element kinetic energy output table
OUGV2	Displacement output table in SORT2
PSDF	Power spectral density table
OGPWG	Grid point weight generator output table
TOL	Time output list
PSDFH	Power spectral density table for modal coordinates
DSCM2	Design sensitivity coefficient matrix
DSCMCOL	Design sensitivity parameters table
DBCOPT	Design optimization history table
OPHSA	Displacement output table in SORT1
* Always written for PARAM,POST,–2 regardless of POSTEXT.	

POSTOPT Default = 2

The POSTOPT parameter is only supported by SOL 401 in the context of the Simcenter Multiphysics environment. In this environment, you can request results output at specific time steps. The POSTOPT parameter then controls if NX Nastran writes the requested results after each time step in which output was requested, or at the end of the subcase for the time steps in which output was requested. This control pertains to F06, OP2, and PUNCH file output requests. If a solution is successful at all time steps, the resulting output files will be the same whether you had selected POSTOPT=1 or POSTOPT=2.

1: NX Nastran writes the requested results to the appropriate output files after each time step in which output was requested.

2: NX Nastran writes the requested results to the appropriate output files at the end of the subcase for the time steps in which output was requested.

POSTU Default = -1

See PARAM, POST,0.

PREFDB Default = 1.0

See ACOUT.

PRGPST Default = NO

Controls the printout of singularities. See AUTOSPC.

PRINT Default = YES PARAM,PRINT,NO suppresses the automatic printing of the flutter summary in flutter analysis.

PROUT Default = -1

PARAM,PROUT,-1 suppresses execution and printout from the ELTPRT module. PARAM,PROUT,1 prints a list of all elements sorted on EID and summary tables giving the range of element identification numbers for each element type.

PRPA, PRPJ Default = 1.0E37

PRPA and PRPJ control the printout of intermediate load matrices for diagnostic purposes during superelement assembly. If the value of PRPA (or PRPJ) is positive, all terms larger in magnitude than the value are printed. If the value of PRPA (or PRPJ) is negative, all terms smaller in magnitude than the value are printed. The default value requests no printout. PARAM,IRES,1 must be present for these parameters to be effective. The PA matrix contains the internal loads transmitted to the downstream superelement. The PJ matrix contains external loads applied on the superelement; that is, it has the same content as the data produced by the Case Control command OLOAD. All of this data may be obtained on restart using the SELR Case Control command option. A related parameter is IRES.

PRTCSTM See CHECKOUT.

PRTEQXIN See CHECKOUT.

PRTGPDT See CHECKOUT.

PRTGPL See CHECKOUT.

PRTGPTT See CHECKOUT.

PRTMAXIM Default = NO

PRTMAXIM controls the printout of the maximums of applied loads, single-point forces of constraint, multipoint forces of constraint, and displacements. The printouts are titled “MAXIMUM APPLIED LOADS”, “MAXIMUM SPCFORCES”, “MAXIMUM MPCFORCES”, and “MAXIMUM DISPLACEMENTS”.

PRTMGG See CHECKOUT.

PRTPG See CHECKOUT.

PRTRESLT Default = YES

PRTRESLT controls the printout of the resultants of applied loads, single-point forces of constraint, and multipoint forces of constraint. The printouts are titled “OLOAD RESULTANTS”, “SPCFORCE RESULTANTS”, and “MPCFORCE RESULTANTS”.

Q & R – Parameters

Q

Default = 0.0

Q specifies the dynamic pressure. Q must be specified in aeroelastic response analysis (SOL 146), and the default value will cause a User Fatal Message.

QSETREM

Default = YES

Specifies whether unused q-set DOF are removed when an external superelement is created with the EXTSEOUT case control command.

If QSETREM = YES, unused q-set DOF are removed from the definition of the external superelement.

If QSETREM = NO, all q-set DOF are retained in the definition of the external superelement. If the model is a mechanism, specifying QSETREM = NO is necessary to retain mechanism modes. However, specifying QSETREM = NO can lead to slow processing times for stresses, strains, and forces, especially if a large number of q-set DOF are defined, but are not associated with calculated modes.

RANCPLX

Default = 0

Specifies the data format for random output. Set to 0 for real data and 1 for complex data.

If the RANCPLX describer is specified on a RANDOM case control command, the RANCPLX describer takes precedence over the RANCPLX parameter.

RANCPLX = 1 is not supported for composites.

RDCNT

Default = YES

Activates RDMODES substructuring for normal modes solutions that include contact.
If RDCNT = YES, use RDMODES substructuring.
If RDCNT = NO, do not use RDMODES substructuring.

For additional information, see the “Recursive Domain Normal Modes Analysis (RDMODES)” section of the *Parallel Processing Guide*.

RDRESVEC Default = YES

Determines if accelerated residual vector calculation option is enabled or disabled when using the multi-level Recursive Domain Lanczos method (RDMODES). See the *NX Nastran Parallel Processing Guide* for details.

RDRESVEC=YES
The accelerated residual vector calculation option is enabled, which takes advantage of the rdsparse option, and is more efficient than the original one in terms of computational time and I/O usage. The residual vectors with the accelerated calculation may differ slightly from the original, which cannot be used in conjunction with rdsparse.

RDRESVEC=NO
The accelerated residual vector calculation option is disabled. When disabled, the rdsparse option will be disabled automatically and full results are computed which typically increases runtime considerably.

RDSPARSE Default = YES

Determines if the sparse eigenvector data recovery option is enabled or disabled when using the DMP, multi-level Recursive Domain Lanczos method (RDMODES). See the *NX Nastran Parallel Processing Guide* for details.

RDSPARSE=YES
The sparse eigenvector recovery option is enabled. The software automatically determines if the sparse eigenvector recovery option occurs based on the number of DOF in the output request.

RDSPARSE=NO
The sparse eigenvector recovery option is disabled (reverts to the data recovery used before NX Nastran 7). When disabled, full results are computed which typically increases runtime considerably.

RESLTOPT Default = 8

RESLTOPT's default value provides component-level force summary output for model checkout (PARAM, CHECKOUT, YES), loads generation (OLOAD output), and SPC and MPC forces. Setting RESLTOPT to a value of 1 produces abbreviated output formats only.

RESVALT

Default = NO

RESVALT determines if residual vector modes contribution to the dynamic physical response is taken into consideration or not.

RESVALT= YES

Residual vector modes contribution to the dynamic physical response is not taken into consideration.

RESVALT=NO (Default)

Residual vector modes contribution to the dynamic physical response is taken into consideration.

RESVALT is considered only if RESVEC=YES is selected. RESVALT is only available for SOL 112, and SOL 200 when ANAYLSIS=MTRAN.

RESVEC

Requests residual vectors for DOF associated with applied loads.

The default behavior for requesting residual vectors is described on the RESVEC case control command. This behaviour occurs when neither the RESVEC case control command, or any residual vector parameters are defined. To alter the default behaviour, it is recommended that you use the RESVEC case control command, and not the parameter inputs.

If both the RESVEC case control command, and any residual vector parameter inputs are defined, the RESVEC case control command takes precedence, and the parameter settings are ignored.

If residual vector parameter inputs are defined, and the RESVEC case control command is undefined, the parameters will take precedence over the defaults described on the RESVEC case control command.

To turn the residual vector request off, you can either include the case control command RESVEC=NO, or you can include the parameter settings PARAM,RESVEC,NO and PARAM,RESVINER,NO.

The residual vector parameter inputs request the following:

PARAM setting	Description
PARAM,RESVEC,YES	Requests residual vectors for DOF associated with applied loads. Also requests residual vectors for all DOF included in USET,U6 and SEUSET,U6 bulk entries.
PARAM,RESVINER,YES	Requests residual vectors for DOF associated with inertia relief loads (unit acceleration of mass). If

	the model is unrestrained, you will need to supply SUPORTi entries.
PARAM,RESVSO,YES	Reorthogonalizes static shapes with mode shapes. If the stiffness is well-conditioned, this is not required.
PARAM,RESVSE.YES	Prints strain energy of the static shapes.
PARAM,RESVSLI,YES	Removes linearly dependent shapes.

RESVINER

Requests residual vectors for DOF associated with inertia relief loads (unit acceleration of mass).

The default behavior for requesting residual vectors is described on the RESVEC case control command remarks. This behaviour occurs when neither the RESVEC case control command, or any residual vector parameters are defined. To alter the default behaviour, it is recommended that you use the RESVEC case control command, and not the parameter inputs.

See related parameter RESVEC.

RESVPGF

Default = 1.E-6

This parameter is used to filter out small terms in the residual vectors. The default is 1.E-6, meaning that terms in the residual vectors whose absolute values are smaller than 1.E-6 will get filtered out before they are used. To eliminate filtering of residual vectors, you can set RESVPGF to 0.0.

RESVRAT

Default = 1.E8

Value used to filter out residual vector trial vectors that have little independent content. To filter out more residual vector trial vectors, decrease the value.

RESVSE

Default = NO

See related parameter RESVEC.

RESVSLI

Default = YES

RESVSO

Default = YES

See related parameter RESVEC.

RGBEAMA

RGBEAME

RGLCRIT

RGSPRGK

In SOL 401, to compute large displacement effects and thermal expansion, the software internally replaces the RBE2 and RBAR elements with either a stiff beam element or a stiff spring element. A coincident grid tolerance is used to determine if a beam or a spring is used. For the RBAR, if the distance between the connecting grids is less than the tolerance, the stiff spring formulation is used. For the RBE2, if the distance between the grid defined in the GN field on the RBE2 entry, and any of the grids defined in the GM fields on the RBE2 entry, is less than the tolerance, the stiff spring formulation is used. You can optionally define the coincident grid tolerance explicitly with the parameter RGLCRIT. By default, it is automatically computed by the software:

Coincident Grid Tolerance = 1E-6 * LMODEL (units=length)

where LMODEL is the largest dimension of the model determined by the software.

You can optionally define the beam stiffness and area explicitly using the parameters RGBEAME and RGBEAMA, respectively. By default, they are automatically computed by:

Beam Stiffness = 1e+2 * EMAX (units=force/length^2)

Beam Area = (LMODEL * 1e-2)^2 (units=length^2)

where EMAX is the largest Young's modulus in the model. If no material is specified in the model, EMAX is set to 1.0E12.

You can optionally define the spring stiffness explicitly using the parameter RGSPRGK. By default, it is automatically computed by:

Spring Stiffness = EMAX * LMODEL (units = force/length)

RLOOPNEW

Default = NO

For frequency response and complex eigenvalue rotor dynamic analysis, specify PARAM,RLOOPNEW,YES to include gyroscopic and circulation terms in the mass, damping, and stiffness matrices when the analysis is performed in the fixed reference system.

RMSINT Default = LINEAR for the trapezoidal approximation.

RMSINT specifies the interpolation method for numerical integration when computing both RMS (Root Mean Square) and N0 (Number of Zero Crossings or Mean Frequency) from PSDF (Power Spectral Density Function).

RMSINT = LINEAR requests the trapezoidal approximation, which is the existing NX Nastran approach.

RMSINT = LOGLOG requests the Log-Log interpolation.

RMSSF Default = 1.0

The value for RMSSF must be greater than 0.0.

RMSSF scales the RMS and CRMS random results. For example, if PARAM,RMSSF,2.0 is specified, the random results are scaled to 2σ. if PARAM,RMSSF,3.0 is specified, the random results are scaled to 3σ.

If the RMSSF describer is specified on a RANDOM case control command, the RMSSF describer takes precedence over the RMSSF parameter.

RMXPANEL Default = 0

When the RMAXMIN case control command is defined in a SOL 112 run, this parameter can be used to process the data recovery in a series of smaller “panels” rather than the entire output set at once. This will reduce the overall amount of scratch disk space required for the run.

“PSIZE” is the number of timesteps to process in a single panel. The highwater disk space used will be a function of psize, and not of the total output request. A good initial value for psize is the greater of 100 or the square root of the number of timesteps. Setting psize =0 (default) will disable the panel method and process the data recovery for all the time steps at once. The START, END, and RMXTRAN=YES keyword options on the RMAXMIN command are not supported when using RMXPANEL. The RMXPANEL method is not compatible with PARAM,DDRMM,-1.

RMXTRAN Default = NO

Used in conjuction with the RMAXMIN case control to allow the user to specify whether transient results are output along with the computed RMAXMIN results. If RMXTRAN=NO (the default value), transient results are not output if RMAXMIN is activated. IF RMXTRAN=YES, transient results will be output according to the DISPLACEMENT, STRESS, and FORCE case control entries.

ROTCMRF Default = 0.0

Specifies the reference rotor speed that is used to compute the reduced modal basis for a SOL 107 rotor dynamic solve with complex modal reduction. Solutions at other reference rotor speeds are computed from the reduced modal basis. Expect the most accurate rotor dynamic analysis results at reference rotor speeds close to the reference rotor speed specified with ROTCMRF.

If rotor superelement reduction is requested with ROTSE bulk entries, unsymmetric a-set reduction is performed at the reference rotor speed specified with ROTCMRF.

The units for the reference rotor speed specified with ROTCMRF are the same as those specified in the RUNIT field of the ROTORD bulk entry.

ROTCOUP No default

Specifies the coupling points for each rotor in a SOL 107 – 112 rotor dynamic analysis.

- If the model contains a single rotor, specify the coupling point with PARAM,ROTCOUP,*gridid*, where *gridid* is the grid ID of the coupling point.
- If the model contains multiple rotors, specify the coupling points with PARAM,ROTCOUP,*setid*, where *setid* is the identification number of a SET case control command that lists the grid ID of the coupling point for each rotor.

For a grid listed on the ROTCOUP parameter specification or in a referenced SET command to be a valid coupling point, it must also be listed on a ROTORB bulk entry.

ROTCSV No default.

Defines the unit number of a CSV (.csv) comma-separated ASCII file to which Campbell diagram results from a complex eigenvalue rotor dynamics analysis are written. Requires:

```
ASSIGN OUTPUT4 = 'filename,csv', UNIT = unit_number, FORM = FORMATTED
```

ROTGPF No default.

Defines the unit number of a GPF (.gpf) file to which Campbell diagram results from a complex eigenvalue rotor dynamics analysis are written. Requires:

```
ASSIGN OUTPUT4 = 'filename.gpf', UNIT = unit_number, FORM = FORMATTED
```

ROTSYNC Default=YES

When ROTSYNC=NO, synchronous analysis in a complex eigenvalue rotor dynamics analysis is skipped.

RPOSTS1 Default = 0

Specifies the output format for random results. Set to 0 for SORT2 and 1 for SORT1.

If the RPOSTS1 describer is specified on a RANDOM case control command, the RPOSTS1 describer takes precedence over the RPOSTS1 parameter.

RSATT Default = NO

PARAM,RSATT,YES specifies that if static loads or U3 USET bulk entries are present, SOL 103 will compute attachment modes. RSATT is only used by I-DEAS Response Analysis and Response Simulation. The attachment modes are written to the OP2 file when PARAM,RSOPT,1 and PARAM,POST,-2 are defined.

RSCON Default = NO

PARAM,RSCON,YES specifies that if U2 USET bulk entries are present, SOL 103 will compute constraint modes. RSCON is only used by I-DEAS Response Analysis and Response Simulation. The constraint modes are written to the OP2 file when PARAM,RSOPT,1 and PARAM,POST,-2 are defined.

RSOPT Default = 0

PARAM,RSOPT,1 and PARAM,POST,-2 specifies that extra data blocks associated with I-DEAS Response Analysis and Response Simulation are written to the OP2 file.

RSPECTRA Default = -1

RSPECTRA = 0 requests that response spectra be calculated for transient analysis. See “User Interface for Response Spectra Generation” in the *NX Nastran Advanced Dynamic Analysis User’s Guide*. Response spectra will be calculated for any superelements or the residual structure for which other output requests are present in the same run. Any punch data produced is sent to the standard NX Nastran PUNCH file. Related parameters are TABID and RSPRINT.

RSPRINT Default = 0

RSPRINT controls the printout of tabulated values of response spectra. RSPRINT = -1 suppresses the printout. The related parameter is RSPECTRA.

S – Parameters

S1G, S1M Default = -1

PARAM,S1i,+1 requests the sorting and/or filtering of the element stresses selected on the DTI,INDTA entry. Stresses at grid points (S1G) and/or in material coordinate systems (S1M) based on the parameters BIGER, NUMOUT, SRTOPT, and SRTELTYP may be requested. The S1G and S1M options also require the presence of PARAM,CURV,1.

Parameter	Quantity	Coordinate System	Location	Elements
S1M≥0	Stresses	Material	Element centers	CQUAD4, CTRIA3
S1G≥0	Stresses	Material	Grid points to which elements connect	CQUAD4, CTRIA3

NUMOUT, in conjunction with BIGER, controls the amount of stress output.

1. NUMOUT = +N requests that N element stresses be printed (or punched) for each element type.

2. NUMOUT=0 outputs all elements in a group when one or more exceeds BIGER. Some of the elements will have stresses smaller than BIGER. This is conceptually the same as describing an element set in case control, and limiting output in this manner. Stress files obtained with element group filtering may be used for xy plotting and other postprocessor options with DMAP alters. By contrast, the stress file when NUMOUT = -2 is more discontinuous, and may not be used for xy plotting.

3. NUMOUT = 0 does not sort but filters according to BIGER by element group. In static analysis, an element group is defined as all case control selected elements for a given load case for SORT1 output. For SORT2 output, an element group is defined as the data for a given element type for all load cases. In transient analysis, an element group is defined as all case control selected elements at a given time for SORT1 output. For SORT2 output an element group is defined as the data for a given element at all time steps. The element group option applies only to output types described above for PARAM,S1. This option is not available with output types selected by PARAMs S1G and S1M.

4. NUMOUT = -1 requests that stresses be sorted and only those stresses with an absolute value that is greater than BIGER will be output.

5. NUMOUT = -2 (the default) does not sort but filters according to BIGER. Related parameters include BIGER, NOELOF, NOELOP, and NOGPF.

BIGER controls the elements for which stresses will be printed. Elements with stresses that are smaller in absolute value than BIGER will not be output. The quantity tested is element type dependent. Related parameters include CURV, NUMOUT, S1G, and S1M. SRTOPT controls the scanning option to be performed.

SRTOPT Value	Description
0	Filter/sort on maximum magnitude.
1	Filter/sort on minimum magnitude.
2	Filter/sort on maximum algebraic value.
3	Filter/sort on minimum algebraic value.

SRTELTYP controls the element type to be processed, as described in the following table.

SRTELTYP Value	Description
0	All element types will be processed.
> 0	Only element type SRTELTYP will be processed.

NUMOUT1 and BIGER1 serve the same function as NUMOUT and BIGER except that they apply only to composite element stresses and do not require PARAM,S1i,=1.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,=1.

S1AG, S1AM Default = -1

See CURV.

SCRSPEC Default = -1 (SOLs 103 and 115 only)

SCRSPEC=0 requests that structural response be calculated for response spectra input in normal modes analysis. See “Response Spectrum Application” in the *NX Nastran Advanced Dynamic Analysis User’s Guide* for a discussion of this capability. The scaled response calculations are made for elements and grid points of the residual structure only. The responses are summed with the ABS, SRSS, NRL, or NRLO convention, depending on the value of PARAM,OPTION. If the SRSS, NRL, or NRLO options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies meet the inequality

$f_{i+1} < \text{CLOSE} \cdot f_i$

Both PARAM,OPTION and PARAM,CLOSE may be set in any subcase, allowing summation by several conventions in a single run.

In MSC.Nastran Version 70, the NRL option was modified slightly to correspond to the NAVSEA-0908-LP-000-3010 specification. NRLO provides the V69 NRL.

SDAMPUP Default = NO

In a modal frequency response solution (SOL 111) which includes elements with frequency dependent stiffness (PELAST and PBUSHT entries), determines if natural frequencies are updated before modal damping is computed.

= NO Natural frequencies and eigenvectors are not recomputed using the frequency dependent stiffness. Instead, they are computed one time, using the nominal stiffness values defined on PELAS or PBUSH. Modal damping is then computed using these less accurate frequencies.

= YES The software updates the frequency-dependent modal stiffness and recomputes the natural frequency and eigenvector while remaining in modal coordinates (h-set). This more accurate frequency is used to compute the modal damping for each frequency response calculation. This option improves accuracy, but may also increase runtime substantially, particularly if the number of modes is large. See Damping in Modal Frequency Response in the *Basic Dynamic Analysis User's Guide*.

SECOMB Default = NO

By default, the results from superelements will not be combined for a superelement analysis. By specifying PARAM,SECOMB,YES, the results from superelements will be combined for a superelement analysis with the following conditions/restrictions:

- Instead of outputting results by subcase, the output results for all subcases will be unioned and output with the residual subcase. Results that are excluded are: (1) any XY results, (2) MPF (i.e. modal participation factors), (3) modal energies, and (4) SDISPLACEMENT/SVELOCITY/SACCELERATION.
- Output for each superelement is still defined based off of the output request specified in the superelement subcase. If the constraint mode method of enforced motion is being used and relative displacement/velocity/acceleration results are requested, the requests specified for the residual will take precedence when SECOMB is set to YES.
- This applies to both internal and external superelements defined in both the main bulk data and as a partitioned superelement.
- If partitioned superelements have grid or element labeling conflicts, PARAM,SECOMB,YES is not valid and results combination will not be allowed.
- If complex displacement, velocity, or acceleration output is requested, the requests must be all real/imaginary or all magnitude/phase. If a mix of real/imaginary and magnitude/phase is requested, results combination will not be allowed.
- PARAM,OMID,YES is not valid if PARAM,SECOMB,YES is valid.
- If the OMODES case control command is used, OMODES must be the same for all superelements.

SEMAP Default = SEMAP

SEMAPOPT Default = 42

SEMAPPRT Default = 3

The superelement map (SEMAP table) contains several lists useful for determining how the program has partitioned superelement models. It is printed automatically each time this table is generated. It consists of three major parts:

GPM – The Grid Point Map contains a list of each grid point, its interior superelement placement, and the SEID of all grid points connected directly to it by elements. Three tables follow that summarize the connectivity between superelements sorted on grid point sequence, SEID, and the number of connections.

ISM – The Individual Superelement Map lists the interior grid points, exterior grid and scalar points, elements, and time and storage space estimates for each superelement.

SDT – The Superelement Definition Table contains the SEID of every superelement in the model, the processing order, and a pictorial representation of the superelement tree.

SEMAP, SEMAPOPT, and SEMAPPRT are used to control the amount of output that is printed and other special features. The possible values for SEMAP are shown in the following table.

SEMAP Value	Output and Application
SEMAP (Default)	ISM, SDT. The lengthy GPM is suppressed. This is the appropriate value for use after the model is stable and only minor changes are to be made.
SEMAPALL	GPM, ISM, SDT. All tables are printed. This value is useful on the initial debug run of a model and when making extensive modeling changes.
SEMAPCON	Only the summary tables of the GPM and the estimation data is output. This is a useful value when iterating to an economic partitioning scheme for large, complex models.

SEMAPEST	Only the estimation data is printed. This is useful when evaluating several alternative partitioning schemes.
SEMAPPUN	No output is printed. The exterior grid points of the superelement with a SEID that is input on SEMAPOPT are placed on a CSUPER entry image on the PUNCH file, allowing the superelement to be used as an external superelement. If SEMAPOPT > 0, the superelement entry is given an SSID of SEMAPOPT. If SEMAPOPT < 0, the exterior points listed are those of the residual structure, but the CSUPER entry is given an SSID of |SEMAPOPT|.
Other special features are available with parameters SEMAPOPT and SEMAPPRT. They are fully described under parameters OPT1 and OPT2 in the description of the TABPRT module in the *NX Nastran DMAP Programmer's Guide.* If the default value of SEMAP is used, the other two parameters may be used to further refine this output, as described in the *NX Nastran DMAP Programmer's Guide* under the TABPRT module description. The printing of the SEMAP table can be avoided by the use of PARAM,SEMAPPRT,-1.	

SENSUOO Default = NO

By default, in dynamic sensitivity analysis in SOL 200, displacements at the o-set due to pseudo-loads do not include any effect due to inertia and damping. If PARAM,SENSUOO,YES is specified then these effects will be computed in a quasi-static manner and included in the sensitivity analysis.

SEOP2CV Default = 0

Use to override the effect of PARAM,OP2FMT when the .op2 file is created by the EXTSEOUT case control command on ILP64.

0: No data conversion. All data in 64-bit format.

1: Convert data to 32-bit format.

2: Allow PARAM,OP2FMT setting to control .op2 file format.

SEP1XOVR Default = 0

The old and new location of moved shell grid points are printed if SEP1XOVR = 16 when the RSSCON shell-to-solid element connector is used. By default, the moved shell grid points are not printed, SEP1XOVR = 0. See the description of TOLRSC for more details.

SEQOUT Default = 0

See OLDSEQ .

SERST Default = AUTO (structured solutions only)

By default, in the structured solutions, all restarts are considered automatic (see “Restarts”in the *NX Nastran User’s Guide*). If none of the following Case Control commands are entered, then SEALL=ALL is the default action: SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL.

The user may wish to partition the analysis into several runs, in which case the above commands may be used in the same manner they are used in the unstructured solutions. By default, the restart will proceed in automatic fashion for each command, regenerating only that data that is affected by modifications in the Bulk Data and Case Control or changes in upstream superelements. If the user wishes to overwrite the data, even if it is not affected by modifications to the data, then PARAM,SERST,MANUAL must be entered.

With PARAM,SERST,AUTO or MANUAL, all superelements will be processed through Phase 0 (see “Superelement Analysis” in the *NX Nastran User’s Guide*). This phase includes execution of the sequencer module (SEQP), initial superelement processing (SEP1), and initial geometry processing (GP1 and GP2) modules, which can result in significant CPU overhead. If this overhead is not desired,

then PARAM,SERST,SEMI will limit Phase 0 and Phase 1 to only those superelements specified on the SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL Case Control commands. If none of these commands is entered, then execution will skip Phase 0 and 1.

In the modal solution sequences (SOLs 110, 111, 112, 145, 146, and 200), the modes of the residual structure are automatically computed in Phase 2 if any SE-type command (e.g., SEMG=n) is requested for the residual structure. If PARAM,SERST,SEMI and no SE-type command is specified for the residual structure, then, by default, its modes will not be recomputed. This logic is intended for restarts from SOL 103 into one of the modal solutions. If, however, the modes have not already been computed or need to be recomputed, then PARAM,SERST,RSMDS must be specified to force the calculation of the residual structure modes.

If PARAM,SERST,SEDR is specified, then Phases 0, 1, and 2 will be skipped. This option is intended for data recovery (Phase 3) runs only.

The options of SEMI, RSMDS, and SEDR are intended for models that are defined on more than one database; i.e., superelements are defined on separate databases (multiple MASTER DBsets) and processed in separate runs. Also, with this technique, databases are attached with the DBLOCATE File Management statement rather than the RESTART File Management statement. In general, these options are not recommended because they disable the automatic restart capability, which compromises the database integrity to the same degree as in the unstructured solution sequences.

SESDAMP Default = NO

Modal damping is calculated for superelements if PARAM,SESDAMP,YES is specified. An SDAMPING Case Control command that selects a TABDMP1 Bulk Data entry must also be specified in the desired superelement's subcase. By default, modal damping is added to viscous damping (B). If you insert PARAM,KDAMP,-1 (or PARAM,KDAMPFL,-1 for fluid superelements) then modal damping will be added to structural damping (K4).

SESEF

Default = -1 (SOLs 103 and 115 only)

If SESEF = 0 in superelement normal modes analysis, the fraction of total strain energy for a superelement in each of the system's modes is output in the vector SESEFA for tip superelements and in SESEFG for nontip superelements. If SESEF = 1, strain energy fractions are output, and expansion of the eigenvectors from a-set size to g-set is branched over for tip superelements. If SESEF = -1 (the default value), no strain energy fractions are computed.

Output requests must be present in order for strain energy fractions to be calculated. If SESEF = 1, there will be no other output results for tip superelements.

SFEF70

Default = NO

Requests the import of a fluid-structure coupling matrix created by SFE AKUSMOD when YES. When NO, the coupling matrix is not imported.

SHIFT1

Default=-1.234

This sets the negative shift used when computing massless mechanism constraint modes. For a very stiff model (1000. hz for the first flexible mode), consider using a larger value.

SHLDAMP

Default=SAME

Determines if the structural damping coefficient (GE) defined on the PSHELL MIDi materials are used.

SAME: The structural damping coefficient (GE) defined on a PSHELL MID1 material will be used by all MIDi for that PSHELL.

DIFF: The structural damping coefficient (GE) defined on each PSHELL MIDi will be used. When each PSHELL MIDi is used as described, any structural damping coefficient (GE) values which are blank default to zero.

SIGMA Default = 0.0

The radiant heat flux is proportional to SIGMA · $(T_{grid} + T_{ABS})^4$, where SIGMA is the Stefan-Boltzmann constant, T_{grid} is the temperature at a grid point, and T_{ABS} is the scale factor for absolute temperature and may be specified by PARAM,TABS. These parameters must be given in units consistent with the rest of the data in the model. The value for SIGMA in SI units is 5.67×10^{-8} watts/m$^2K^4$. The default value causes radiant heat effects to be discarded.

SKPAMP Default = 0

For SOLs 145, 146, and 200, SKPAMP = -1 suppresses all unsteady aerodynamic calculations. The automatic restart of the structured solution sequences performs a similar function without this parameter. Specifying it ensures suppression of the calculations, regardless of the determination of the automatic restart.

SLOOPID Default = 0 (SOL 129 and 159 only)

In a nonlinear transient analysis (SOLs 129 and 159) restart, SLOOPID identifies the initial conditioning previous nonlinear analysis run (SOLs 106 and 153 respectively). Setting SLOOPID greater than 0 will cause SOLs 129 and 159 to start from the static deformed position.

SMALLQ Default = 1.0E-10

By default, NX Nastran does not remove unused q-set degrees-of-freedom from the solution set. If you are performing residual structure qms you should set this parameter to 0.0 so that the unused q-set degrees-of-freedom are removed.

SNORM Default = 20.0

SNORM > 0.0 requests the generation of unique grid point normals for adjacent shell elements (see Figure 6-1). Unique grid point normals are generated for the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements. The grid point normal is the average of the local normals from all adjacent shell elements including CQUAD8 and CTRIA6 elements. If grid point normals are present, they are used in all element calculations of the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

SNORM	Tolerance in Degrees
> 0.0	Unique grid point normals are generated if each angle between the grid point normal and each local normal of the adjacent shell elements is smaller than SNORM. SNORM Bulk Data entries overwrite a generated normal.
= 0.0	The generation of grid point normals is turned off. The user can define normals with the SNORM Bulk Data entry.
< 0.0	Grid point normals are not generated. SNORM Bulk Data entries are ignored.

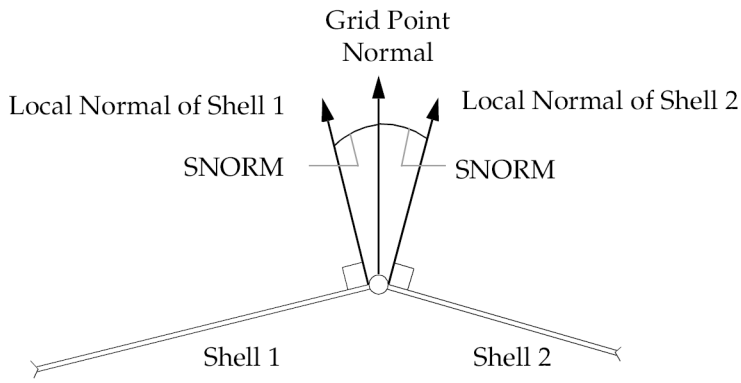


Figure 6-1. Unique Grid Point Normal for Adjacent Shell Elements

SNORMPRT Default = -1

PARAM,SNORMPRT,>0 writes the grid point normals of the model in the basic coordinate system to the.f06 and/or.pch files.

SNORMPRT	Switch to Print Out Normals
≤ 0	No output.
1	Print out to the punch file (.pch).
2	Print out to the print file (.f06).
3	Print out to the punch (.pch) and print file (.f06).

SOFTEXIT Default = NO

In SOL 200, if soft convergence is achieved during optimization, before completing the maximum number of design iterations, the user may request an exit with PARAM,SOFTEXIT,YES.

SOLID Default = 0

SOLID also allows several models to be stored in the same graphics database created by PARAM,POST,0.

SOLVSUB Default: 1

Valid only when PARAM,CORROPT,YES exists.

0: Skip eigenvalue solution.

1: Perform eigenvalue solution.

SPCGEN Default = 0

SPCGEN requests the creation of SPCi Bulk Data entries from the identified singularities. See AUTOSPC.

SPCSTR Default = NO

Used for fluid/structure analysis only.

When surface vibrations (displacement, velocity, or acceleration) on the structure are known and defined as an input excitation, you may only need the structural DOFs in order to enforce this loading to the acoustic mesh. You can set the SPCSTR parameter to YES to constrain the structural DOFs in the analysis set, but only after the structural excitation is transferred over to the fluid.

All structural DOFs are constrained by using SPCs, and SPCDs are applied as the enforced motion.

SRCOMPS Default = NO

SRCOMPS controls the computation and printout of ply strength ratios. If SRCOMPS = YES, ply strength ratios are output for composite elements that have failure indices requested.

For shell composites, when PARAM,SRCOMPS,YES is defined, the software computes both the failure index and the strength ratio.

For solid composites, when PARAM,SRCOMPS,YES is defined, the software only computes the strength ratio, and not both the failure index and the strength ratio.

SRTELTYP Default = 0

See S1, S1G, S1M.

SRTOPT Default = 0

See S1, S1G, S1M.

START Default = 0

See OLDSEQ .

STIME Default = 0.0 (SOLs 129 and 159 only)

In SOLs 129 and 159 restarts from previous SOLs 129 and 159 runs, the user provides t_N , where t_N is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from t_N as though the original run contained the new subcase data.

STRUCTMP Default = 0

See FLUIDMP.

SUBID Default = 1 (SOLs 106 and 153)

In SOL 106, by default, the restart proceeds from the last LOOPID in the last subcase. SUBID may be used to specify an earlier subcase by specifying the sequential number (for SEID = 0) of the subcase. In SOLs 106 and 153, PARAM,LOOPID may also be specified for an earlier LOOPID.

SUPER Default = 0 (nonsuperelement sequences)

Default = -1 (superelement sequences)

See OLDSEQ.

SWPANGLE Default = 1.0

The angular increment in degrees at which failure indices and strength ratios are computed and output for laminates in SOL 108 and 111.

T – Parameters

TABID Default = 2

TABID controls the punch output for response spectra. A related parameter is RSPECTRA.

TABS

Default = 0.0

TABS is used to convert units of the temperature input (°F or °C) to the absolute temperature (°R or °K). Specify:

PARAM,TABS,273.16	When Celsius is used
PARAM,TABS,459.69	When Fahrenheit is used

For SOL 601,153 and 601,159, the value of TABS must be 0.0 (absolute temperature), 273.15 (Celsius) or 459.67 (Fahrenheit).

Refer to the Bulk Data entry, "CREEP" for a creep analysis with SOLs 106 or 153. Refer to PARAM,SIGMA for heat transfer analysis.

TESTNEG

Default = –2 for Newton's method in SOLs 106 and 153

Default = 1 for Arc-length method in SOLs 106 and 153

In nonlinear static analysis, this parameter specifies the action to take when negative terms are encountered on the factor diagonal of matrix decomposition. Negative terms indicate that the differential stiffness has introduced a structural instability. The instability may be real (structural buckling) or mathematical (the current iteration appears unstable, but a stable solution exists).

TESTNEG	Results
-1	Stop if negative terms occur.
1 or 0	Continue if negative terms occur.
-2	If negative terms exist, do not use differential stiffness.
2	Do not use differential stiffness.

TINY

Default = 1.E-3

Any elements with strain energy that is less than a TINY percentage of the total strain energy for any superelement will not be printed or made available for postprocessing. TINY may also be used to suppress the printing of small numbers in the constraint check matrix [E_{mh}] described in "Geometry Processing in SubDMAP PHASEO" in the *NX Nastran User's Guide*.

TOLRSC

Default = 0.05

When the RSSCON shell-to-solid element connector is used, the connecting grid points of the shell element are moved on to the solid face if the grid points are close enough. The tolerable distance of the shell grid point to the solid edge or face is $\varepsilon \cdot h$ where h is the height of the solid edge; see the sample figure below. The relative tolerance is user modifiable using the parameter.

```
PARAM,TOLRSC, ε
```

The default for the relative tolerance is $\varepsilon = 0.05$. Rigid body invariance is satisfied with double-precision accuracy if the shell grid points are adjusted.

TORSIN

Determines if the torsional mass moment of inertia is included on the CROD and CBAR mass matrices.

0: No torsional mass moment of inertia for CROD and CBAR (default).

1: Include torsional mass moment of inertia for CROD and CBAR.

2: Include torsional mass moment of inertia for CBAR.

3: Include torsional mass for CROD.

TSTATIC

Default = -1 (SOLs 129 and 159 only)

If TSTATIC = 1, a static solution may be obtained while ignoring inertial and damping forces. This option is available only with the adaptive time-stepping method (see METHOD = “ADAPT” on the Bulk Data entry, TSTEPNL).

U – Parameters

UNITSYS

Character-default=“ ”.

UNITSYS is used by Simcenter Pre/Post to identify the units system. It has no impact on the NX Nastran solution sequence. NX Nastran always assumes that units are consistent and no conversions are performed.

The character string generated by Simcenter Pre/Post for UNITSYS has the format of FORCE-LENGTH or FORCE-LENGTH-TEMPERATURE. Time units are always in seconds. All other units are defined in the table below.

Examples:
PARAM,UNITSYS,N-M

PARAM,UNITSYS,N-M-R
PARAM,UNITSYS,N-M-C

Input	Force	Length	Temperature	Mass
N-M N-M-C N-M-F N-M-K N-M-R	Newton (N)	meter (m)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	kilogram (kg)
N-MM N-MM-C N-MM-F N-MM-K N-MM-R	Newton (N)	millimeter (mm)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	tonne (t)
MN-MM MN-MM-C MN-MM-F MN-MM-K MN-MM-R	milli-Newton (mN)	millimeter (mm)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	kilogram (kg)
CN-CM CN-CM-C CN-CM-F CN-CM-K CN-CM-R	centi-Newton (cN)	centimeter (cm)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	kilogram (kg)
KGF-M KGF-M-C KGF-M-F KGF-M-K KGF-M-R	kilogram-force (kgf)	meter (m)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	kgf-sec^2/m
KGF-MM KGF-MM-C KGF-MM-F KGF-MM-K KGF-MM-R	kilogram-force (kgf)	millimeter (mm)	blank = Celsius C = Celsius F= Fahrenheit K = Kelvin R = Rankine	kgf-sec^2/mm
LBF-FT LBF-FT-C LBF-FT-F LBF-FT-K LBF-FT-R	pound-force (lbf)	foot (ft)	blank = Fahrenheit C = Celsius F= Fahrenheit K = Kelvin R = Rankine	slug
LBF-IN LBF-IN-C LBF-IN-F LBF-IN-K LBF-IN-R	pound-force (lbf)	inch (in)	blank = Fahrenheit C = Celsius F= Fahrenheit K = Kelvin R = Rankine	lbf-sec^2/in
PDL-FT PDL-FT-C PDL-FT-F	poundal (pdl)	foot (ft)	blank = Fahrenheit C = Celsius F= Fahrenheit	pound-mass (lbm)

PDL-FT-K PDL-FT-R			K = Kelvin R = Rankine	

UNSYMF Default = No

In SOL 106, nonlinear statics, PARAM,UNSYMF,YES is required to include damping effects in the calculation of complex eigenvalues. See PARAM,NMLOOP.

UPDTBSH Default = NO

UPDTBSH controls the update of boundary shapes generated by auxiliary boundary model analysis in SOL 200. By default, the auxiliary boundary models and shapes are generated only once at the initial design cycle and will not be updated in subsequent cycles even if the shape of the primary model is changing. PARAM,UPDTBSH,YES requests that the auxiliary models and shapes are updated and reanalyzed at every cycle.

USETPRT Default = -1

USETSEL Default = 0

USETPRT controls the tabular printout of the degree-of-freedom sets. See “Degree-of-Freedom Sets”.

Sequence	Print	USETPRT
None	None(default)	-1
Internal	Row sort only	0
	Column sort only	1
	Row and Column sort	2
External	Row sort only	10
	Column sort only	11
	Row and Column sort	12

The degrees-of-freedom can be listed in ascending order according to their internal or external sequence number, but not both. The external sequence number is the grid, scalar, or extra point identification number. The internal sequence number is the number assigned after resequencing (see PARAM,OLDSEQ).

For a given sequence there are two types of tables that may be printed: row sort and column sort. For row sort, a table is printed for each set selected by USETSEL. Here is an example of row sort (USETPRT=0 or 10):

```
U S E T   D E F I N I T I O N   T A B L E   ( I N T E R N A L   S E Q U E N C E ,   R O W   S O R T )
                                    A   DISPLACEMENT SET
        -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-       -10-

1=      2-1       2-2
```

For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT = 1 or 11):

```
U S E T   D E F I N I T I O N   T A B L E   ( I N T E R N A L   S E Q U E N C E ,   C O L U M N   S O R T )

  EXT GP. DOF  INT DOF  INT GP.   SB   SG   L   A   F   N   G   R   O   S   M   E
-------------------------------------------------------------------------------------------------------
     1 - 1     1-      1 G      1                       1   1               1
       - 2     2-               2                       2   2               2
       - 3     3-                    1                  3   3               3
       - 4     4-                    2                  4   4               4
       - 5     5-                    3                  5   5               5
       - 6     6-                    4                  6   6               6
```

USETSEL specifies the sets which will be printed in the row sort (USETPRT = 0 or 10). In order to select specific sets to be printed, you must sum their corresponding decimal equivalent numbers. For example, sets A, L, and R are selected with USETSEL = 128 + 256 + 8 = 392. For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT = 1 or 11):

USETSEL	Sets Printed
-1	All sets as defined in “Degree-of-Freedom Sets”.
0	Mutually exclusive sets only; i.e., sets M, SB, SG, O, Q, R, C, B, E, and A.

USETSTRi Input-character-default=’ ‘

USETSTR1 through USETSTR4 specifies the sets that will be printed by the specification of parameters USETPRT and USETSEL. Any set in Degree-of-Freedom Sets may be specified. A “:” is used as a separator. In the following example, the m-set (degrees-of-freedom eliminated by multipoint constraints) and s-set (degrees-of-freedom eliminated by single point constraints) are specified.

Example: PARAM,USETSTR1,M:S

V – Parameters

VMOPT Default = 0

The VMOPT parameter is only applicable if the fluid is modeled by MFLUID bulk entries that are referenced by a MFLUID case control command.

VMOPT=0 (default) The modes of the structure only are first computed without virtual mass (VM) effects to obtain Ritz vectors to use in a generalized coordinate solution, similar in concept to the older Generalized Dynamic Reduction method. The modes are re-computed in the generalized basis after the VM effects are added. When an o-set exists (usually because of the presence of ASETi-type entries, or for super element reduction), the VM effects are added to the component modes (o-set level). When there is not an o-set the VM effects are added in the residual structure Phase II operations (a-set level), after the structure-mass-only eigensolution. The rigid body mass of the VM is output automatically, computed about the location of the grid point listed on PARAM, GRDPNT.

VMOPT=1, The virtual fluid mass will be included in the mass matrix at the same time as all other mass elements. In other words, the component modes will reflect the virtual mass. By default, virtual mass is included after the component modes are computed. This is the most expensive option in terms of computer resource requirements and computation time. It is practical for only relatively small-size models, or when the VM is in one super element only, with most elements in the super element wetted.

VMOPT=2, This is the more efficient method and is summarized in the following steps:
Step 1: Compute the normal modes of the structure without the fluid mass: the `dry' modes.
Step 2: Generate reduced virtual mass using the reduced, modal basis from the dry modes.
Step 3: Compute the normal modes of the reduced virtual mass problem from step two to produce the `wet' modes.
Distributed memory parallel (DMP) execution is available for both the dry modes calculation (step 1) and the virtual mass generation (step 2) to reduce overall solution time.
The following DMP methods can be used to compute dry modes: RDMODES and FDMODES, GDMODES and HDMODES with GPART=1. The DMP global iterative solver is also enabled for virtual mass generation by specifying ITER=YES on the NASTRAN card.

Note

The VMOPT=2 option should not be used with MAX normalization specified on the EIGRL card. VMOPT=2 will only yield correct results with MASS normalization.

VREF Default = 1.0

In modal flutter analysis, the velocities are divided by VREF to convert units or to compute flutter indices.

W & X – Parameters

W3, W4, W3FL, W4FL Default = 0.0

The viscous damping matrix expressed in physical space for direct structural transient analysis (SOL 109, 129, and 159) is assembled from the equation:

$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}] + \frac{G}{W3}[K^1_{dd}] + \frac{1}{W4}[K^4_{dd}]$$

In coupled fluid-structure analysis, W3 and W4 are applied to the structural portion of the model and W3FL and W4FL to the fluid portion of the model. The default value of 0.0 for W3, W4, W3FL, and W4FL causes the $[K^1_{dd}]$ and $[K^4_{dd}]$ terms to be ignored. G is defined by specifying PARAM,G or PARAM,GFL. $[K^4_{dd}]$ arises from GE on MATi entry and DMIG entry of structural damping. The units of W3, W4, W3FL, and W4FL are rad/sec.

In SOLs 129 and 159, W4 may vary between subcases. However, the linear portion of the model uses only the W4 value from the first subcase and the values in the subsequent subcases are applied to the nonlinear portion of the model.

When specified in a modal transient response analysis (SOL 112) or a modal-based analysis that writes an interface file for use with external multi-body dynamics or control system software, the viscous damping matrix is transformed to modal space. For more information, see the WMODAL parameter.

WMODAL Default = NO

WMODAL specifies a structural-to-viscous damping conversion method that uses the solved modal frequencies as conversion factors. When one or both of the W3 and W4 parameters are also specified (see the W3, W4, W3FL, W4FL parameter), the structural-to-viscous damping conversion is calculated using:

$$[B_{qq}] = [\Phi_{dq}^T]\left([B_{dd}^1] + [B_{dd}^2] + \frac{G}{W3}[K_{dd}^1] + \frac{1}{W4}[K_{dd}^4]\right)[\Phi_{dq}]$$

$$+ \begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \cdots & 0 \\ 0 & 1/\sqrt{\omega_2} & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\sqrt{\omega_q} \end{bmatrix} [\Phi_{dq}^T]\left(G[K_{dd}^1] + [K_{dd}^4]\right)[\Phi_{dq}] \begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \cdots & 0 \\ 0 & 1/\sqrt{\omega_2} & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\sqrt{\omega_q} \end{bmatrix}$$

where ω_i are the modal frequencies in rad/sec and the resulting viscous damping matrix is expressed in modal space. $[B^1_{dd}]$ and $[B^2_{dd}]$ represent viscous damping contributions from damping elements (CVISC, CDAMPi) and DMIG entry of viscous damping, respectively. The remaining terms represent structural damping. $G[K^1_{dd}]$ results from PARAM,G specification. $[K^4_{dd}]$ results from GE on MATi entry and DMIG entry of structural damping.

For rigid body modes, the diagonal entries, $1/\sqrt{\omega}$, are partitioned out of the calculation. The FZERO parameter is used to identify rigid body modes (see the FZERO parameter).

$[B_{qq}]$ can be used directly in a modal transient response analysis (SOL 112). It can also be used in a multi-body dynamics or control system simulation when a normal modes analysis (SOL 103), modal frequency response analysis (SOL 111), or modal transient response analysis (SOL 112) input file contains either the ADAMSMNF or MBDEXPORT case control commands. For a multi-body dynamics simulation, the damping matrix written to the interface file contains only the values along the diagonal of $[B_{qq}]$. For a control system simulation, the state-space representation written to the interface file uses all of $[B_{qq}]$.

WRTMAT Default: 1

PARAM, WRTMAT is valid only for when PARAM,CORROPT,YES is defined.

1: Write reduced A-SET mass (MXX) and stiffness (KXX) matrices to OP2 file.

2: In addition to MXX and KXX, write back expansion matrix (GOA) from A-SET to Display Set to OP2 file.

WTMASS Default = 1.0

The terms of the structural mass matrix are multiplied by the value of WTMASS when they are generated. In coupled fluid-structure analysis WTMASS is applied to the structural portion of the model only. WTMASS applies to MFLUID entries but it is not recommended for use in hydroelastic problems.

XFLAG Default = 2

By default (XFLAG = 2), when temperature loads and element deformations are present, the element strain energy for the linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^T K_e u - \frac{1}{2}u^T P_{et}$$

where u is the deformation, K_e is the element stiffness and P_{et} is the element load vector for temperature differences and element deformations. This formula is the same strain energy calculation used for nonlinear elements.

If XFLAG is set to 0, the element strain energy for linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^T K_e u - u^T P_{et}$$

6.2 Parameter Applicability Tables

Table 6-1 lists parameter applicability to the solution sequences (SOLs 101 through 115).

Table 6-2 lists parameter applicability to the solution sequences (SOLs 116 through 701).

The following nomenclature is used:

B	Must be specified in the Bulk Data Section only.
E	May be specified in either the Bulk Data and/or Case Control Section.
blank	Is not supported for that solution number.

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
ACOUT		E			E	E	E	E	E	E		
ACSYM						B			B			
ADPCON				E								
ADSTAT							B			B		
AESDISC												
AESMAXIT												
AESMETH												
AESRNDM												
AESTOL												
AFNORM		B							B	B		
AGGPCH		E			E	E	E	E	E	E		
ALPHA1					B	B	B	B	B	B		
ALPHA2					B	B	B	B	B	B		
ALTRED	B		B								B	
AMPCZ	B	B	B	B	B	B	B	B	B	B	B	B
ASCOUP		E			E	E	E	E	E	E		
ASING	E	E	E	E	E	E	E	E	E	E	E	E
AUNITS												

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
AUTOADJ												
AUTOMPC	B	B	B	B	B	B	B	B	B	B	B	B
AUTOSEEL	E	E	E	E	E	E	E	E	E	E	E	E
AUTOSPC	E	E	E	E	E	E	E	E	E	E	E	E
AUTOSPCR				E								
AUTOSPRT		E			E	E	E	E	E	E		
BAILOUT	E	E	E	E	E	E	E	E	E	E	E	E
BETA												
BIGER	E	E	E								E	E
BIGER1	E	E	E								E	E
BIGER2	E	E	E								E	E
BOLTFACT	B	B	B		B	B	B	B	B	B		
BSHDAMP					B	B	B	B	B	B		
BUCKLE				E								
CA1, CA2	E	E	E	E	E	E	E	E	E	E	E	E
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E	E
CDIF												
CDPRT	B											
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E	E
CK41, CK42	E	E	E	E	E	E	E	E	E	E	E	E
CLOSE		B										B
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E	E

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
CNTASET	E											
COLPHEXA												
COMPMATT				B								
CONFAC	B	B	B	B	B	B	B	B	B	B	B	B
CORROPT		E										
COUPMASS	E	E	E	E	E	E	E	E	E	E	E	E
CP1, CP2	E		E	E		E	E		E	E	E	
CURV	E	E	E								E	E
CURVPLOT	E	E	E	E							E	E
DBALL	E	E	E	E	E	E	E	E	E	E	E	E
DBCCONV	E	E	E	E	E	E	E	E	E	E	E	E
DBCDIAG	E	E	E	E	E	E	E	E	E	E	E	E
DBCOVWRT	E	E	E	E	E	E	E	E	E	E	E	E
DBDICT	B	B	B	B	B	B	B	B	B	B	B	B
DBDN	E	E	E	E	E	E	E	E	E	E	E	E
DBDRPRJ	B	B	B	B	B	B	B	B	B	B	B	B
DBDRVER	B	B	B	B	B	B	B	B	B	B	B	B
DBRCV	E	E	E	E	E	E	E	E	E	E	E	E
DBUP	E	E	E	E	E	E	E	E	E	E	E	E
DDRMM									B	B		
DESPCH												
DESPCH1												
DFREQ						B			B			

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
DIGITS	E	E			E	E	E	E	E	E	E	E
DOF123		E										
DOPT	E	E	E								E	E
DPEPS												
DPREONLY												
DSNOKD			B									
DSZERO	B	B	B									
DYNSPCF		E			E	E	E	E	E	E		E
ELITASPC	B											
EPPRT	E	E	E	E	E	E	E	E	E	E	E	E
EPZERO	E	E	E	E	E	E	E	E	E	E	E	E
ERROR	E	E	E	E	E	E	E	E	E	E	E	E
EST	E	E	E	E	E	E	E	E	E	E	E	E
EXTBEMI	E	E										
EXTBEMO	E	E										
EXTDR	B	B			B	B	B	B	B	B		
EXTDROUT	B	B			B	B	B	B	B	B		
EXTDRUNT	B	B			B	B	B	B	B	B		
EXTOUT	B	B	B	B	B	B	B	B	B	B		
EXTRCV	E	E	E	E	E	E	E	E	E	E	E	E
EXTUNIT	B	B	B	B	B	B	B	B	B	B	B	B
F56	B	B	B	B	B	B	B	B	B	B	B	B
FACTOR	B	B	B	B	B	B	B	B	B	B	B	B

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
FCTCi					B	B	B	B	B	B		
FCTMi					B	B	B	B	B	B		
FCTZi	B				B	B	B	B	B	B		
FIXEDB	E	E										
FKSYMFAC				E								
FLEXINCR												
FLUIDMP									E			
FLUIDSE		B			B	B	B	B	B	B		
FOLLOWK		E	E	E	E	E	E	E	E	E		E
FRQDEPO						B			B			
FZERO		E				E	E	E	E	E		
G		B(1)			B	B	B	B	B	B		
GEOMU	E	E	E	E	E	E	E	E	E	E	E	E
GFL					B	B	B	B	B	B		
GPECT	E	E	E	E	E	E	E	E	E	E	E	E
GRDPNT	E	E	E	E	E	E	E	E	E	E	E	E
GUSTAERO												
HEATSTAT	B											
HFREQ								B	B	B		
HFREQFL								B	B	B		
IFP	E	E	E	E	E	E	E	E	E	E	E	E
IFTM									B			
INREL = -1	E		E									

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
INREL = -2	E											
INRLM		E			E	E	E	E	E	E		E
IRES	E		E	E		E	E		E	E	E	
ITAPE	B	B	B									
IUNIT	B	B	B									
K6ROT	E	E	E	E	E	E	E	E	E	E	E	E
KDAMP								B	B	B		
KDAMPFL								B	B	B		
KDIAG				E								
KGGCPCH	B											
KGGLPCH	B	B	B	B	B	B	B	B	B	B	B	B
LAMLIN		B										
LANGLE				B								
LFREQ								B	B	B		
LFREQFL								B	B	B		
LGDISP				E								
LGSTRN												
LMDYN		E			E	E	E	E	E	E		
LMFACT	E	E	E	E	E	E	E	E	E	E		
LMODES		E						B	B	B		
LMODESFL								B	B	B		
LMSTAT	E		E	E								
LOADU	E	E	E	E	E	E	E	E	E	E	E	E

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
LOOPID				E								
MACH												
MATNL												
MAXLP												
MAXRATIO	E	E	E	E	E	E	E	E	E	E	E	E
MECHFIL		E						E	E	E		
MECHFIX		E						E	E	E		
MECHPRT		E										
MESH	B											
METHCMRS		E			E	E	E	E	E	E		
MMFIL						E	E					
MODACC									B	B		
MODEL	B	B	B	B	B	B	B	B	B	B	B	B
MODTRK					B	B	B	B	B	B		
MPCX	B	B	B	B	B	B	B	B	B	B	B	B
MPCZERO	E	E	E	E	E	E	E	E	E	E	E	E
NASPRT												
NDAMP												
NINTPTS	E	E	E								E	E
NLAYERS				E								
NLHTLS				E								
NLMAX						E	E					
NLMIN						E	E					

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
NLTOL				E								
NMLOOP				E	E	E	E	E	E	E		
NOCOMPS	E	E	E	E							E	E
NOELOF	E	E	E								E	E
NOELOP	E	E	E								E	E
NOFISR	E	E	E	E							E	E
NOGPF	E	E	E								E	E
NOMSGSTR	E	E	E								E	E
NONCUP		B							B	B		
NQSET		E			E	E	E	E	E	E		E
NUMOUT	E	E	E								E	E
NUMOUT1	E	E	E								E	E
NUMOUT2	E	E	E								E	E
OCMP	E	E	E	E							E	E
OEE	E	E	E	E							E	E
OEF	E	E	E	E	E	E	E	E	E	E	E	E
OEFX	E	E	E	E	E	E	E	E	E	E	E	E
OEPT	E	E	E	E	E	E	E	E	E	E	E	E
OES	E	E	E	E	E	E	E	E	E	E	E	E
OESE	E	E	E	E	E	E	E	E	E	E	E	E
OESX	E	E	E	E	E	E	E	E	E	E	E	E
OG	E	E	E	E							E	E
OGEOM	E	E	E	E	E	E	E	E	E	E	E	E

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
OGPF	E	E	E	E							E	E
OGPS	E	E	E				E			E	E	E
OLDSEQ	B	B	B	B	B	B	B	B	B	B	B	B
OMAXR	E	E	E	E	E	E	E	E	E	E	E	E
OMID	B	B	B		B	B	B	B	B	B	B	B
OMPT	E	E	E	E	E	E	E	E	E	E	E	E
OPG	E		E	E		E	E		E	E	E	
OPGEOM												
OPGTKG												
OPPHIB												
OPPHIPA												
OPTEXIT												
OPTION		B										B
OQG	E	E	E	E	E	E	E	E	E	E	E	E
OSWELM	B	B	B	B	B	B	B	B	B	B	B	B
OSWPPT	B	B	B	B	B	B	B	B	B	B	B	B
OUG	E	E	E	E	E	E	E	E	E	E	E	E
OUGCORD	E	E	E	E	E	E	E	E	E	E	E	E
OUMU		E										E
OUNIT1	E	E	E	E	E	E	E	E	E	E	E	E
OUNIT2	E	E	E	E	E	E	E	E	E	E	E	E
OUTOPT	E	E	E	E							E	E
PANELMP									E			

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
PATVER	E	E	E	E	E	E	E	E	E	E	E	E
PBRPROP	B	B	B	B	B	B	B	B	B	B	B	B
PDRMSG	E	E	E	E	E	E	E	E	E	E	E	E
PHIBGN					B	B	B	B	B	B		
PHIDEL					B	B	B	B	B	B		
PHINUM					B	B	B	B	B	B		
PKRSP										B		
PLTMSG	E	E	E	E	E	E	E	E	E	E	E	E
POST	E	E	E	E	E	E	E	E	E	E	E	E
POSTEXT	E	E	E	E	E	E	E	E	E	E	E	E
POSTOPT												
POSTU	E	E	E	E	E	E	E	E	E	E	E	E
PREFDB							E	E	E	E		
PRGPST	E	E	E	E	E	E	E	E	E	E	E	E
PRINT												
PROUT	E	E	E	E	E	E	E	E	E	E	E	E
PRPA	E		E									
PRPJ	E		E			E	E		E	E	E	
PRTCSTM	E	E	E	E	E	E	E	E	E	E	E	E
PRTEQXIN	E	E	E	E	E	E	E	E	E	E	E	E
PRTGPDT	E	E	E	E	E	E	E	E	E	E	E	E
PRTGPL	E	E	E	E	E	E	E	E	E	E	E	E

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
PRTGPTT	E	E	E	E	E	E	E	E	E	E	E	E
PRTMAXIM	E	E	E		E	E	E	E	E	E	E	E
PRTMGG	E	E	E	E	E	E	E	E	E	E	E	E
PRTPG	E	E	E	E	E	E	E	E	E	E	E	E
PRTRESLT	E	E	E		E	E	E	E	E	E	E	E
Q												
QSETREM	B	B			B	B	B	B	B	B	B	B
RANCPLX						B			B			
RDCNT	E	E							E	E		
RDRESVEC		B							B	B		
RDSPARSE		B							B			
RESLTOPT	E	E	E	E	E	E	E	E	E	E	E	E
RESVALT										B		
RESVEC		E		E				E	E	E		
RESVINER		E		E				E	E	E		
RESVPGF		E		E				E	E	E		
RESVRAT		E		E				E	E	E		
RESVSE		E		E				E	E	E		
RESVSLI		E		E				E	E	E		
RESVSO		E		E				E	E	E		
RGBEAMA												
RGBEAME												
RGLCRIT												

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
RGSPRGK												
RLOOPNEW					B	B		B	B			
RMSINT						B			B			
RMSSF						B			B			
RMXPANEL										E		
RMXTRAN										B		
ROTCMRF					E							
ROTCOUP					B	B	B	B	B	B		
ROTCSV					B			B				
ROTGPF					B			B				
ROTSYNC					B			B				
RPOSTS1						B			B			
RSATT		B										
RSCON		B										
RSOPT		B										
RSPECTRA							E			E		
RSPRINT							E			E		
S1AG	E	E	E	E							E	E
S1AM	E	E	E	E							E	E
S1G	E	E	E	E							E	E
S1M	E	E	E	E							E	E
SCRSPEC		B										B
SDAMPUP									B			

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
SECOMB	B	B	B	B	B	B	B	B	B	B		
SEMAP	B	B	B	B	B	B	B	B	B	B	B	B
SEMAPOPT	B	B	B	B	B	B	B	B	B	B	B	B
SEMAPPRT	B	B	B	B	B	B	B	B	B	B	B	B
SENSUOO												
SEOP2CV	B	B			B	B	B	B	B	B		
SEP1XOVR	B	B	B	B	B	B	B	B	B	B	B	B
SEQOUT	B	B	B	B	B	B	B	B	B	B	B	B
SERST	B	B	B	B	B	B	B	B	B	B	B	B
SESDAMP		E			E	E	E	E	E	E		E
SESEF		E										E
SFEF70		E			E	E	E	E	E	E		
SHIFT1		E						E	E	E		
SHLDAMP					B	B	B	B	B	B		
SIGMA												
SKPAMP												
SLOOPID												
SMALLQ	B	B	B	B	B	B	B	B	B	B	B	B
SNORM	B	B	B	B	B	B	B	B	B	B	B	B
SNORMPRT	B	B	B	B	B	B	B	B	B	B	B	B
SOFTEXIT												
SOLID	B	B	B	B	B	B	B	B	B	B	B	B
SOLVSUB		E										

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
SPCGEN	E	E	E	E	E	E	E	E	E	E	E	E
SRCOMPS	B	B	B	B	B	B	B	B	B	B	B	B
SRTELTYP	E	E	E								E	E
SRTOPT	E	E	E								E	E
START	B	B	B	B	B	B	B	B	B	B	B	B
STIME												
STRUCTMP									E			
SUBID				E								
SUPER	B	B	B	B	B	B	B	B	B	B	B	B
SWPANGLE						E			E			
TABID							E			E		
TABS												
TESTNEG				E								
TINY	E	E	E								E	E
TOLRSC	B	B	B	B	B	B	B	B	B	B	B	B
TORSIN	B	B	B	B	B	B	B	B	B	B	B	B
TSTATIC												
UNITSYS	B	B	B	B	B	B	B	B	B	B	B	B
UNSYMF				B								
UPDTBSH												
USETPRT	E	E	E	E	E	E	E	E	E	E	E	E
USETSEL	E	E	E	E	E	E	E	E	E	E	E	E
USETSTRi	E	E	E	E	E	E	E	E	E	E	E	E

Table 6-1. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (101 through 115)											
	101	103	105	106	107	108	109	110	111	112	114	115
VMOPT		E	E	E	E	E	E	E	E	E	E	E
VREF												
W3		E(1)					E		E(1)	E		
W3FL							E			E		
W4		E(1)					E		E(1)	E		
W4FL							E			E		
WMODAL		E(1)							E(1)	E		
WRTMAT		E										
WTMASS	E	E	E	E	E	E	E	E	E	E	E	E
XFLAG	E		E								E	

Footnotes:

(1) Only applicable when used with the ADAMSMNF or MBDEXPORT case control commands.
(2) Not applicable to hyperelastic analysis.

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
ACOUT										E		
ACSYM										E		
ADPCON			E				E	E				
ADSTAT												

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
AESDISC				B						B		
AESMAXIT				B						B		
AESMETH				B						B		
AESRNDM				B						B		
AESTOL				B						B		
AFNORM												
AGGPCH										E		
ALPHA1		B	B		B	B	B	B		B		B
ALPHA2		B	B		B	B	B	B		B		B
ALTRED												
AMPCZ	B	B	B				B	B	B			
ASCOUP										E		
ASING	E	E	E	E	E	E	E	E	E	E		
AUNITS				B						B		
AUTOADJ										B		
AUTOMPC	B	B	B				B	B	B			
AUTOSEEL	E	E	E	E	E	E	E	E	E	E		
AUTOSPC	E	E	E	E	E	E	E	E	E	E		
AUTOSPCR							E					
AUTOSPRT					E	E				E		
BAILOUT	E	E	E	E	E	E	E	E	E	E		
BETA			E					E				
BIGER	E			E			E		E	E		
BIGER1	E			E			E		E	E		
BIGER2	E			E			E		E	E		
BOLTFACT												

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
BSHDAMP		B		B		B				B		
BUCKLE												
CA1, CA2	E	E	E	E	E	E	E	E	E	E		
CB1, CB2	E	E	E	E	E	E	E	E	E	E		
CDIF										E		
CDPRT												
CHECKOUT	E	E	E	E	E	E	E	E	E	E		
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E		
CK41, CK42	E	E	E	E	E	E	E	E	E	E		
CLOSE									B			
CM1, CM2	E	E	E	E	E	E	E	E	E	E		
CNTASET												
COLPHEXA											B	
COMPMATT												
CONFAC	B	B	B	B	B	B	B	B	B	B		
CORROPT												
COUPMASS	E	E	E	E	E	E	E	E	E	E	E	
CP1, CP2	E	E	E	E		E	E	E	E	E		
CURV	E			E			E		E	E		
CURVPLOT	E			E					E	E		
DBALL		E	E	E	E	E	E	E	E	E		
DBCCONV		E	E	E	E	E	E	E	E	E		
DBCDIAG		E	E	E	E	E	E	E	E	E		
DBCOVWRT		E	E	E	E	E	E	E	E	E		
DBDICT		B	B	B	B	B	B	B	B	B		
DBDN		E	E	E	E	E	E	E	E	E		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
DBDRPRJ		B	B	B	B	B	B	B	B	B		
DBDRVER		B	B	B	B	B	B	B	B	B		
DBRCV		E	E	E	E	E	E	E	E	E		
DBUP		E	E	E	E	E	E	E	E	E		
DDRMM						B				E		
DESPCH										E		
DESPCH1										E		
DFREQ		B				B				B		
DIGITS		E	E	E	E	E		E	E			
DOF123												
DOPT				E			E		E	E		
DPEPS										B		
DPREONLY									B			
DSNOKD										B		
DSZERO									B			
DYNSPCF		E			E	E			E	E		
ELITASPC												
EPPRT		E	E	E	E	E	E	E	E	E		
EPZERO		E	E	E	E	E	E	E	E	E		
ERROR		E	E	E	E	E	E	E	E	E		
EST		E	E	E	E	E	E	E	E	E		
EXTBEMI												
EXTBEMO												
EXTDR									B			
EXTDROUT									B			
EXTDRUNT									B			

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
EXTOUT			B	B	B	B	B	B	B	B		
EXTRCV		E	E	E	E	E	E	E	E	E		
EXTUNIT		B	B	B	B	B	B	B	B	B		
F56	B	B	B	B	B	B	B	B	B	B	B	B
FACTOR		B	B	B	B	B	B	B	B	B		
FCTCi												
FCTMi												
FCTZi												
FIXEDB									E	E		
FKSYMFAC												
FLEXINCR				B								
FLUIDMP										E		
FLUIDSE									B	B		
FOLLOWK	E								E	E		
FRQDEPO						B				B		
FZERO					E	E						
G		B	E		B	B		B		B		
GEOMU		E	E	E	E	E	E	E	E	E		
GFL						B				B		
GPECT		E	E	E	E	E	E	E	E	E		
GRDPNT		E	E	E	E	E	E	E	E	E	E	
GUSTAERO					B	B				B		
HEATSTAT												
HFREQ					B	B				B		
HFREQFL						B				B		
IFP		E	E	E	E	E	E	E	E	E		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
IFTM						B						
INREL = -1										B		
INREL = -2												
INRLM	E	E	E	E	E	E		E		E		
IRES		E	E	E	E	E	E	E		E		
ITAPE									B	B		
IUNIT									B	B		
K6ROT		E	E	E	E	E	E	E	E	E		
KDAMP					B					B		
KDAMPFL						B				B		
KDIAG							E					
KGGCPCH												
KGGLPCH	B	B	B				B	B	B	B		
LAMLIN												
LANGLE			B				B	B				
LFREQ					B	B				B		
LFREQFL						B				B		
LGDISP			E				E	E			B	B
LGSTRN												B
LMDYN												
LMFACT												
LMODES					B	B			E	B		
LMODESFL						B				B		
LMSTAT												
LOADU		E	E				E	E	E	E		
LOOPID			E				E	E				

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
MACH					B	B				B		
MATNL											B	
MAXLP			E					E				
MAXRATIO		E	E	E	E	E	E	E	E	E	E	
MECHFIL												
MECHFIX												
MECHPRT												
MESH							B	B		B		
METHCMRS		E	E		E	E		E		E		
MMFIL												
MODACC						B				B		
MODEL		B	B	B	B	B	B	B	B	B		
MODTRK												
MPCX		B	B	B	B	B	B	B	B	B		
MPCZERO	E	E	E	E	E	E	E	E	E	E		
NASPRT										B		
NDAMP			E					E				
NINTPTS				E			E		E	E		
NLAYERS			E									
NLHTLS							E					
NLMAX												
NLMIN												
NLTOL								E				
NMLOOP												
NOCOMPS				E			E		E	E		
NOELOF				E			E		E	E		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
NOELOP				E			E		E	E		
NOFISR				E			E		E	E		
NOGPF				E			E		E	E		
NOMSGSTR				E			E		E	E		
NONCUP										B		
NQSET		E	E	E	E	E		E	E			
NUMOUT				E			E		E	E		
NUMOUT1				E			E		E	E		
NUMOUT2				E			E		E	E		
OCMP									E	E		
OEE	E								E	E		
OEF	E	E	E	E	E	E	E	E	E	E		
OEFX	E	E	E	E	E	E	E	E	E	E		
OEPT	E	E	E	E	E	E	E	E	E	E		
OES	E	E	E	E	E	E	E	E	E	E		
OESE	E	E	E	E	E	E	E	E	E	E		
OESX	E	E	E	E	E	E	E	E	E	E		
OG	E			E			E		E	E		
OGEOM	E	E	E	E	E	E	E	E	E	E	E	
OGPF	E			E					E	E		
OGPS	E			E					E	E		
OLDSEQ		B	B	B	B	B	B	B	B	B		
OMAXR		E	E	E	E	E	E	E	E	E	E	
OMID	B	B		B	B	B			B	B		
OMPT	E	E	E	E	E	E	E	E	E	E	E	
OPG	E	E	E		E	E		E		E	E	

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
OPGEOM	E			B	B	B				B		
OPGTKG	E			B	B	B				B		
OPPHIB					B	B				B		
OPPHIPA					B	B				B		
OPTEXIT										B		
OPTION												
OQG	E	E	E	E	E	E	E	E	E	E		
OSWELM	B	B	B	B	B	B			B	B		
OSWPPT	B	B	B	B	B	B			B	B		
OUG	E	E	E	E	E	E	E	E	E	E		
OUGCORD	E	E	E	E	E	E	E	E	E	E	E	
OUMU									E	E		
OUNIT1		E	E	E	E	E	E	E	E	E		
OUNIT2		E	E	E	E	E	E	E	E	E		
OUTOPT				E			E		E	E		
PANELMP										E		
PATVER		E	E	E	E	E	E	E	E	E		
PBRPROP	B	B	B	B	B	B	B	B	B	B		
PDRMSG		E	E	E	E	E	E	E	E	E		
PHIBGN												
PHIDEL												
PHINUM												
PKRSP												
PLTMSG		E	E	E	E	E	E	E	E	E		
POST		E	E	E	E	E	E	E	E	E	E	B
POSTEXT	E	E	E	E	E	E	E	E	E	E	E	

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
POSTOPT											E	
POSTU		E	E	E	E	E	E	E	E	E		
PREFDB										E		
PRGPST		E	E	E	E	E	E	E	E	E	E	
PRINT					B					B		
PROUT		E	E	E	E	E	E	E	E	E	E	
PRPA				E			E			E		
PRPJ		E		E		E	E	E		E		
PRTCSTM		E	E	E	E	E	E	E	E	E		
PRTEQXIN		E	E	E	E	E	E	E	E	E		
PRTGPDT		E	E	E	E	E	E	E	E	E		
PRTGPL		E	E	E	E	E	E	E	E	E		
PRTGPTT		E	E	E	E	E	E	E	E	E		
PRTMAXIM		E	E	E	E	E	E	E	E	E		
PRTMGG		E	E	E	E	E	E	E	E	E		
PRTPG		E	E	E	E	E	E	E	E	E		
PRTRESLT		E	E	E	E	E	E	E	E	E		
Q				B	B	B				B		
QSETREM		B	B	B	B	B		B	B			
RANCPLX												
RDCNT												
RDRESVEC										B		
RDSPARSE												
RESLTOPT	E	E	E	E	E	E	E	E	E	E		
RESVALT												
RESVEC						E			E	E		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
RESVINER						E			E	E		
RESVPGF						E			E	E		
RESVRAT						E			E	E		
RESVSE						E			E	E		
RESVSLI						E			E	E		
RESVSO						E			E	E		
RGBEAMA											B	
RGBEAME											B	
RGLCRIT											B	
RGSPRGK											B	
RLOOPNEW												
RMSINT												
RMSSF												
RMXPANEL												
RMXTRAN												
ROTCMRF												
ROTCOUP												
ROTCSV												
ROTGPF												
ROTSYNC												
RPOSTS1												
RSATT												
RSCON												
RSOPT												
RSPECTRA												
RSPRINT												

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
S1AG				E			E		E	E		
S1AM				E			E		E	E		
S1G				E			E		E	E		
S1M				E			E		E	E		
SCRSPEC												
SDAMPUP												
SECOMB			B	B		B	B	B		B		
SEMAP		B	B	B	B	B	B	B	B	B		
SEMAPOPT		B	B	B	B	B	B	B	B	B		
SEMAPPRT		B	B	B	B	B	B	B	B	B		
SENSUOO										B		
SEOP2CV			B	B	B	B		B	B			
SEP1XOVR		B	B	B	B	B	B	B	B	B		
SEQOUT		B	B	B	B	B	B	B	B	B		
SERST		B	B	B	B	B	B	B	B	B		
SESDAMP	E	E	E		E				E			
SESEF									E			
SFEF70										E		
SHIFT1												
SHLDAMP		B		B		B				B		
SIGMA								B				
SKPAMP					B	B			B	B		
SLOOPID			E									
SMALLQ		B	B	B	B	B	B	B	B	B		
SNORM		B	B	B	B	B	B	B	B	B		
SNORMPRT		B	B	B	B	B	B	B	B	B		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
SOFTEXIT										B		
SOLID		B	B	B	B	B	B	B	B	B		
SOLVSUB												
SPCGEN		E	E	E	E	E	E	E	E	E		
SRCOMPS	B	B	B	B	B	B	B	B	B	B		
SRTELTYP				E			E		E	E		
SRTOPT				E			E		E	E		
START		B		B	B	B	B	B	B	B		
STIME			E					E				
STRUCTMP										E		
SUBID							E					
SUPER		B	B	B	B	B	B	B	B	B		
SWPANGLE												
TABID												
TABS							E	E		E		
TESTNEG							E					
TINY				E			E		E	E	E	
TOLRSC		B	B	B	B	B	B	B	B	B		
TORSIN	B	B	B	B	B	B	B	B	B	B		
TSTATIC			E					E				
UNITSYS	B	B	B	B	B	B	B	B	B	B	B	B
UNSYMF												
UPDTBSH										E		
USETPRT		E	E	E	E	E	E	E	E	E		
USETSEL		E	E	E	E	E	E	E	E	E		
USETSTRi		E	E	E	E	E	E	E	E	E		

Table 6-2. PARAMeter Names in the SOLs

PARAMeter Name	Solution Numbers (116 through 701)											
	116	118	129	144	145	146	153	159	187	200	401	601 & 701
VMOPT		E	E	E	E	E	E	E	E	E		
VREF					B					B		
W3			B					E		B		
W3FL										B		
W4			B					E		B		
W4FL										B		
WMODAL												
WRTMAT												
WTMASS		E	E	E	E	E	E	E	E	E	E	
XFLAG				E						E		

Footnotes:

(1) Only applicable when used with the ADAMSMNF or MBDEXPORT case control commands.

Chapter 7: Item Codes

7.1 Item Codes Overview

Item codes are integer numbers assigned to specific output quantities such as the third translational component of displacement, the major principal stress at Z1 in a CQUAD4 element, or the torque in a CBAR element. Item codes are specified on the following input statements:

- DRESP1 entry for Design Sensitivity and Optimization (SOL 200).
- X-Y Plotting commands. See “Plotting” in the *NX Nastran User’s Guide*.
- DTI,INDTA entry for stress sorting.

The following tables provide item codes for:

- Element Stress or Strain.
- Element Force.
- Fluid Virtual Mass Pressure.
- Heat Transfer Flux.
- Slideline Contact Output.
- Element Strain Energy.

The following superscripts appear in the tables and indicate:

1. Data for components marked with the superscript (1) are included in the datablock MES output from module DRMS1. See the *NX Nastran DMAP Programmer’s Guide*.
2. Composite Element Stresses and Failure Indices. See “Overview of Laminated Composite Materials” in the *NX Nastran User’s Guide*.

Shell element item code remarks

1. The CENTER and CORNER options on the STRESS, STRAIN, and FORCE case control commands determine the CQUAD4, CQUADR, and CTRIAR item code tables. If CENTER is requested (default), the item code tables labeled as "Center Only" should be used. If CORNER is requested, the item code tables labeled as "Center and Corners" should be used.

For example, when CENTER is requested for a CQUAD4 element, the CQUAD(33) item code format should be used. When CORNER is requested for a CQUAD4 element, the CQUAD(144) item code format should be used.

The CENTER and CORNER options on the STRESS, STRAIN, and FORCE case control commands are ignored by the CTRIA3, CTRIA6, and CQUAD8 elements. The CTRIA3 element item code table is "Center Only". The CTRIA6 and CQUAD8 item code tables are "Center and Corner".

When including shell element item codes on a DRESP1 bulk entry for Design Sensitivity and Optimization (SOL 200), do not mix elements which will produce a different item code table on the same DRESP1 stress, strain or force response.

2. Linear format shell element item codes are output in the element coordinate system. One exception; when PARAM,CURV,1 is defined, stress for the CQUAD4 and CTRIA3 elements is output in the material coordinate system.

3. Composite item codes are output in the material coordinate system defined on the element, and not the material coordinate system of the individual plies.

4. Nonlinear format shell element item codes are output in the element coordinate system.

5. Hyperelastic shell element item codes are output in the CID defined on the PLPLANE.

Solid element item code remarks

1. Linear format solid element item codes are output in the material system.

2. Nonlinear format solid element item codes are output in the element coordinate system. One exception; when ELRESCS is set to “1” on the NXSTRAT bulk entry in solutions 601 or 701, nonlinear format solid element item codes are output in the material coordinate system.

3. Hyperelastic solid element item codes are output in the basic coordinate system.

7.2 Element Stress (or Strain) Item Codes

All item codes refer to stresses (or strains) unless otherwise denoted.

If output is magnitude/phase, the magnitude replaces the real part, and the phase replaces the imaginary part. Strain item codes are equivalent to stress item codes. However, strain is computed for only some elements. See “Element Summary – Small Strain Elements” in the *NX Nastran Element Library*.

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBAR (34)	2[1]	End A-Point C	2[1]	End A-Point C	RM
	3[1]	End A-Point D	3[1]	End A-Point D	RM
	4[1]	End A-Point E	4[1]	End A-Point E	RM
	5[1]	End A-Point F	5[1]	End A-Point F	RM
	6[1]	Axial	6[1]	Axial	RM
	7	End A maximum	7[1]	End A-Point C	IP
	8	End A minimum	8[1]	End A-Point D	IP
	9	Safety margin in tension	9[1]	End A-Point E	IP
	10[1]	End B-Point C	10[1]	End A-Point F	IP
	11[1]	End B-Point D	11[1]	Axial	IP
	12[1]	End B-Point E	12[1]	End B-Point C	RM
	13[1]	End B-Point F	13[1]	End B-Point D	RM
	14	End B maximum	14[1]	End B-Point E	RM
	15	End B minimum	15[1]	End B-Point F	RM
	16	Safety margin in compression	16[1]	End B-Point C	IP
			17[1]	End B-Point D	IP
			18[1]	End B-Point E	IP
			19[1]	End B-Point F	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBAR (100) Intermediate Stations	2	Station Distance/Length	2	Station Distance/Length	RM
	3	Point C	3	Point C	RM
	4	Point D	4	Point D	RM
	5	Point E	5	Point E	RM
	6	Point F	6	Point F	RM
	7	Axial	7	Axial	RM
	8	Maximum	8	Maximum	RM
	9	Minimum	9	Minimum	RM
	10	Margin of Safety (Item codes above are given for End A. For codes 2 through 10 at intermediate stations add (K-1)*9 where K is the station number, and for codes at End B, K=number of stations plus 1.)	10	Point C	IP
			11	Point D	IP
			12	Point E	IP
			13	Point F	IP
			14	Axial	IP
			15	Maximum	IP
			16	Minimum (Item codes above are given for End A. For codes 2 through 16 at intermediate stations add (K-1)*15 where K is the station number, and for codes at End B, K=number of stations plus 1.)	IP

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBEAM (2) Linear Format	2	External grid point ID	2	External grid point ID	
	3	Station distance/length	3	Station distance/length	
	4[1]	Long. Stress at Point C	4[1]	Long. Stress at Point C	RM
	5[1]	Long. Stress at Point D	5[1]	Long. Stress at Point D	RM
	6[1]	Long. Stress at Point E	6[1]	Long. Stress at Point E	RM
	7[1]	Long. Stress at Point F	7[1]	Long. Stress at Point F	RM
	8	Maximum stress	8[1]	Long. Stress at Point C	IP
	9	Minimum stress	9[1]	Long. Stress at Point D	IP
	10	Safety margin in tension	10[1]	Long. Stress at Point E	IP
	11	Safety margin in compression Item codes are given for End A. Addition of the quantity (K-1)*10 to the item code points to the same information for other stations, where K is the station number. K=11 for End B, and K=2 through 10 for intermediate stations.	11[1]	Long. Stress at Point F Item codes are given for End A. Addition of the quantity (K-1)*10 to the item code points to the same information for other stations, where K is the station number. K=11 for End B, and K=2 through 10 for intermediate stations.	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBEAM (94) Nonlinear Format	2	External grid point ID	Not applicable		
	3	C (Character)			
	4	Long. Stress at point C			
	5	Equivalent stress			
	6	Total strain			
	7	Effective plastic strain			
	8	Effective creep strain (Item codes 3 through 8 are repeated for points D, E, and F. Then the entire record (from 2 through N) is repeated for End B of the element.)			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3	Circumferential angle	3	Circumferential angle	
	4[1]	Long. Stress at Point C	4[1]	Long. Stress at Point C	RM
	5[1]	Long. Stress at Point D	5[1]	Long. Stress at Point D	RM
	6[1]	Long. Stress at Point E	6[1]	Long. Stress at Point E	RM
	7[1]	Long. Stress at Point F	7[1]	Long. Stress at Point F	RM
	8	Maximum stress	8[1]	Long. Stress at Point C	IP
	9	Minimum stress	9[1]	Long. Stress at Point D	IP
	10	Safety margin in tension	10[1]	Long. Stress at Point E	IP
	11	Safety margin in compression (Item codes are given for End A. Item codes 12 through 21 point to the same information for end B.)	11[1]	Long. Stress at Point F (Item codes are given for End A. Item codes 12 through 21 point to the same information for End B.)	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBUSH (102) Linear Format	2	Translation-x	2	Translation-x	R
	3	Translation-y	3	Translation-y	R
	4	Translation-z	4	Translation-z	R
	5	Rotation-x	5	Rotation-x	R
	6	Rotation-y	6	Rotation-y	R
	7	Rotation-z	7	Rotation-z	R
			8	Translation-x	I
			9	Translation-y	I
			10	Translation-z	I
			11	Rotation-x	I
			12	Rotation-y	I
			13	Rotation-z	I

Element Name (Code)	Real Stresses or Strains	
	Item Code	Item
CBUSH (226) Nonlinear Format	5	Stress Translational X
	6	Stress Translational Y
	7	Stress Translational Z
	8	Strain Translational X
	9	Strain Translational Y
	10	Strain Translational Z
	14	Stress Rotational X
	15	Stress Rotational Y
	16	Stress Rotational Z
	17	Strain Rotational X
	18	Strain Rotational Y
	19	Strain Rotational Z

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CBUSH1D (40)	1	Element ID	Not applicable		
	2	Axial force			
	3	Axial displacement			
	4	Axial velocity			
	5	Axial stress			
	6	Axial strain			
	7				
	8				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CDUM3 thru CDUM9 (55-61)	2[1]	S1	2[1]	S1	RM
	3[1]	S2	3[1]	S2	RM
	4[1]	S3	4[1]	S3	RM
	5[1]	S4	5[1]	S4	RM
	6[1]	S5	6[1]	S5	RM
	7[1]	S6	7[1]	S6	RM
	8[1]	S7	8[1]	S7	RM
	9[1]	S8	9[1]	S8	RM
	10[1]	S9	10[1]	S9	RM
			11[1]	S1	IP
			12[1]	S2	IP
			13[1]	S3	IP
			14[1]	S4	IP
			15[1]	S5	IP
			16[1]	S6	IP
			17[1]	S7	IP
			18[1]	S8	IP
			19[1]	S9	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CELAS1 (11)	2[1]	Stress	2[1]	Stress	RM
			3[1]	Stress	IP
CELAS2 (12)	2[1]	Stress	2[1]	Stress	RM
			3[1]	Stress	IP
CELAS3 (13)	2[1]	Stress	2[1]	Stress	RM
			3[1]	Stress	IP

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CGAP (86)	2	Normal x	Not applicable		
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CHEXA (67) Linear Format	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	9[1]	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	IP
	13	von Mises or octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-121	Repeat items 5 through 17 for 8 corners	
	19	Third principal y cosine			
	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	25	Third principal z cosine			
	26-193	repeat items 5 through 25 for 8 corners			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CHEXA (93) Nonlinear	2	Grid/Gauss	Not applicable		
	3	External grid ID (0=center)			
	4	Stress-X			
	5	Stress-Y			
	6	Stress-Z			
	7	Stress-XY			
	8	Stress-YZ			
	9	Stress-ZX			
	10	Equivalent stress			
	11	Effective plastic strain			
	12	Effective creep strain			
	13	Strain-X			
	14	Strain-Y			
	15	Strain-Z			
	16	Strain-XY			
	17	Strain-YZ			
	18	Strain-ZX			
	19-146	Repeat items 3 through 18 for 8 corners			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CHEXA (269) Composite Center Only	2	Lamina number	2	Lamina number	
	3	Fiber location	3	Fiber location	
	4	0	4	0	
	5	Normal-1	5	Normal-1	RM
	6	Normal-2	6	Normal-2	RM
	7	Normal-3	7	Normal-3	RM
	8	Shear-12	8	Shear-12	RM
	9	Shear-23	9	Shear-23	RM
	10	Shear-13	10	Shear-13	RM
	11	von Mises	11	Normal-1	IP
			12	Normal-2	IP
			13	Normal-3	IP
			14	Shear-12	IP
			15	Shear-23	IP
			16	Shear-13	IP

If CPLYBT is specified on the stress or strain request, repeat words 3-11 once for real stresses and strains, or repeat words 3-16 once for complex stresses and strains.

If CPLYBMT is specified on the stress or strain request, repeat words 3-11 twice for real stresses and strains, or repeat words 3-16 twice for complex stresses and strains.

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CHEXA (269) Composite Center and Corners	2	Lamina number	2	Lamina number	
	3	Fiber location	3	Fiber location	
	4	0	4	0	
	5	Normal-1	5	Normal-1	RM
	6	Normal-2	6	Normal-2	RM
	7	Normal-3	7	Normal-3	RM
	8	Shear-12	8	Shear-12	RM
	9	Shear-23	9	Shear-23	RM
	10	Shear-13	10	Shear-13	RM
	11	von Mises	11	Normal-1	IP
	12	Grid 1	12	Normal-2	IP
	13-19	Same as items 5-11	13	Normal-3	IP
	20	Grid 2	14	Shear-12	IP
	21-27	Same as items 5-11	15	Shear-23	IP
	28	Grid 3	16	Shear-13	IP
	29-35	Same as items 5-11	17	Grid 1	
	36	Grid 4	18-29	Same as items 5-16	
	37-43	Same as items 5-11	30	Grid 2	
			31-42	Same as items 5-16	
			43	Grid 3	
			44-55	Same as items 5-16	
			56	Grid 4	
			57-68	Same as items 5-16	

If CPLYBT is specified on the stress or strain request, repeat words 3-43 once for real stresses and strains, or repeat words 3-68 once for complex stresses and strains.

If CPLYBMT is specified on the stress or strain request, repeat words 3-43 twice for real stresses and strains, or repeat words 3-68 twice for complex stresses and strains.

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CHEXAFD (202) Hyperelastic with 8 grid points	2	Grid/Gauss	Not applicable		
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Cauchy stress-YZ			
	9	Cauchy stress-ZX			
	10	Pressure $p=\frac{1}{3}(\sigma_x+\sigma_y+\sigma_z)$			
	11	Volumetric strain J-1			
	12	Logarithmic strain-X			
	13	Logarithmic strain-Y			
	14	Logarithmic strain-Z			
	15	Logarithmic strain-XY			
	16	Logarithmic strain-YZ			
	17	Logarithmic strain-ZX			
	18-122	Repeat items 3 through 17 for 7 Gauss/grid points			
CHEXAFD (207) Hyperelastic with 20 grid points	2-17	Same as CHEXAFD (202)	Not applicable		
	18-407	Repeat items 3 through 17 for 26 Gauss points			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CONROD (10) Linear Format	2[1]	Axial stress	2[1]	Axial stress	RM
	3	Axial safety margin	3[1]	Axial stress	IP
	4[1]	Torsional stress	4[1]	Torsional stress	RM
	5	Torsional safety margin	5[1]	Torsional stress	IP
CONROD (92) Nonlinear Format	2	Axial stress	Not applicable		
	3	Equivalent stress			
	4	Total strain			
	5	Effective plastic strain			
	6	Effective creep strain			
	7	Linear torsional stress			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPENTA (68) Linear Format	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	9[1]	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	IP
	13	von Mises or Octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-95	Repeat items 5 through 17 for 6 corners	
	19	Third principal y cosine			
	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	25	Third principal z cosine			
	26-151	Repeat items 5 through 25 for 6 corners			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPENTA (91) Nonlinear Format	2	Grid/Gauss	Not applicable		
	3	External grid ID (0=center)			
	4	Normal x stress			
	5	Normal y stress			
	6	Normal z stress			
	7	Shear xy stress			
	8	Shear yz stress			
	9	Shear zx stress			
	10	Equivalent stress			
	11	Effective plastic strain			
	12	Effective creep strain			
	13	Normal x strain			
	14	Normal y strain			
	15	Normal z strain			
	16	Shear xy strain			
	17	Shear yz strain			
	18	Shear zx strain			
	19-114	Repeat items 3 through 18 for 6 corners			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPENTA (270) Composite Center Only	2	Lamina number	2	Lamina number	
	3	Fiber location	3	Fiber location	
	4	0	4	0	
	5	Normal-1	5	Normal-1	RM
	6	Normal-2	6	Normal-2	RM
	7	Normal-3	7	Normal-3	RM
	8	Shear-12	8	Shear-12	RM
	9	Shear-23	9	Shear-23	RM
	10	Shear-13	10	Shear-13	RM
	11	von Mises	11	Normal-1	IP
			12	Normal-2	IP
			13	Normal-3	IP
			14	Shear-12	IP
			15	Shear-23	IP
			16	Shear-13	IP
If CPLYBT is specified on the stress or strain request, repeat words 3-11 once for real stresses and strains, or repeat words 3-16 once for complex stresses and strains. If CPLYBMT is specified on the stress or strain request, repeat words 3-11 twice for real stresses and strains, or repeat words 3-16 twice for complex stresses and strains.					

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPENTA (270) Composite Center and Corners	2	Lamina number	2	Lamina number	
	3	Fiber location	3	Fiber location	
	4	0	4	0	
	5	Normal-1	5	Normal-1	RM
	6	Normal-2	6	Normal-2	RM
	7	Normal-3	7	Normal-3	RM
	8	Shear-12	8	Shear-12	RM
	9	Shear-23	9	Shear-23	RM
	10	Shear-13	10	Shear-13	RM
	11	von Mises	11	Normal-1	IP
	12	Grid 1	12	Normal-2	IP
	13-19	Same as items 5-11	13	Normal-3	IP
	20	Grid 2	14	Shear-12	IP
	21-27	Same as items 5-11	15	Shear-23	IP
	28	Grid 3	16	Shear-13	IP
	29-35	Same as items 5-11	17	Grid 1	
			18-29	Same as items 5-16	
			30	Grid 2	
			31-42	Same as items 5-16	
			43	Grid 3	
			44-55	Same as items 5-16	

If CPLYBT is specified on the stress or strain request, repeat words 3-35 once for real stresses and strains, or repeat words 3-55 once for complex stresses and strains.

If CPLYBMT is specified on the stress or strain request, repeat words 3-35 twice for real stresses and strains, or repeat words 3-55 twice for complex stresses and strains.

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPENTAFD (204) Hyperelastic with 6 grid points	2-17	Same as CHEXAFD (201)	Not applicable		
	18-92	Repeat items 3 through 17 for 5 Gauss points			
CPENTAFD (209) Hyperelastic with 15 grid points	2-17	Same as CHEXAFD (201)	Not applicable		
	18-317	Repeat items 3 through 17 for 20 Gauss points			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN3 (271) Triangle plane strain Linear Format (Center Only)	2	Normal stress in x	2	Normal stress in x	RM
	3	Normal stress in y	3	Normal stress in x	IP
	4	Normal stress in z	4	Normal stress in y	RM
	5	In-plane shear stress	5	Normal stress in y	IP
	6	Von Mises stress	6	Normal stress in z	RM
			7	Normal stress in z	IP
			8	In-plane shear stress	RM
			9	In-plane shear stress	IP

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN4 (272) Quadrilateral plane strain Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
	27	Grid ID, 4 for corner 4	22-29	Same as 4 through 11 for corner 2	
	28-32	Same as 4 through 8 for corner 4	30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	
			39	Grid ID, 4 for corner 4	
			40-47	Same as 4 through 11 for corner 4	

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN6 (273) Triangle plane strain Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
			22-29	Same as 4 through 11 for corner 2	
			30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN8 (274) Quadrilateral plane strain Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
	27	Grid ID, 4 for corner 4	22-29	Same as 4 through 11 for corner 2	
	28-32	Same as 4 through 8 for corner 4	30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	
			39	Grid ID, 4 for corner 4	
			40-47	Same as 4 through 11 for corner 4	

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS3 (275) Triangle plane stress Linear Format (Center Only)	2	Normal stress in x	2	Normal stress in x	RM
	3	Normal stress in y	3	Normal stress in x	IP
	4	Normal stress in z	4	Normal stress in y	RM
	5	In-plane shear stress	5	Normal stress in y	IP
	6	Von Mises stress	6	Normal stress in z	RM
			7	Normal stress in z	IP
			8	In-plane shear stress	RM
			9	In-plane shear stress	IP

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS4 (276) Quadrilateral plane stress Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
	27	Grid ID, 4 for corner 4	22-29	Same as 4 through 11 for corner 2	
	28-32	Same as 4 through 8 for corner 4	30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	
			39	Grid ID, 4 for corner 4	
			40-47	Same as 4 through 11 for corner 4	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS6 (277) Triangle plane stress Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
			22-29	Same as 4 through 11 for corner 2	
			30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS8 (278) Quadrilateral plane stress Linear Format (Center and Corners)	3	Grid ID, 0 for centroid	3	Grid ID, 0 for centroid	
	4	Normal stress in x	4	Normal stress in x	RM
	5	Normal stress in y	5	Normal stress in x	IP
	6	Normal stress in z	6	Normal stress in y	RM
	7	In-plane shear stress	7	Normal stress in y	IP
	8	Von Mises stress	8	Normal stress in z	RM
	9	Grid ID, 1 for corner 1	9	Normal stress in z	IP
	10-14	Same as 4 through 8 for corner 1	10	In-plane shear stress	RM
	15	Grid ID, 2 for corner 2	11	In-plane shear stress	IP
	16-20	Same as 4 through 8 for corner 2	12	Grid ID, 1 for corner 1	
	21	Grid ID, 3 for corner 3	13-20	Same as 4 through 11 for corner 1	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	22-26	Same as 4 through 8 for corner 3	21	Grid ID, 2 for corner 2	
	27	Grid ID, 4 for corner 4	22-29	Same as 4 through 11 for corner 2	
	28-32	Same as 4 through 8 for corner 4	30	Grid ID, 3 for corner 3	
			31-38	Same as 4 through 11 for corner 3	
			39	Grid ID, 4 for corner 4	
			40-47	Same as 4 through 11 for corner 4	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN3 (281) Triangle plane strain Nonlinear format (Center Only)	2	Normal stress in x	Not applicable		
	3	Normal stress in y			
	4	Normal stress in z			
	5	In-plane shear stress			
	6	Equivalent stress			
	7	Effective plastic/inelastic strain			
	8	Effective creep strain			
	9	Strain in x			
	10	Strain in y			
	11	Strain in z			
	12	In-plane shear strain			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN6 (283) Triangle plane strain Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 3 times for corner locations.				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN8 (284) Quadrilateral plane strain Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 4 times for corner locations.				

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS3 (285) Triangle plane stress Nonlinear format (Center Only)	2	Normal stress in x	Not applicable		
	3	Normal stress in y			
	4	Normal stress in z			
	5	In-plane shear stress			
	6	Equivalent stress			
	7	Effective plastic/inelastic strain			
	8	Effective creep strain			
	9	Strain in x			
	10	Strain in y			
	11	Strain in z			
	12	In-plane shear strain			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS6 (287) Triangle plane stress Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 3 times for corner locations.				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS8 (288) Quadrilateral plane stress Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 4 times for corner locations.				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN4 (289) Quadrilateral plane strain Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 4 times for corner locations.				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS4 (290) Quadrilateral plane stress Nonlinear format (Center and Corners)	3	Grid ID, 0 for centroid	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Equivalent stress			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	9	Effective plastic/inelastic strain			
	10	Effective creep strain			
	11	Strain in x			
	12	Strain in y			
	13	Strain in z			
	14	In-plane shear strain			
	Words 3 through 14 repeat 4 times for corner locations.				

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN3 (291) Triangle plane strain Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-35	Words 3 through 13 repeat 2 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN4 (292) Quadrilateral plane strain Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	13	In-plane shear strain			
	14-46	Words 3 through 13 repeat 3 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN6 (293) Triangle plane strain Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-35	Words 3 through 13 repeat 2 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTN8 (294) Quadrilateral plane strain Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-46	Words 3 through 13 repeat 3 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS3 (295) Triangle plane stress Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-35	Words 3 through 13 repeat 2 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS4 (296) Quadrilateral plane stress Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-46	Words 3 through 13 repeat 3 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS6 (297) Triangle plane stress Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-35	Words 3 through 13 repeat 2 times.			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPLSTS8 (298) Quadrilateral plane stress Hyperelastic (Grid)	3	Grid ID	Not applicable		
	4	Normal stress in x			
	5	Normal stress in y			
	6	Normal stress in z			
	7	In-plane shear stress			
	8	Pressure			
	9	Volume strain			
	10	Strain in x			
	11	Strain in y			
	12	Strain in z			
	13	In-plane shear strain			
	14-46	Words 3 through 13 repeat 3 times.			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPYRAM (255) Linear Format	2	Stress/Strain coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	
	6	Normal x	6	Normal x	RM
	7	Shear xy	7	Normal y	RM
	8	First principal	8	Normal z	RM
	9	First principal x cosine	9	Shear xy	RM
	10	Second principal x cosine	10	Shear yz	RM
	11	Third principal x cosine	11	Shear zx	RM
	12	Mean pressure	12	Normal x	IP
	13	von Mises or octahedral shear stress	13	Normal y	IP
	14	Normal y	14	Normal z	IP
	15	Shear yz	15	Shear xy	IP
	16	Second principal	16	Shear yz	IP
	17	First principal y cosine	17	Shear zx	IP
	18	Second principal y cosine	18-82	Repeat items 5 through 17 for five corners	
	19	Third principal y cosine			
	20	Normal z			
	21	Shear zx			
	22	Third principal			
	23	First principal z cosine			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-130	Repeat items 5 through 25 for five corners			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPYRAM (256) Nonlinear Format	3	0 (Result at Center)	Not applicable		
	4	Stress in X			
	5	Stress in Y			
	6	Stress in Z			
	7	Stress in XY			
	8	Stress in YZ			
	9	Stress in ZX			
	10	Equivalent stress			
	11	Equivalent plastic strain			
	12	Effective creep strain			
	13	Strain in X			
	14	Strain in Y			
	15	Strain in Z			
	16	Strain in XY			
	17	Strain in YZ			
	18	Strain in ZX			
	19	Grid 1 ID			
	20–34	Same as items 4–18 for corner 1			
	35	Grid 2 ID			
	36–50	Same as items 4–18 for corner 2			
	51	Grid 3 ID			
	52–66	Same as items 4–18 for corner 3			
	67	Grid 4 ID			
	68–82	Same as items 4–18 for corner 4			
	83	Grid 5 ID			
	84–98	Same as items 4–18 for corner 5			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CPYRAMFD (257) Pyramid with 5-grids (258) Pyramid with 13-grids Hyperelastic	2	Grid	Not applicable		
	3	Grid ID			
	4	Stress in X			
	5	Stress in Y			
	6	Stress in Z			
	7	Stress in XY			
	8	Stress in YZ			
	9	Stress in ZX			
	10	Pressure $p=\frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$			
	11	Volumetric strain			
	12	Strain in X			
	13	Strain in Y			
	14	Strain in Z			
	15	Strain in XY			
	16	Strain in YZ			
	17	Strain in ZX			
	18-77	Repeat items 3 through 17 for 5 grid points			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD4 (33) Linear Format Center Only	2	Z1=Fiber Distance 1	2	Z1=Fiber Distance 1	
	3[1]	Normal x at Z1	3[1]	Normal x at Z1	RM
	4[1]	Normal y at Z1	4[1]	Normal x at Z1	IP
	5[1]	Shear xy at Z1	5[1]	Normal y at Z1	RM
	6	Shear angle at Z1	6[1]	Normal y at Z1	IP
	7	Major principal at Z1	7[1]	Shear xy at Z1	RM
	8	Minor principal at Z1	8[1]	Shear xy at Z1	IP
	9	von Mises or maximum shear at Z1	9	Z2=Fiber distance 2	
	10	Z2=Fiber distance 2	10[1]	Normal x at Z2	RM
	11[1]	Normal x at Z2	11[1]	Normal x at Z2	IP
	12[1]	Normal y at Z2	12[1]	Normal y at Z2	RM
	13[1]	Shear xy at Z2	13[1]	Normal y at Z2	IP
	14	Shear angle at Z2	14	Shear xy at Z2	RM
	15	Major principal at Z2	15	Shear xy at Z2	IP
	16	Minor principal at Z2			
	17	von Mises or maximum shear at Z2			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD4 (144) Linear Format Center and Corners	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Z1-Fiber distance	4	Z1-Fiber distance	
	5	Normal x at Z1	5	Normal x at Z1	RM
	6	Normal y at Z1	6	Normal x at Z1	IP
	7	Shear xy at Z1	7	Normal y at Z1	RM
	8	Shear angle at Z1	8	Normal y at Z1	IP
	9	Major principal at Z1	9	Shear xy at Z1	RM
	10	Minor principal at Z1	10	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	11	Z2-Fiber distance	
	12	Z2-Fiber distance	12	Normal x at Z2	RM
	13	Normal x at Z2	13	Normal x at Z2	IP
	14	Normal y at Z2	14	Normal y at Z2	RM
	15	Shear xy at Z2	15	Normal y at Z2	IP
	16	Shear angle at Z2	16	Shear xy at Z2	RM
	17	Major principal at Z2	17	Shear xy at Z2	IP
	18	Minor principal at Z2	18	Grid 1	
	19	von Mises or maximum shear at Z2	19-32	Same as 4 through 17 for corner 1	
	20	Grid 1	33	Grid 2	
	21-36	Same as items 4 through19 for corner 1	34-47	Same as items 4 through 17 for corner 2	
	37	Grid 2	48	Grid 3	
	38-53	Same as items 4 through19 for corner 2	49-62	Same as items 4 through 17 for corner 3	
	54	Grid 3	63	Grid 4	

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	55-70	Same as items 4 through 19 for corner 3	64-77	Same as items 4 through 17 for corner 4	
	71	Grid 4			
	72-87	Same as items 4 through 19 for corner 4			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD4[2] (95) Composite Center Only	2	Lamina Number	2	Lamina Number	
	3	Normal-1	3	Normal-1	RM
	4	Normal-2	4	Normal-2	RM
	5	Shear-12	5	Shear-12	RM
	6	Shear-1Z	6	Shear-1Z	RM
	7	Shear-2Z	7	Shear-2Z	RM
	8	Shear angle	8	Normal-1	IP
	9	Major principal	9	Normal-2	IP
	10	Minor principal	10	Shear-12	IP
	11	Maximum shear	11	Shear-1Z	IP
			12	Shear-2Z	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD4 (90) Nonlinear Format Center Only	2	Z1=Fiber Distance 1 (Plane stress only)	Not applicable		
	3	Stress-X (at Z1, if plane stress)			
	4	Stress-Y (at Z1, if plane stress)			
	5	Stress-Z (Plane strain only)			
	6	Stress-XY (at Z1, if plane stress)			
	7	Equivalent stress (at Z1, if plane stress)			
	8	Plastic strain (at Z1, if plane stress)			
	9	Creep strain (at Z1, if plane stress)			
	10	Strain-X (at Z1, if plane stress)			
	11	Strain-Y (at Z1, if plane stress)			
	12	Strain-Z (plane strain only)			
	13	Strain-XY (at Z1, if plane stress)			
	14-25	Items 2 through 13 repeated for fiber distance Z2 (plane stress only)			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD4 (254) Nonlinear Format Center and Corners	3	0 (Result at Center)	Not applicable		
	4	Z1=Fiber Distance			
	5	Stress in X at Z1			
	6	Stress in Y at Z1			
	7	Stress in Z at Z1			
	8	Stress in XY at Z1			
	9	Equivalent stress at Z1			
	10	Effective plastic/nlelastic strain at Z1			
	11	Effective creep strain at Z1			
	12	Strain in X at Z1			
	13	Strain in Y at Z1			
	14	Strain in Z at Z1			
	15	Strain in XY at Z1			
	16	Z2=Fiber Distance			
	17-27	Items 5 through 15 repeat for fiber distance Z2			
	28	Grid 1 ID			
	29–52	Same as items 4–27 for corner 1			
	53	Grid 2 ID			
	54–77	Same as items 4–27 for corner 2			
	78	Grid 3 ID			
	79–102	Same as items 4–27 for corner 3			
	103	Grid 4 ID			
	104–127	Same as items 4–27 for corner 4			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUAD8 (64) Linear Format Center and Corners		Same as CQUAD4(144)		Same as CQUAD4(144)	
CQUAD8[2] (96) Composite Center Only	2-11	Same as CQUAD4(95)	2-12	Same as CQUAD4(95)	
CQUAD8 (241) Nonlinear Format Center and Corners		Same as CQUAD4(254)	Not applicable		

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADFD (201) Hyperelastic with 4 grid points Center and Corners	2	Grid/Gauss	Not applicable		
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Pressure $p=\frac{1}{3}(\sigma_x+\sigma_y+\sigma_z)$			
	9	Volumetric strain J-1			
	10	Logarithmic strain-X			
	11	Logarithmic strain-Y			
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for 3 Gauss points			
CQUADFD (208) Hyperelastic with 5-9 grid points Center and Corners	2-13	Same as CQUADFD (201)	Not applicable		
	14-101	Repeat items 3 through 13 for 8 Gauss points			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADR (82) Center and Corners		Same as CQUAD4(144)		Same as CQUAD4(144)	
CQUADR (228) Center only (not supported by old formulation)		Same as CQUAD4(33)		Same as CQUAD4(33)	
CQUADR[2] (232) Composite Center Only (not supported by old formulation)	2-11	Same as CQUAD4(95)	2-12	Same as CQUAD4(95)	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX4 (243) Linear Format Center and Corners Grid and Gauss	3	0 (Result at Center)	3	0 (Result at Center)	
	4	Radial	4	Radial	RM
	5	Azimuthal	5	Radial	IP
	6	Axial	6	Azimuthal	RM
	7	Shear stress	7	Azimuthal	IP
	8	Maximum principal	8	Axial	RM
	9	Maximum shear	9	Axial	IP
	10	von Mises or octahedral	10	Shear	RM
	11	Grid 1 ID	11	Shear	IP
	12–18	Same as items 4 through 10 for corner 1	12	Grid 1 ID	
	19	Grid 2 ID	13–20	Same as items 4 through 11 for corner 1	
	20–26	Same as items 4 through 10 for corner 2	21	Grid 2 ID	
	27	Grid 3 ID	22-29	Same as items 4 through 11 for corner 2	
	28–34	Same as items 4 through 10 for corner 3	30	Grid 3 ID	
	35	Grid 4 ID	31-38	Same as items 4 through 11 for corner 3	
	36–42	Same as items 4 through 10 for corner 4	39	Grid 4 ID	
			40-47	Same as items 4 through 11 for corner 4	

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX4 (247) Nonlinear Format Center and Corners	3	0 (Result at Center)	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Equivalent Stress			
	9	Effective Plastic Strain			
	10	Effective Creep Strain			
	11	Radial Strain			
	12	Azimuthal Strain			
	13	Axial Strain			
	14	Shear Strain			
	15	Grid 1 ID			
	16–26	Same as items 4–14 for corner 1			
	27	Grid 2 ID			
	28–38	Same as items 4–14 for corner 2			
	39	Grid 3 ID			
	40–50	Same as items 4–14 for corner 3			
	51	Grid 4 ID			
	52–62	Same as items 4–14 for corner 4			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX4FD (262) Hyperelastic Grid or Gauss Center and Corners	2	Cord Type (GRID or GAUS)	Not applicable		
	3	Grid ID			
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Pressure			
	9	Volumetric Strain			
	10	Radial Strain			
	11	Azimuthal Strain			
	12	Axial Strain			
	13	Shear Strain			
	14-46	Items 3 through 13 repeated for 4 Grid/Gauss Points			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX8 (245) Linear Format Grid and Gauss Center and Corners	3	0 (Result at Center)	3	0 (Result at Center)	
	4	Radial	4	Radial	RM
	5	Azimuthal	5	Radial	IP
	6	Axial	6	Azimuthal	RM
	7	Shear stress	7	Azimuthal	IP
	8	Maximum principal	8	Axial	RM
	9	Maximum shear	9	Axial	IP
	10	von Mises or octahedral	10	Shear	RM
	11	Grid 1 ID	11	Shear	IP
	12–18	Same as items 4 through 10 for corner 1	12	Grid 1 ID	
	19	Grid 2 ID	13–20	Same as items 4 through 11 for corner 1	
	20–26	Same as items 4 through 10 for corner 2	21	Grid 2 ID	
	27	Grid 3 ID	22-29	Same as items 4 through 11 for corner 2	
	28–34	Same as items 4 through 10 for corner 3	30	Grid 3 ID	
	35	Grid 4 ID	31-38	Same as items 4 through 11 for corner 3	
	36–42	Same as items 4 through 10 for corner 4	39	Grid 4 ID	
			40-47	Same as items 4 through 11 for corner 4	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX8 (249) Nonlinear Format	3	0 (Result at Center)	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Equivalent Stress			
	9	Effective Plastic Strain			
	10	Effective Creep Strain			
	11	Radial Strain			
	12	Azimuthal Strain			
	13	Axial Strain			
	14	Shear Strain			
	15	Grid 1 ID			
	16–26	Same as items 4–14 for corner 1			
	27	Grid 2 ID			
	28–38	Same as items 4–14 for corner 2			
	39	Grid 3 ID			
	40–50	Same as items 4–14 for corner 3			
	51	Grid 4 ID			
	52–62	Same as items 4–14 for corner 4			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX8FD Hyperelastic Center and Corners (264)	3	Grid ID	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Pressure			
	9	Volumetric Strain			
	10	Radial Strain			
	11	Azimuthal Strain			
	12	Axial Strain			
	13	Shear Strain			
	14-46	Items 3 through 13 repeated for 4 corner Grid Locations			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CQUADX8FD Hyperelastic Gauss Locations (266)	3	Grid ID	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Pressure			
	9	Volumetric Strain			
	10	Radial Strain			
	11	Azimuthal Strain			
	12	Axial Strain			
	13	Shear Strain			
	14-101	Items 3 through 13 repeated 8 times for a total of 9 Gauss locations			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CROD (1) Linear Format		Same as CONROD(10)		Same as CONROD(10)	
CROD (89) Nonlinear Format		Same as CONROD(92)		Not applicable	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CSHEAR (4)	2	Maximum shear	2	Maximum shear	RM
	3[1]	Average shear	3	Maximum shear	IP
	4	Safety margin	4[1]	Average shear	RM
			5[1]	Average shear	IP

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTETRA (39) Linear Format	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	9[1]	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	IP
	13	von Mises or octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-69	Repeat items 5 through 17 for four corners	
	19	Third principal y cosine			
	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
	25	Third principal z cosine			
	26-109	Repeat items 5 through 25 for four corners			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTETRA (85) Nonlinear Format	2	Grid/Gauss	Not applicable		
	3	External grid ID (0=center)			
	4	Stress-X			
	5	Stress-Y			
	6	Stress-Z			
	7	Stress-XY			
	8	Stress-YZ			
	9	Stress-ZX			
	10	Equivalent stress			
	11	Effective plastic strain			
	12	Effective creep strain			
	13	Strain-X			
	14	Strain-Y			
	15	Strain-Z			
	16	Strain-XY			
	17	Strain-YZ			
	18	Strain-ZX			
	19-82	Repeat items 3 through 18 for four corners			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTETRAFD (205) Hyperelastic with 4 grid points	2-17	Same as CHEXAFD (202)	Not applicable		
CTETRAFD (210) Hyperelastic with 10 grid points	2-17	Same as CHEXAFD (202)	Not applicable		
	18-77	Repeat items 3 through 17 for 4 Gauss points			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX3 (242) Linear Format Center and Corners Grid or Gauss	3	0 (Result at Center)	3	0 (Result at Center)	
	4	Radial	4	Radial	RM
	5	Azimuthal	5	Radial	IP
	6	Axial	6	Azimuthal	RM
	7	Shear stress	7	Azimuthal	IP
	8	Maximum principal	8	Axial	RM
	9	Maximum shear	9	Axial	IP
	10	von Mises or octahedral	10	Shear	RM
	11	Grid 1 ID	11	Shear	IP
	12–18	Same as items 4 through 10 for corner 1	12	Grid 1 ID	
	19	Grid 2 ID	13–20	Same as items 4 through 11 for corner 1	
	20–26	Same as items 4 through 10 for corner 2	21	Grid 2 ID	
	27	Grid 3 ID	22-29	Same as items 4 through 11 for corner 2	
	28–34	Same as items 4 through 10 for corner 3	30	Grid 3 ID	
			31-38	Same as items 4 through 11 for corner 3	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX3 (246) Nonlinear Format Center and Corners	3	0 (Result at Center)	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Equivalent Stress			
	9	Effective Plastic Strain			
	10	Effective Creep Strain			
	11	Radial Strain			
	12	Azimuthal Strain			
	13	Axial Strain			
	14	Shear Strain			
	15	Grid 1 ID			
	16–26	Same as items 4–14 for corner 1			
	27	Grid 2 ID			
	28–38	Same as items 4–14 for corner 2			
	39	Grid 3 ID			
	40–50	Same as items 4–14 for corner 3			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX3FD Hyperelastic (261) Grid and Gauss	3	Grid ID	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Pressure			
	9	Volumetric Strain			
	10	Radial Strain			
	11	Azimuthal Strain			
	12	Axial Strain			
	13	Shear Strain			
	14-35	Items 3 through 13 repeated for 3 corner Grid/Gauss Locations			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX6 (244) Linear Format Grid and Gauss Center and Corners	3	0 (Result at Center)	3	0 (Result at Center)	
	4	Radial	4	Radial	RM
	5	Azimuthal	5	Radial	IP
	6	Axial	6	Azimuthal	RM
	7	Shear stress	7	Azimuthal	IP
	8	Maximum principal	8	Axial	RM
	9	Maximum shear	9	Axial	IP
	10	von Mises or octahedral	10	Shear	RM
	11	Grid 1 ID	11	Shear	IP
	12–18	Same as items 4 through 10 for corner 1	12	Grid 1 ID	
	19	Grid 2 ID	13–20	Same as items 4 through 11 for corner 1	
	20–26	Same as items 4 through 10 for corner 2	21	Grid 2 ID	
	27	Grid 3 ID	22-29	Same as items 4 through 11 for corner 2	
	28–34	Same as items 4 through 10 for corner 3	30	Grid 3 ID	
			31-38	Same as items 4 through 11 for corner 3	

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX6 (248) Nonlinear Format Center and Corners	3	0 (Result at Center)	Not applicable		
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Equivalent Stress			
	9	Effective Plastic Strain			
	10	Effective Creep Strain			
	11	Radial Strain			
	12	Azimuthal Strain			
	13	Axial Strain			
	14	Shear Strain			
	15	Grid 1 ID			
	16–26	Same as items 4–14 for corner 1			
	27	Grid 2 ID			
	28–38	Same as items 4–14 for corner 2			
	39	Grid 3 ID			
	40–50	Same as items 4–14 for corner 3			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRAX6FD (263) Center and Corners Grid or Gauss Hyperelastic	2	Cord Type (GRID or GAUS)	Not applicable		
	3	Grid ID			
	4	Radial Stress			
	5	Azimuthal Stress			
	6	Axial Stress			
	7	Shear Stress			
	8	Pressure			
	9	Volumetric Strain			
	10	Radial Strain			
	11	Azimuthal Strain			
	12	Axial Strain			
	13	Shear Strain			
	14-35	Items 3 through 13 repeated for 3 Grid/Gauss Points			

Element Stress (or Strain) Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRIA3 (74) Linear Format Center Only	2-17	Same as CQUAD4(33)	2-15	Same as CQUAD4(33)	
CTRIA3[2] (97) Composite Center Only	2-11	Same as CQUAD4(95)	2-12	Same as CQUAD4(95)	
CTRIA3 (88) Nonlinear Format Center Only	2-25	Same as CQUAD4(90)	Not applicable		

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRIA6 (75) Linear Format Center and Corners	2-70	Same as CQUAD4(144)	2-62	Same as CQUAD4(144)	
CTRIA6[2] (98) Composite Center Only	2-11	Same as CQUAD4(95)	2-12	Same as CQUAD4(95)	
CTRIA6 (240) Nonlinear Format Center and Corners	3-102	Same as CQUAD4(254)	Not applicable		

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRIAFD (206) Hyperelastic with 3 grid points	2-13	Same as CQUADFD(201)	Not applicable		
CTRIAFD (211) Hyperelastic with 6 grid points	2-13	Same as CQUADFD(201)	Not applicable		
	14-35	Items 3 through 12 repeated for 2 Gauss points			

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTRIAR (70) Center and Corners	1-70	Same as CQUAD4(144)	2-62	Same as CQUAD4(144)	
CTRIAR (227) Center Only (not supported by old formulation)	2-17	Same as CQUAD4(33)	2-15	Same as CQUAD4(33)	
CTRIAR[2] (233) Composite Center Only (not supported by old formulation)	2-11	Same as CQUAD4(95)	2-12	Same as CQUAD4(95)	

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./ Phase
CTUBE (3) Linear Format		Same as CONROD (10)		Same as CONROD(10)	
CTUBE (87) Nonlinear Format		Same as CONROD(92)	Not applicable		

7.3 Element Force Item Codes

All items are element forces (or moments) unless otherwise indicated.

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBAR (34)	2[1]	Bending End A plane 1	2[1]	Bending End A plane 1	RM
	3[1]	Bending End A plane 2	3[1]	Bending End A plane 2	RM
	4[1]	Bending End B plane 1	4[1]	Bending End B plane 1	RM
	5[1]	Bending End B plane 2	5[1]	Bending End B plane 2	RM
	6[1]	Shear plane 1	6[1]	Shear plane 1	RM
	7[1]	Shear plane 2	7[1]	Shear plane 2	RM
	8[1]	Axial force	8[1]	Axial force	RM
	9[1]	Torque	9[1]	Torque	RM
			10[1]	Bending End A plane 1	IP
			11[1]	Bending End A plane 2	IP
			12[1]	Bending End B plane 1	IP
			13[1]	Bending End B plane 2	IP
			14[1]	Shear plane 1	IP
			15[1]	Shear plane 2	IP
			16[1]	Axial force	IP
			17[1]	Torque	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBAR (100)	2	Station Distance/ Length	2	Station Distance/ Length	RM
	3	Bending Moment Plane 1	3	Bending Moment Plane 1	RM
	4	Bending Moment Plane 2	4	Bending Moment Plane 2	RM
	5	Shear Force Plane 1	5	Shear Force Plane 1	RM
	6	Shear Force Plane 2	6	Shear Force Plane 2	RM
	7	Axial	7	Axial	RM
	8	Torque	8	Torque	RM
	9	Item codes are given for End A. Addition of the quantity (K-1) * 7 to the item code points to the same information for other stations, where K is the station number. K=8 for End B and 2 through 7 for intermediate stations.	9	Bending Moment Plane 1	IP
			10	Bending Moment Plane 2	IP
			11	Shear Force Plane 1	IP
			12	Shear Force Plane 2	IP
			13	Axial	IP
			14	Torque (Item codes above are given for End A. For codes 2 through 14 at intermediate stations add (K-1) * 13 and K is the station number, and for codes at End B, K+number of stations plus 1.)	IP

Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBEAM	2	External grid point ID	2	External grid point ID	
(2)	3	Station distance/length	3	Station distance/length	
	4[1]	Bending moment plane 1	4[1]	Bending moment plane 1	RM
	5[1]	Bending moment plane 2	5[1]	Bending moment plane 2	RM
	6[1]	Web shear plane 1	6[1]	Web shear plane 1	RM
	7[1]	Web shear plane 2	7[1]	Web shear plane 2	RM
	8[1]	Axial force	8[1]	Axial force	RM
	9[1]	Total torque	9[1]	Total torque	RM
	10[1]	Warping torque	10[1]	Warping torque	RM
		(Item codes are given for End A. Addition of the quantity (K-1) 9 to the item code points to the same information for other stations, where K is the station number. K=11 for End B and 2 through 10 for intermediate stations.)	11[1]	Bending moment plane 1	IP
			12[1]	Bending moment plane 2	IP
			13[1]	Web shear plane 1	IP
			14[1]	Web shear plane 2	IP
			15[1]	Axial force	IP
			16[1]	Total torque	IP
			17[1]	Warping torque	IP
				(Item codes are given for End A. Addition of the quantity (K-1) 16 to the item code points to the same information for other stations, where K is the station number. K=11 for End B and 2 through 10 for intermediate stations.)	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBEAR (280)	2	Force-x	2	Force-x	RM
	3	Force-y	3	Force-y	RM
	4	Force-z	4	Force-z	RM
	5	Moment-x	5	Moment-x	RM
	6	Moment-y	6	Moment-y	RM
	7	Moment-z	7	Moment-z	RM
			8	Force-x	IP
			9	Force-y	IP
			10	Force-z	IP
			11	Moment-x	IP
			12	Moment-y	IP
			13	Moment-z	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3[1]	Bending moment plane 1	3[1]	Bending moment plane 1	RM
	4[1]	Bending moment plane 2	4[1]	Bending moment plane 2	RM
	5[1]	Shear plane 1	5[1]	Shear plane 1	RM
	6[1]	Shear plane 2	6[1]	Shear plane 2	RM
	7[1]	Axial force	7[1]	Axial force	RM
	8[1]	Torque (Item codes are given for End A. Item codes 9 through 15 point to the same information for End B.)	8[1]	Torque	RM
			9[1]	Bending moment plane 1	IP
			10[1]	Bending moment plane 2	IP
			11[1]	Shear plane 1	IP
			12[1]	Shear plane 2	IP
			13[1]	Axial force	IP
			14[1]	Torque (Item codes are given for End A. Item codes 15 through 27 point to the same information for End B.)	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CBUSH (102) Linear Format	2	Force-x	2	Force-x	RM
	3	Force-y	3	Force-y	RM
	4	Force-z	4	Force-z	RM
	5	Moment-x	5	Moment-x	RM
	6	Moment-y	6	Moment-y	RM
	7	Moment-z	7	Moment-z	RM
			8	Force-x	IP
			9	Force-y	IP
			10	Force-z	IP
			11	Moment-x	IP
			12	Moment-y	IP
			13	Moment-z	IP

Element Name (Code)	Real Element Forces	
	Item Code	Item
CBUSH (226) Nonlinear Format	2	Force-x
	3	Force-y
	4	Force-z
	11	Moment-x
	12	Moment-y
	13	Moment-z

Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CDAMP1 (20)		Same as CELAS1		Same as CELAS1	
CDAMP2 (21)		Same as CELAS1		Same as CELAS1	
CDAMP3 (22)		Same as CELAS1		Same as CELAS1	
CDAMP4 (23)		Same as CELAS1		Same as CELAS1	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CDUM3 thru CDUM9 (55-61)	2[1]	F1	2[1]	F1	RM
	3[1]	F2	3[1]	F2	RM
	4[1]	F3	4[1]	F3	RM
	5[1]	F4	5[1]	F4	RM
	6[1]	F5	6[1]	F5	RM
	7[1]	F6	7[1]	F6	RM
	8[1]	F7	8[1]	F7	RM
	9[1]	F8	9[1]	F8	RM
	10[1]	F9	10[1]	F9	RM
			11[1]	F1	IP
			12[1]	F2	IP
			13[1]	F3	IP
			14[1]	F4	IP
			15[1]	F5	IP
			16[1]	F6	IP
			17[1]	F7	IP
			18[1]	F8	IP
			19[1]	F9	IP

Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CELAS1	2[1]	Force	2[1]	Force	RM
(11)			3[1]	Force	IP
CELAS2 (12)		Same as CELAS1		Same as CELAS1	
CELAS3 (13)		Same as CELAS1		Same as CELAS1	
CELAS4 (14)		Same as CELAS1		Same as CELAS1	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CFAST (119)	2	mz bending End A plane 1	2	mz bending End A plane 1	RM
	3	my bending End A plane 2	3	my bending End A plane 2	RM
	4	mz bending End B plane 1	4	mz bending End B plane 1	RM
	5	my bending End B plane 2	5	my bending End B plane 2	RM
	6	fy shear force plane 1	6	fy shear force plane 1	RM
	7	fz shear force plane 2	7	fz shear force plane 2	RM
	8	fx axial force	8	fx axial force	RM
	9	mx torque	9	mx torque	RM
			10	mz bending End A plane 1	IP
			11	my bending End A plane 2	IP
			12	mz bending End B plane 1	IP
			13	my bending End B plane 2	IP
			14	fy shear force plane 1	IP
			15	fz shear force plane 2	IP
			16	fx axial force	IP
			17	mx torque	IP

Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CONROD (10)	2[1]	Axial force	2[1]	Axial force	RM
	3[1]	Torque	3[1]	Axial force	IP
			4[1]	Torque	RM
			5[1]	Torque	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CQUAD4 (33) Center Only	2[1]	Membrane force x	2[1]	Membrane force x	RM
	3[1]	Membrane force y	3[1]	Membrane force y	RM
	4[1]	Membrane force xy	4[1]	Membrane force xy	RM
	5[1]	Bending moment x	5[1]	Bending moment x	RM
	6[1]	Bending moment y	6[1]	Bending moment y	RM
	7[1]	Bending moment xy	7[1]	Bending moment xy	RM
	8[1]	Shear x	8[1]	Shear x	RM
	9[1]	Shear y	9[1]	Shear y	RM
			10[1]	Membrane force x	IP
			11[1]	Membrane force y	IP
			12[1]	Membrane force xy	IP
			13[1]	Bending moment x	IP
			14[1]	Bending moment y	IP
			15[1]	Bending moment xy	IP
			16[1]	Shear x	IP
			17[1]	Shear y	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CQUAD4[2]	2-3	Theory or blank		Not applicable	
(95)	4	Lamina number			
Composite	5	FP (failure index for direct stresses)			
Center Only	6	Failure mode for			
		Maximum strain theory			
	7	FB or -1 (failure index for interlaminar shear-stress)			
	8	MAX of FP, FB or -1			
	9	Failure flag			

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CQUAD4 (144) Center and Corners	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Membrane x	4	Membrane x	RM
	5	Membrane y	5	Membrane y	RM
	6	Membrane xy	6	Membrane xy	RM
	7	Bending x	7	Bending x	RM
	8	Bending y	8	Bending y	RM
	9	Bending xy	9	Bending xy	RM
	10	Shear x	10	Shear x	RM
	11	Shear y	11	Shear y	RM
	12	Grid 1	12	Membrane x	IP
	13-20	Same as 4 through 11 for corner 1	13	Membrane y	IP
			14	Membrane xy	IP
	21	Grid 2	15	Bending x	IP
	22-29	Same as 4 through 11 for corner 2	16	Bending y	IP
			17	Bending xy	IP
	30	Grid 3	18	Shear x	IP
	31-38	Same as 4 through 11 for corner 3	19	Shear y	IP
			20	Grid 1	
	39	Grid 4	21-36	Same as 4 through 19 for corner 1	
	40-47	Same as 4 through 11 for corner 4	37	Grid 2	
			38-53	Same as 4 through 19 for corner 2	
			54	Grid 3	
			55-70	Same as 4 through 19 for corner 3	
			71	Grid 4	
			72-87	Same as 4 through 19 for corner 4	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CQUAD8 (64) Center and Corners		Same as CQUAD4(144)		Same as CQUAD4(144)	
CQUAD8[2] (96) Composite Center Only		Same as CQUAD4(95)		Same as CQUAD4(95)	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CQUADR (82) Center and Corners		Same as CQUAD4(144)		Same as CQUAD4(144)	
CQUADR (228) Center Only (not supported by old formulation)		Same as CQUAD4(33)		Same as CQUAD4(33)	
CQUADR[2] (232) Composite Center Only (not supported by old formulation)		Same as CQUAD4(95)		Same as CQUAD4(95)	

Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CROD (1)		Same as CONROD(10)		Same as CONROD(10)	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CSHEAR (4)	2[1]	Force 4 to 1	2[1]	Force 4 to 1	RM
	3[1]	Force 2 to 1	3[1]	Force 2 to 1	RM
	4[1]	Force 1 to 2	4[1]	Force 1 to 2	RM
	5[1]	Force 3 to 2	5[1]	Force 3 to 2	RM
	6[1]	Force 2 to 3	6[1]	Force 2 to 3	RM
	7[1]	Force 4 to 3	7[1]	Force 4 to 3	RM
	8[1]	Force 3 to 4	8[1]	Force 3 to 4	RM
	9[1]	Force 1 to 4	9[1]	Force 1 to 4	RM
	10[1]	Kick force on 1	10[1]	Force 4 to 1	IP
	11[1]	Shear 12	11[1]	Force 2 to 1	IP
	12[1]	Kick force on 2	12[1]	Force 1 to 2	IP
	13[1]	Shear 23	13[1]	Force 3 to 2	IP
	14[1]	Kick force on 3	14[1]	Force 2 to 3	IP
	15[1]	Shear 34	15[1]	Force 4 to 3	IP
	16[1]	Kick force on 4	16[1]	Force 3 to 4	IP
	17[1]	Shear 41	17[1]	Force 1 to 4	IP
			18[1]	Kick force on 1	RM
			19[1]	Shear 12	RM
			20[1]	Kick force on 2	RM
			21[1]	Shear 23	RM
			22[1]	Kick force on 3	RM
			23[1]	Shear 34	RM
			24[1]	Kick force on 4	RM
			25[1]	Shear 41	RM
			26[1]	Kick force on 1	IP
			27[1]	Shear 12	IP
			28[1]	Kick force on 2	IP
			29[1]	Shear 23	IP
			30[1]	Kick force on 3	IP
			31[1]	Shear 34	IP
			32[1]	Kick force on 4	IP
			33[1]	Shear 41	IP

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CTRIA3 (74) Center Only		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTRIA3[2] (97) Composite Center Only		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIA6 (75) Center and Corners	1-38	Same as CQUAD4(144)	1-70	Same as CQUAD4(144)	
CTRIA6[2] (98) Composite Center Only		Same as CQUAD4(95)		Same as CQUAD4(95)	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CTRIAR (70) Center and Corners	1-38	Same as CQUAD4(144)	1-70	Same as CQUAD4(144)	
CTRIAR (227) Center Only (not supported by old formulation)	2-9	Same as CQUAD4(33)	2-17	Same as CQUAD4(33)	
CTRIAR[2] (233) Composite Center Only (not supported by old formulation)	2-9	Same as CQUAD4(95)		Not applicable	

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/ Phase
CTUBE (3)		Same as CONROD(10)		Same as CONROD(10)	
CVISC (24)		Not applicable		Same as CONROD(10)	
CWELDP (118) ELPAT or PARTPAT CWELDC (117) ELIMID or GRIDID with MSET=OFF CWELD (200) ELIMID or GRIDID with MSET=ON	2	mz bending End A plane 1	2	mz bending End A plane 1	RM
	3	my bending End A plane 2	3	my bending End A plane 2	RM
	4	mz bending End B plane 1	4	mz bending End B plane 1	RM
	5	my bending End B plane 2	5	my bending End B plane 2	RM
	6	fy shear force plane 1	6	fy shear force plane 1	RM
	7	fz shear force plane 2	7	fz shear force plane 2	RM
	8	fx axial force	8	fx axial force	RM
	9	mx torque	9	mx torque	RM
			10	mz bending End A plane 1	IP
			11	my bending End A plane 2	IP
			12	mz bending End B plane 1	IP
			13	my bending End B plane 2	IP
			14	fy shear force plane 1	IP
			15	fz shear force plane 2	IP
			16	fx axial force	IP
			17	mx torque	IP

7.4 Fluid Virtual Mass Pressure Item Codes

Element Name	Real Fluid Pressure		Complex Fluid Pressure		
	Code	Item	Code	Item	Real/Mag. or Imag./Phase
Plate	2	Fluid pressure	2	Pressure	RM
Family			3	Pressure	IP

Element Name (Code)	Code	Item
Heat Transfer Elements	2	Element type
	3	
	4	x gradient
	5	y gradient
	6	z gradient
	7	x flux
	8	y flux
	9	z flux
CHBDYE (107)	4	Applied load
	5	Free convection
	6	Forced convection
	7	Radiation
	8	Total
CHBDYG (108)	Same as CHBDYE	Same as CHBDYE
CHBDYP (109)	Same as CHBDYE	Same as CHBDYE

7.5 Slideline Contact Item Codes

Element Name (Code)	Real Element Data	
	Item Code	Item
CSLIFID (116)	1	Slave grid point
	2	Contact region identification number
	3	Master grid 1
	4	Master grid 2
	5	Surface coordinate
	6	Normal force
	7	Shear force
	8	Normal stress
	9	Shear stress
	10	Normal gap

Element Name (Code)	Real Element Data	
	Item Code	Item
	11	Slip
	12	Slip ratio (Shear force/u*normal force)
	13-14	Slip code (Character)

7.6 Element Strain Energy Item Codes

Element Name	Real Element Data	
	Item Code	Item
Element groups A and B	2	Element strain energy
Element groups A and B	3	Percent of total energy
Element group A	4	Element strain energy density

Remarks

Element group A includes elements CBAR, CBEAM, CBEND, CHEXA, CONROD, CPENTA, CQUAD4, CQUADR, CQUADX4, CQUADX8, CROD, CSHEAR, CTETRA, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, and CTUBE. Element group B includes elements of CELAS1, CELAS2, CELAS3, and CGAP.

Chapter 8: Degree-of-Freedom Sets

8.1 Degree-of-Freedom Set Definitions

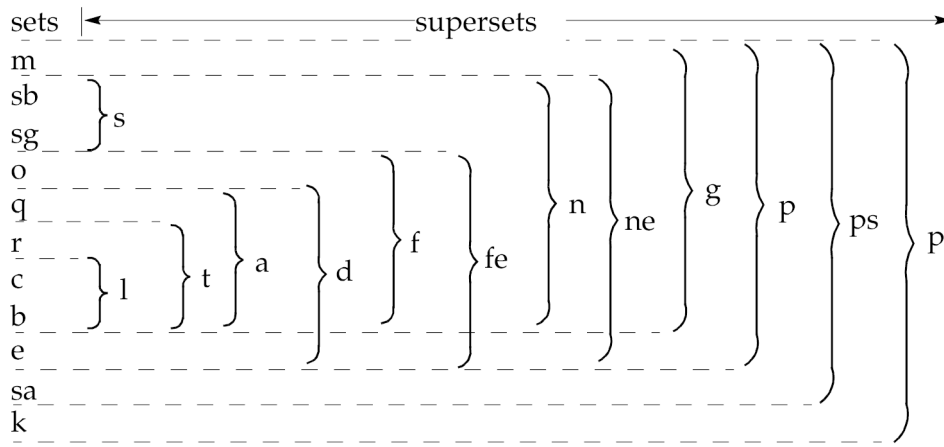
Each degree-of-freedom is a member of one mutually exclusive set. Set names have the following definitions.

Set Name	Definition
m	degrees-of-freedom eliminated by multipoint constraints
sb*	degrees-of-freedom eliminated by single-point constraints that are included in boundary condition changes and by the AUTOSPC feature
sg*	degrees-of-freedom eliminated by single-point constraints that are specified on the PS field on GRID Bulk Data entries
o	degrees-of-freedom omitted by structural matrix partitioning
q	generalized degrees-of-freedom for dynamic reduction or component mode synthesis
r	reference degrees-of-freedom used to determine free body motion
c	degrees-of-freedom that are free during component mode synthesis or dynamic reduction
b	degrees-of-freedom fixed during component mode analysis or dynamic reduction
e	extra degrees-of-freedom introduced in dynamic analysis
sa	permanently constrained aerodynamic degrees-of-freedom
k	aerodynamic degrees-of-freedom
u1 – u9	user defined degrees-of-freedom

* Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. Since these sets are not used explicitly in the solution sequences, this need not concern the user. However, those who use these sets in their own DMAPs should avoid redundant specifications when using these sets for partitioning or merging operations. That is, a degree-of-freedom should not be specified on both a PS field of a GRID entry (sg set) and on a selected SPC entry (sb

set). Redundant specifications will cause UFM 2120 in the VEC module and behavior listed in the *NX Nastran DMAP Programmer's Guide* for the UPARTN module. These sets are exclusive, however, from the other mutually exclusive sets.

Each degree-of-freedom is also a member of one or more combined sets called "supersets." Supersets have the following definitions:



Set Name (+ indicates union of two sets)	Meaning
s = sb + sg	all degrees-of-freedom eliminated by single point constraints
l = b + c	the structural degrees-of-freedom remaining after the reference degrees-of-freedom are removed (degrees-of-freedom left over)
t = l + r	the total set of physical boundary degrees-of-freedom for superelements
a = t + q	the set assembled in superelement analysis
d = a + e	the set used in dynamic analysis by the direct method
f = a + o	unconstrained (free) structural degrees-of-freedom
fe = f + s	free structural degrees-of-freedom plus extra degrees-of-freedom

Set Name (+ indicates union of two sets)	Meaning
n = f + e	all structural degrees-of-freedom not constrained by multipoint constraints
ne = n + e	all structural degrees-of-freedom not constrained by multipoint constraints plus extra degrees-of-freedom
g = n + m	all structural (grid) degrees-of-freedom including scalar degrees-of-freedom
p = g + e	all physical degrees-of-freedom
ps = p + sa	physical and constrained (SPCi) aerodynamic degrees-of-freedom
pa = ps + k	physical set for aerodynamics
(fr = o) + *l*	statically independent set minus the statically determinate supports (fr = f − q − r)
v = o + c + r	the set free to vibrate in dynamic reduction and component mode synthesis

The a-set and o-set are created in the following ways:

1. If only OMITi entries are present, then the o-set consists of degrees-of-freedom listed explicitly on OMITi entries. The remaining f-set degrees-of-freedom are placed in the b-set, which is a subset of the a-set.

2. If ASETi or QSETi entries are present, then the a-set consists of all degrees-of-freedom listed on ASETi entries and any entries listing its subsets, such as QSETi, SUPORTi, CSETi, and BSETi entries. Any OMITi entries are redundant. The remaining f-set degrees-of-freedom are placed in the o-set.

3. If there are no ASETi, QSETi, or OMITi entries present but there are SUPORTi, BSETi, or CSETi entries present, then the entire f-set is placed in the a-set and the o-set is not created.

4. There must be at least one explicit ASETi, QSETi, or OMITi entry for the o-set to exist, even if the ASETi, QSETi, or OMITi entry is redundant.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

ξ_o = rigid body (zero frequency) modal degrees-of-freedom

ξ_f = finite frequency modal degrees-of-freedom

ξ_i = $\xi_o + \xi_f$, the set of all modal degrees-of-freedom

One vector set is defined that combines physical and modal degrees-of-freedom:

u_h = $\xi_i + u_e$, the set of all modal degrees-of-freedom

The membership of each degree-of-freedom can be printed by use of the Bulk Data entries PARAM,USETPRT and PARAM,USETSEL.

8.2 Degree-of-Freedom Set Bulk Data Entries

Degrees-of-freedom are placed in sets as specified by the user on the following Bulk Data entries:

Name	Bulk Data Entry Name
m	MPC, MPCADD, RBAR, RBE1, RBE2, RBE3, RROD, RSPLINE, RTRPLT
sb	SPC, SPC1, SPCADD, BNDGRID, (PARAM,AUTOSPC,YES)
sg	GRID, GRDSET (PS field)
o	OMIT, OMIT1, GRID (SEID field), SESET
q	QSET, QSET1
r	SUPORT, SUPORT1
c	CSET, CSET1
b	BSET, BSET1
e	EPOINT
sa	CAEROi
k	CAEROi

Name	Bulk Data Entry Name
a	ASET, ASET1, Superelement exterior degrees-of-freedom, CSUPEXT
u0 – u9	USET, USET1

In superelement analysis, the appropriate entry names are preceded by the letters SE, and have a field reserved for the superelement identification number. This identification is used because a boundary (exterior) grid point may be in one mutually exclusive set in one superelement and in a different set in the adjoining superelement. The SE-type entries are internally translated to the following types for the referenced superelement.

Entry Type	Equivalent Type
SEQSETi	QSETi
SESUP	SUPORT
SECSETi	CSETi
SEBSETi	BSETi

8.3 User Defined Degree-of-Freedom Sets

The user defined degree-of-freedom sets U1 – U9 are created with the USET and USET1 bulk entries. They are typically used with DMAP alters.

Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases.

Set	Special Case Description	Solution Sequences
U1	No conflict exists. You may use in all situations.	
U2	Used with the RSCON parameter to select dof for SOL 103 constraint mode creation. See the RSCON parameter.	103
	Used by the Simcenter FE Model Update product (SOL 200, ANALYSIS=MODES) when the parameter PARAM, OPTEXIT, 8 exists.	200

Set	Special Case Description	Solution Sequences
U3	Used with the RSATT parameter to select dof for SOL 103 attachment mode creation. See the RSATT parameter.	103
U4 and U5	Optionally used with the AUTOMPC parameter to add and remove dof to the m-set, respectively. See the AUTOMPC parameter.	101–129, 153–187
U6	Optionally used with the RESVEC parameter to apply a unit load for residual vector creation. See the RESVEC parameter.	103, 106–112, 145, 146, 187, 200
U7 and U8	Used with the MBDEXPORT case control command to define the input/output dof when exporting a control system interface file. See the MBDEXPORT case control command.	103, 115
U9	No conflict exists. You may use in all situations.	

Chapter 9: Bulk Data Entries

9
BULK

9.1 Key to Descriptions

The name of the entry. Must be entered as shown.

The field names in fields 2 through 9 are for reference only. Names enclosed in quotation marks represent character constants; e.g., “THRU” on ASET1 entry.

A brief sentence about the function of the entry is given.

If a box in fields 2 through 9 is shaded, then the field must be left blank.

If the box in field 10 is shaded, then no continuation entries may follow.

Each of the fields is briefly described. Further details may be discussed under Remarks.

Character constants are enclosed in quotation marks to distinguish them from field names. Do not input these quotation marks.

Under contents, “blank” usually means that this feature can be optionally selected by the user. It may also mean the default action or value is described under Remarks.

The field’s type (Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, then a value must be specified by the user.

The remarks are generally arranged in order of importance and indicate such things as the entry’s relationship to other entries or commands, restrictions, and recommendations on usage, or further details regarding the fields.

ACMODL

Defines modeling parameters for the interface between the fluid and the

Format:

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	FSTOL				

Example:

ACMODL					0.002				

Field	Contents
INTER	Type of interface between the fluid and the structure. ter=“IDENT” or “DIFF”; Default=“DIFF”).
INFOR	Indicates whether FSET and
	integer > 0 or blank).
	SET1 entry that contains a list of structural grid points rk 2. (Integer > 0 or blank).
FSTOL	Tolerance, in units of length, used in (Real > 0.0; Default=0.001).

Remarks:

1. Only one ACMODL entry is allowed. I gram will assume INTER=“DIFF” and FSTOL=.001.
2. If INFOR=“ALL”, then both FSET checked at only those grid points
3. See the *NX Nastran Reference Man* determining the fluid-structure i
4. The points referenced by FSET and SSET must lie exactly on the fluid-structure interface. A fatal message is issued if: (a) INTER=“IDENT” and a point in SSET or I nt in FSET or SSET, res or FSET does not lie on
5. If fl ral surface, then INTE

The Bulk Data Section

The Bulk Data Section contains entries that specify model geometry, element connectivity, element and material properties, constraints (boundary conditions), and loads. Some entries, such as loads and constraints, are selected by an appropriate Case Control command.

Entries are prepared in either fixed or free field format. The descriptions in this section show only the fixed format. Entries that are used by the MSGMESH program are not included in this guide.

9.2 Bulk Data Entry Descriptions

Each Bulk Data entry is described as follows:

Description

A brief sentence about the function of the entry is given.

Format

The name of the entry is given in the first field. Subsequent fields are described under the Field and Contents section. Shaded fields must be left blank. If field 10 is shaded, then no continuation entries are permitted. Character strings enclosed in quotation marks must be specified without the quotation marks as shown in the example.

Example

A typical example is given.

Field and Contents

Each of the fields 2 through 9 that is named in the Format section is briefly described under Contents. The field's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The field must be specified by the user if no default value is given.

Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as how the Bulk Data entry is selected in the Case Control Section, its relationship to other entries, restrictions and recommendations on its use, and further descriptions of the fields.

9.3 Bulk Data Syntax Rules

Real, Integer, and Character Input Data

The three types of input data are described as follows:

Integer	Cannot contain a decimal point.
Real	Must contain a decimal point.
Character	Can be alphanumeric. The first character cannot be numeric. It must be 8 characters or less in length. Imbedded blanks are not allowed.

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

7.0	.7E1	0.7+1
.70+1	7.E+0	70.-1

Free and Fixed Field Format Summary

Bulk data inputs have 80 characters on each line and the lines are divided into fields. How the lines are divided depends upon the data format and if you are using small fields or large fields.

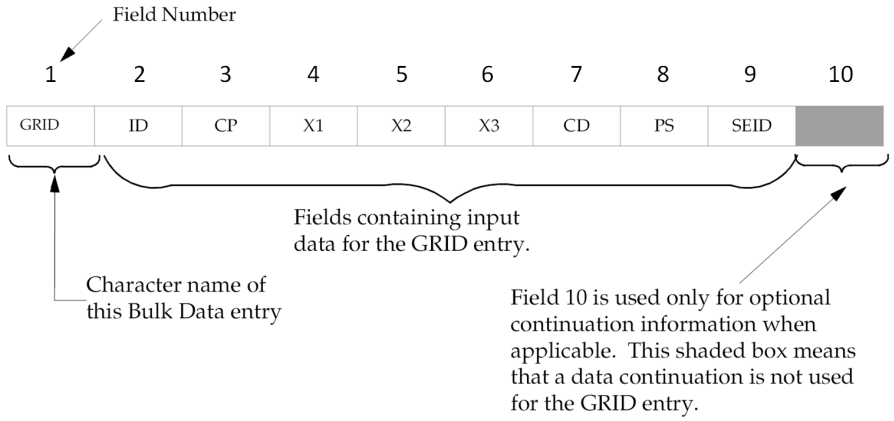
- Free format uses commas to separate the data fields. When you are using small fields with free format, a maximum of 8 characters is allowed per field. When you are using large fields with free format, some of the fields are extended to a maximum of 16 characters per field.

- Fixed format data is aligned in fixed columns of specific width. When you are using small fields, the fixed column width is 8 characters. When you are using large fields, some of the fields are extended to a fixed width of 16 characters.

Bulk Entry Summary

In the Quick Reference Guide, each row of a bulk entry definition contains 10 fields. The first field in the first row contains the character name of the bulk entry (e.g., GRID, CBAR, MAT1, etc.). Then fields two through nine contain data. The tenth field in each row is reserved for a continuation character when more lines follow the first.

For example, consider the format of the GRID bulk entry.

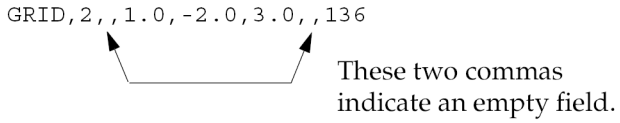


Free Field Format

In free field format, data fields are separated by commas. For example, the GRID entry

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

appears in free field format with small fields as:



The rules for free field format are as follows:

- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integers and characters must be eight characters or less. Exceeding eight characters will cause a fatal error. This also applies to data defined in large fields.
- Free field format allows continuation data on a single input line. For example, the following SPC1 bulk entry is defined on a single line:

  ```
  SPC1,1,123456,109,110,111,112,113,114,+LINE2,+LINE2,115,116,117
  ```

  Removing the continuation fields will also work for this example:

  ```
  SPC1,1,123456,109,110,111,112,113,114,115,116,117
  ```

  See Continuations.

- Free field data cannot contain embedded blanks. The following is an example of an embedded blank:

```
GRID,2,,1     0,-2.0,3.0,,136
```

Embedded blank not allowed

- A dollar sign terminates the entry and comments may follow.

Fixed Field Format

In fixed format, the data fields are aligned in fixed columns of specific width.

When you are using small fields, the column width is a fixed 8 characters. For example, the GRID entry

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

appears in fixed field format with small fields as:

```
GRID           2              1.0    -2.0     3.0             136
```

The first two lines in this example are used to show the column alignment.

The rules for fixed field format are as follows:

- Data in the first and in the last field of each input line must be left justified.
- Data in the other columns can be defined anywhere within the bounds of a specific column.
- Integers and characters must be eight characters or less. Exceeding eight characters will cause a fatal error. This also applies to data defined in large fields.
- You cannot enter more characters than the field width.
- A dollar sign terminates the entry and comments may follow.

Small Fields

Small fields have a maximum of 8 characters per field:

1	2	3	4	5	6	7	8	9	10
<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->	<-8 Char->

The following is an example of the GRID entry with small fields:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

The fixed field input for this example is:

```
GRID            2              1.0     -2.0      3.0              136
```

The free field input for this example is:

```
GRID,2,,1.0,-2.0,3.0,,136
```

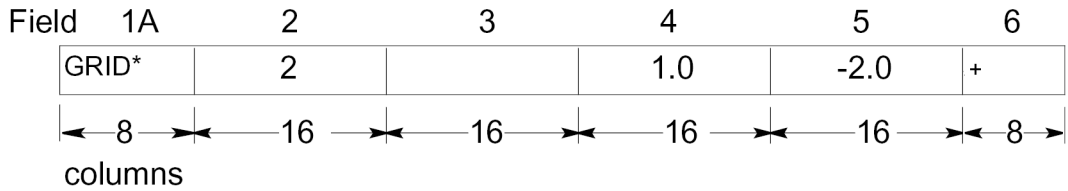
See Continuations for more examples.

Large Fields

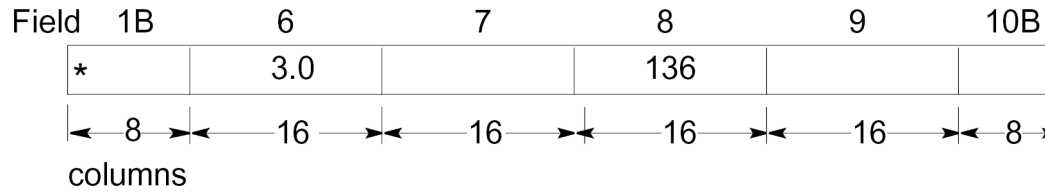
Large field format is used when the small field format does not provide enough significant digits. The first and last field of each line contains eight characters, and the fields in between contain 16 characters. Large field entries are denoted by an asterisk (*) immediately following the character string in the first field of the first line and also in the first field of any continuation lines.

The following is an example of the GRID bulk entry example in large field format:

First Line:



Second Line:



The fixed field input for this example is:

```
GRID*                  2                                1.0             -2.0+
*                    3.0                                136
```

The free field input for this example is:

```
GRID*,2,,1.0,-2.0,+
*,3.0,,136
```

See Continuations for more examples.

Continuations

Continuations are used when data continues beyond the first input line. There are several ways to define continuations. Each is described below using the following example bulk entry:

Example Bulk Entry:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	EI	E2	F1	F2	
	K1	K2	I12						

Small Field Continuation Format 1:

You can enter + or * in the last field of a line and + at the beginning of the next line. For example:

PBAR	39	6	2.9	1.86	2.92	.48			+
+	0.	0.	0.	1.	1.	1.	1.	0.	+
+	.86	.86							

or

PBAR	39	6	2.9	1.86	2.92	.48			*
+	0.	0.	0.	1.	1.	1.	1.	0.	*
+	.86	.86							

Small Field Continuation Format 2:

You can simply leave the continuation fields blank. For example:

PBAR	39	6	2.9	1.86	2.92	.48			
	0.	0.	0.	1.	1.	1.	1.	0.	
	.86	.86							

Small Field Continuation Format 3:

You can enter a unique character string in the last field of a line, then a continuation line begins with the same character string. In the example below, +PB1 at the end of the first line is also included at the beginning of the second line, +PB2 at the end of the second line is also included at the beginning of the third line, and so on.

This format allows the continuation lines to be unsorted in the bulk data section. This was useful when punch cards where used and sorting them was difficult.

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

Large Field Continuation Format 1:

You can enter + or * in the last field of a line and the first field on the continuation must include * to designate the large field format. For example:

PBAR*	39	6	2.9	1.86	+
*	2.92	.48			+
*	0.	0.	0.	1.	+
*	1.	1.	1.	0.	+
*	.86	.86			

or

PBAR*	39	6	2.9	1.86	*
*	2.92	.48			*
*	0.	0.	0.	1.	*

*	1.	1.	1.	0.	*
*	.86	.86			

Large Field Continuation Format 2:

You can simply leave the last continuation fields blank, and the first field must include the * to designate the large field format. For example:

PBAR*	39	6	2.9	1.86	
*	2.92	.48			
*	0.	0.	0.	1.	
*	1.	1.	1.	0.	
*	.86	.86			

Large Field Continuation Format 3:

You can enter a unique character string in the last field of a line, then the next line begins with * plus the same character string to designate the large field format. In the large field example below, PB1 at the end of the first line is also included at the beginning of the second line, PB2 at the end of the second line is also included at the beginning of the third line, and so on.

This format allows the continuation lines to be unsorted in the bulk data section. This was useful when punch cards where used and sorting them was difficult.

PBAR*	39	6	2.9	1.86	+PB1
*PB1	2.92	.48			+PB2
*PB2	0.	0.	0.	1.	+PB3
*PB3	1.	1.	1.	0.	+PB4
*PB4	.86	.86			

Replication

Replication is a limited data generation capability which may be used in a fixed or free-field format for small field input and in some limited cases for large field input. (The *NX Nastran User's Guide* contains additional information on replication.)

1. Duplication of fields from the preceding entry is accomplished by coding the symbol =.

2. Duplication of all trailing fields from the preceding entry is accomplished by coding the symbol ==.

3. Incrementing a value from the previous entry is indicated by coding *x or *(x), where x is the value of the increment. “x” should be a real number for real fields or an integer for integer fields.

4. Repeated replication is indicated by coding =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.

5. Data items may be enclosed within parentheses or the parentheses may be deleted.

6. If NASTRAN MESH is specified in the File Management Section, MSGMESH includes the capabilities.

   - Continuation entry fields may be incremented or decremented.

   - Repeated replication is indicated by coding =(n) in field 1, where n is the number of entry images to be generated using the values of increments from the current or preceding replication entry.

     Entered entries:

     ```
     GRID,101 ,17,1 .0,10.5,,17,3456
     =(4), *(1),=,*(0.2),==$
     ```

     Generated entries:

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.5		17	3456		
GRID	103	17	1.4	10.5		17	3456		
GRID	104	17	1.6	10.5		17	3456		
GRID	105	17	1.8	10.5		17	3456		

   - A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

     ```
     BSET1,123,1,2,3,4,5,6,7
     ,,*7,*7,*7,*7,*7,*7,*7
     =(3)
     ```

     Generated entries:

BSET1	123	1	2	3	4	5	6	7	+00001
++00001		8	9	10	11	12	13	14	+00002
++00002		15	16	17	18	19	20	21	+00003
++00003		22	23	24	25	26	27	28	+00004
++00004		29	30	31	32	33	34	35	+00005

- An “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of 9 entries may be replicated as a group. The default continuation entry increment is 10. Example:

  Entered entries:

  ```
  CTRIA3,10,1,1,10,11/+C1
  =(D),*(1),=,=,*(1),*(1)/*(20)
  +C1,,,2.0,1.0,1.0
  =(2),==
  ```

  Generated entries:

CTRIA3	10	1	1	10	11				+C1
+C1			2.0	1.0	1.0				
CTRIA3	11	1	1	11	12				+C21
+C21			2.0	1.0	1.0				
CTRIA3	12	1	1	12	13				+C41
+C41			2.0	1.0	1.0				

- Parentheses are optional on replication entries and an equal sign may replace an asterisk.

  The following is an example of the use of replication, automatic continuation field generation, and the free field format.

  ```
  GRID,101 ,17,1.0,10.5,,17,3456
  =,*1,=,*0.2, *(0.1), == $ COMMENTS MAY APPEAR AFTER $
  =3
  EIGR,13,GIV,,30.
  ,MASS
  CBAR,1 ,1 ,101 ,102,0.,0.,1.,,+0
  =,*1,=,*1,*1====*1
  +0,56
  *1,=$
  ```

  The above free-field entries will generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

  > **Note**
  >
  > A “,” should always be used after the “*1” for the continuation increment even if fixed field format is being used.

CBAR	1	1	101	102	0.	0.	1.		+0
+0	56								

CBAR	2	1	102	103	0.	0.	1.		+1
+1	56								
EIGR	13			30.					+000001
++000001	MASS								
GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.6		17	3456		
GRID	103	17	1.4	10.7		17	3456		
GRID	104	17	1.6	10.8		17	3456		
GRID	105	17	1.8	10.9		17	3456		

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user's responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

9.4 Bulk Data Entry Summary

This section contains a summary of all Bulk Data entries under the following headings:

- Geometry
- Elements
- Materials
- Material Selection by Property
- Constraints and Partitioning
- Loads
- Solution Control
- Miscellaneous

Geometry

Grid Points

GRDSET Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

GRID	Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.
ROTORB	Defines bearing grids for the stationary portion of a model in a rotor dynamics solution.
ROTORG	Defines the rotating portion of a model in a rotor dynamics solution.
SEQEP	Redefines the sequence of extra points to optimize bandwidth.
SEQGP	Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

Coordinate Systems

BAROR	Defines default values for field 3 and fields 6 through 8 of the CBAR entry.
BEAMOR	Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.
CORDiC	Cylindrical coordinate system definition.
CORDiR	Rectangular coordinate system definition.
CORDiS	Spherical coordinate system definition.
CORD3G	Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system.
MATCID	Overrides the material coordinate system for CHEXA, CPENTA, CTETRA, and CPYRAM solid elements when the elements reference a PSOLID property. Also overrides the material coordinate system for CHEXA and CPENTA solid elements when the elements reference a PCOMPS property.

Scalar Points

EPOINT	Defines extra points for use in dynamic problems.
SEQEP	Redefines the sequence of extra points to optimize bandwidth.
SEQGP	Grid and scalar point number re-sequencing.
SPOINT	Defines scalar points.

Fluid Points

ACMODL	Defines modeling parameters for the interface between the fluid and the structure.
GRID	Defines fluid points in coupled fluid-structural analysis.

Cyclic Symmetry

CYAX	Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.
CYJOIN	Defines the boundary points of a segment in cyclic symmetry problems.

Superelement Analysis

CSUPER	Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.
CSUPEXT	Assigns exterior points to a superelement.
EXTRN	Defines a boundary connection for an external superelement.
GRID	Defines interior points for a superelement.
POINT	Define geometric points for use with the SELOC entry.
RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
SEBNDRY	Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
SEBULK	Defines superelement boundary search options and a repeated, mirrored, or collector superelement.
SECONCT	Explicitly defines grid and scalar point connection procedures for a partitioned superelement.
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SEEXCLD	Defines grid points that will be excluded during the attachment of a partitioned superelement.

SELABEL	Defines a label or name to be printed in the superelement output headings.
SELOC	Defines a partitioned superelement relocation by listing three non-colinear points in the superelement and three corresponding points not belonging to the superelement.
SEMPLN	Defines a mirror plane for mirroring a partitioned superelement.
SEQSEP	Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.
SESET	Defines interior grid points for a superelement.
SETREE	Specifies superelement reduction order.

Aeroelastic Control Points

AECOMP	Defines a component for use in aeroelastic monitor point definition.
AECOMPL	Defines a component for use in aeroelastic monitor point definition as a union of other components.
MONPNT1	Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.
UXVEC	Specification of a vector of aerodynamic control point (extra point) values.

Elements

Line Elements

BAROR	Default for orientation and property for CBAR.
BEAMOR	Default for orientation and property for CBEAM.
BOLT	Selects the CBEAM/CBAR elements to be included in the bolt pre-load calculation.
CBAR	Defines a simple beam element.
CBEAM	Defines a beam element.
CBEND	Connection definition for a curved beam.

CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CFAST	Defines a shell patch connection with direct stiffness input.
CONROD	Defines a rod element without reference to a property entry.
CROD	Defines a tension-compression-torsion element.
CTUBE	Defines a tension-compression-torsion tube element.
CWELD	Defines a weld or fastener connecting two surface patches or points.
PBAR	Defines the properties of a simple beam element (CBAR entry).
PBARL	Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.
PBCOMP	Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry.
PBEAM	Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.
PBEAML	Defines the properties of a beam element by cross-sectional dimensions.
PBEND	Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PFAST	Defines properties for CFAST connector.
PROD	Defines the properties of a rod element (CROD entry).
PTUBE	Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).
PWELD	Defines the properties of connector (CWELD) elements.
SWLDPRM	Define parameters for CWELD/CFAST connectors.

Surface Elements

CPLSTNi	Defines a plane strain quadrilateral or triangular element for use in linear or nonlinear analysis.

CPLSTSi	Defines a plane stress quadrilateral or triangular element for use in linear or nonlinear analysis.
CQUAD	Defines a plane-strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.
CQUAD4	Defines an isoparametric membrane-bending or plane-strain quadrilateral plate element.
CQUAD8	Defines a curved quadrilateral shell or plane-strain element with eight grid points.
CQUADR	Defines an isoparametric membrane and bending quadrilateral plate element.
CSHEAR	Defines a shear panel element.
CTRIA3	Defines an isoparametric membrane-bending or plane-strain triangular plate element.
CTRIA6	Defines a curved triangular shell element or plane-strain elemetn with six grid points.
CTRIAR	Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element.
PCOMP	Defines the properties of an n-ply composite material laminate.
PCOMPG	Defines the properties of an n-ply composite material laminate with global ply IDs.
PGPLSN	Defines the properties for generalized plane strain elements. Valid for SOL 401.
PLPLANE	Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane-strain or axisymmetric element.
PPLANE	Defines the properties of plane stress elements or plane strain elements.
PSHEAR	Defines the properties of a shear panel (CSHEAR entry).
PSHELL	Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.
PSHL3D	Defines the properties of 3D-shell elements. Valid for SOLs 601 and 701 only.

SNORM	Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

Solid Elements

CHEXA	Defines the connections of the six-sided solid element with eight to twenty grid points.
CHEXCZ	Defines the connections of the six-sided cohesive element with eight to twenty grid points.
CPENTA	Defines the connections of a five-sided solid element with six to fifteen grid points.
CPENTCZ	Defines the connections of a five-sided cohesive element with six to fifteen grid points.
CPYRAM	Defines the connection of the five-sided solid element with five to thirteen grid points.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
PCOMPS	Defines the properties of an n-ply composite material laminate for CHEXA and CPENTA solid elements.
PLSOLID	Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
PSOLCZ	Defines the properties of cohesive elements (CHEXCZ and CPENTCZ entries).
PSOLID	Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

Scalar and Bushing Elements

CBUSH	Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.
CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CELASi	Connection definition for scalar spring, also property definition for i=2 or 4.
GENEL	Defines a general element.

PBUSH	Defines the nominal property values for a generalized spring-and-damper structural element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PBUSHT	Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
PELAS	Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
PELAST	Defines the frequency dependent properties for a PELAS Bulk Data entry.

Axisymmetric Elements

CCHOCK3	Defines a triangular chocking element.
CCHOCK4	Defines a quadrilateral chocking element.
CCHOCK6	Defines a triangular chocking element with midside nodes.
CCHOCK8	Defines a quadrilateral chocking element with midside nodes.
CQUADX4	Defines an isoparametric and axisymmetric quadrilateral cross-section ring element for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CQUADX8	Defines an isoparametric and axisymmetric quadrilateral cross-section ring element with midside nodes for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRAX3	Defines an isoparametric and axisymmetric triangular cross-section ring element for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRAX6	Defines an isoparametric and axisymmetric triangular cross-section ring element with midside nodes for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
PCHOCK	Defines the properties of a chocking element described on a CCHOCKi entry.

PSOLID	Defines the properties of axisymmetric elements (CQUADX4, CQUADX8, CTRAX3, and CTRAX6 entries).

Rigid Elements

RBAR	Defines a rigid bar with six degrees-of-freedom at each end.
RBE1	Defines a rigid body connected to an arbitrary number of grid points.
RBE2	Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.
RBE3	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.
RROD	Defines a pin-ended element that is rigid in translation.
RSPLINE	Defines multipoint constraints for the interpolation of displacements at grid points.
RSSCON	Defines multipoint constraints to model clamped connections of shell-to-solid elements.
RTRPLT	Defines a rigid triangular plate.

Mass Elements

CMASSi	Connection definition for scalar mass, also property definition for i=2 or 4.
CONM1	Defines a 6 x 6 symmetric mass matrix at a geometric grid point.
CONM2	Defines concentrated mass at a grid point.
NSM	Defines a set of non-structural mass by ID.
NSM1	Alternate form for NSM entry. Defines non-structural mass entries by VALUE, ID list.
NSMADD	Non-structural mass set combination. Defines non-structural mass as the sum of the sets listed.
NSML	Defines a set of lumped non-structural mass by ID.
NSML1	Alternate form for NSML entry. Defines lumped non-structural mass entries by VALUE, ID list.
PMASS	Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

Damping Elements

CBUSH1D	See line elements.
CDAMPi	Connection definition for a scalar damper, also property definition for i=2 or 4.
CVISC	Defines a viscous damper element.
PBUSH1D	See line elements.
PDAMP	Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.
PDAMP5	Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.
PDAMPT	Defines the frequency-dependent properties for a PDAMP Bulk Data entry.
PVISC	Defines properties of a one-dimensional viscous damping element (CVISC entry).

Fluid and Acoustic Elements

BFLUID	Defines a fluid boundary by referencing BSURFS, BCPROPS or BEDGE entries for SOL 601,106 only.
CAABSF	Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.
CHACAB	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHACBR	Defines the acoustic barrier element.
CHEXA	Connection definition for a hexahedron element in coupled fluid-structural analysis.
CPENTA	Connection definition for a pentahedron element in coupled fluid-structural analysis.
CPYRAM	Connection definition for a pyramid element in coupled fluid-structural analysis.
CTETRA	Connection definition for a tetrahedron element in coupled fluid-structural analysis.

ELIST	Defines a list of structural elements for virtual fluid mass.
PAABSF	Defines the properties of a frequency-dependent acoustic absorber element.
PACABS	Defines the properties of the acoustic absorber element.
PACBAR	Defines the properties of the acoustic barrier element.
PANEL	Selects sets of structural grid points, elements, or physical properties that define one or more panels.
PSOLID	Defines the properties of solid elements (CHEXA, CPENTA, CPYRAM, CTETRA entries).

Heat Transfer Elements

BDYOR	Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.
CHBDYi	Connection definition for surface element (CHBDYE, CHBDYG, CHBDYP).
PHBDY	A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

The following elastic elements may also be used as heat conduction elements.

Linear:	CBAR, CROD, CONROD, CTUBE, CBEAM, CBEND.
Membrane:	CTRIA3, CTRIA6, CQUAD4, CQUAD8.
Solid:	CTETRA, CHEXA, CPENTA.

Dummy Elements

ADUMi	Defines attributes of the dummy elements ($1 \le i \le 9$).
CDUMi	Defines a dummy element ($1 \le i \le 9$).
PDUMi	Defines the properties of a dummy element ($1 \le i \le 9$). Referenced by the CDUMi entry.
PLOTEL	Defines a one-dimensional dummy element for use in plotting.

Contact, Glue, and Gap Elements

BCONP	Defines the parameters for a contact region and its properties.
BCPROP	Defines a surface-to-surface contact or glue region by property IDs of shell elements.
BCPROPS	Defines a surface-to-surface contact or glue region by PSOLID Property ID.
BCRPARA	Defines parameters for a surface-to-surface contact region.
BCTADD	Defines a surface-to-surface contact set as a union of contact sets defined on BCTSET entries.
BCTPARA	Defines parameters for a SOL 601 or 701 contact set.
BCTPARM	Control parameters for the surface-to-surface contact algorithm.
BCTSET	Defines contact pairs of a 2D (SOL 601 only) or 3D contact set (SOLs 101, 103, 111, 112, 601 and 701).
BEDGE	Defines a glue or contact edge region.
BFRIC	Defines frictional properties between two bodies in contact.
BGADD	Defines a surface-to-surface glue set as a union of glue sets defined on BGSET entries.
BGPARM	Control parameters for the glue algorithm.
BGSET	Defines glued contact pairs.
BLSEG	Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.
BSURF	Defines a 3D contact or glue region by shell element IDs.
BSURFS	Defines a 3D contact or glue region by the faces of the CHEXA, CPENTA or CTETRA elements.
BWIDTH	Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.
CGAP	Defines a gap or friction element.
PGAP	Defines the properties of the gap element (CGAP entry).

Crack Tip Elements

CRAC2D	Defines a two-dimensional crack tip element.
CRAC3D	Defines a three-dimensional crack tip element.
CRAKTP	Specifies information related to a crack tip.
PRAC2D	Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.
PRAC3D	Defines the properties of the CRAC3D structural element.
VCEV	Defines virtual crack tip extension vectors.

Aerodynamic Elements

AEFACT	Defines real numbers for aeroelastic analysis.
AELINK	Defines relationships between or among AESTAT and AESURF entries.
AELIST	Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity.
AESTAT	Specifies rigid body motions to be used as trim variables in static aeroelasticity.
AESURF	Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.
AESURFS	Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry.
CAERO1	Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords.
CAERO2	Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.
CAERO3	Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.
CAERO4	Defines an aerodynamic macro element for Strip theory.
CAERO5	Defines an aerodynamic macro element for Piston theory.

CSSCHD	Defines a scheduled control surface deflection as a function of Mach number and angle of attack.
PAERO1	Defines associated bodies for the panels in the Doublet-Lattice method.
PAERO2	Defines the cross-sectional properties of aerodynamic bodies.
PAERO3	Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.
PAERO4	Defines properties of each strip element for Strip theory.
PAERO5	Defines properties of each strip element for Piston theory.

Aerodynamic to Structure Interconnection

SPLINE1	Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE2	Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE3	Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.
SPLINE4	Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.
SPLINE5	Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points.

Rotor Dynamics Elements

CBEAR	Defines speed-dependent bearing connectivity.
PBEAR	Defines speed-dependent stiffness and viscous damping matrices for bearing connection.

Materials

Isotropic

MAT1	Defines the material properties for linear isotropic materials.
MAT4	Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.
MATCZ	Defines damage model and material properties for cohesive elements (CHEXCZ and CPENTCZ).
MATHE	Specifies hyperelastic (rubber-like) material properties for advanced nonlinear analysis.
MATHEM	Specifies Mullins effect (Ogden-Roxburgh model) on hyperelastic (MATHE) material.
MATHEV	Specifies viscoelastic effect (Holzapfel model) on hyperelastic (MATHE) material.
MATHP	Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
MATPLCY	Specifies properties for a material used to model cyclic plasticity. Valid for SOLs 601 and 701 only.
MATSMA	Specifies properties for shape-memory alloy material.
MATVE	Specifies viscoelastic material properties for advanced nonlinear analysis.
PLCYISO	Specifies isotropic hardening rule for plastic-cyclic material. Valid for SOLs 601 and 701 only.
PLCYKIN	Specifies kinematic hardening rule for plastic-cyclic material. Valid for SOLs 601 and 701 only.
PLCYRUP	Specifies the rupture criterion for plastic-cyclic material. Valid for SOLs 601 and 701 only.
RADM	Defines the radiation properties of a boundary element for heat transfer analysis.
TABVE	Defines series of moduli and decay coefficients used for viscoelastic material definition.

Anisotropic

MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT3	Defines linear orthotropic material properties for axisymmetric, plane stress, and plane strain elements. Not valid for use with chocking elements.
MAT5	Defines the thermal material properties for anisotropic materials.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MAT9	Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see the PSOLID entry description).
MAT11	Defines linear orthotropic material properties for axisymmetric, plane stress, plane strain, and solid elements.
MATDMG	Defines damage-related material properties for progressive ply failure in composite solid elements. Used in combination with MAT1 or MAT11 entries. Valid for SOL 401 only.
MATFT	Defines material properties for use with composite ply failure theories.

Temperature Dependent

MATTi	Table references for temperature-dependent MATi materials.
MATTC	Specifies temperature-dependent creep coefficients on CREEP entry fields via TABLEM1 entries.
RADMT	Specifies table references for temperature dependent RADM entry radiation boundary properties.
TABLEMi	Tabular functions for generating temperature-dependent material properties.
TABLEST	Table references for temperature dependent MATS1 materials.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

TEMPPi	Defines a temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TMCPARA	Defines parameters for SOL 601,153 and SOL 601,159 thermo-mechanical coupling (TMC) analysis.

Stress Dependent

CREEP	Defines creep characteristics based on experimental data or known empirical creep law.
MATG	Defines the material properties for gasket materials.
MATS1	Specifies stress-dependent material properties for use in applications involving nonlinear materials.
MATSR	Specifies strain-rate dependent properties for use with a MATS1 entry. Valid for SOLs 601 and 701 only.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Fluid

MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis.
MATPOR	Defines acoustic absorber material properties.
MFLUID	Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

Constraints and Partitioning

Single Point Constraints

GRID	Includes single point constraint definition.
GRDSET	Includes deafult for single point constraints.

SPC	Defines a set of single-point constraints and enforced motion—enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis.
SPC1	Defines a set of single point constraints.
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.
SPCOFFi	Defines degrees-of-freedom to be excluded from the AUTOSPC operation.

Multipoint Constraints

CWELD	Defines a weld or fastener connecting two surface patches or points.
MPC	Defines a linear relationship for two or more degrees-of-freedom.
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.
PWELD	Defines the properties of connector (CWELD) elements.
RBAR	Defines multipoint constraints for rigid bar.
RBEi	Defines multipoint constraints for RBE1, RBE2, and RBE3.
RROD	Defines multipoint constraints for rigid rod.
RSPLINE	Defines multipoint constraints for spline element.
RTRPLT	Defines multipoint constraints for rigid triangular plate.

Partitioning

ASET	Defines degrees-of-freedom in the analysis set (a-set).
ASET1	Defines degrees-of-freedom in the analysis set (a-set).
CSUPEXT	Assigns exterior points to a superelement.
GRID	Defines interior points for a superelement.
OMIT	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMIT1	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
ROTSE	Defines the modal reduction type and additional a-set grids for a rotor superelement.
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SESET	Defines interior grid points for a superelement.

Free Body Supports

CYSUP	Defines fictitious supports for cyclic symmetry analysis.
SUPORTi	Defines degrees-of-freedom for determinate reactions.

Component Mode Boundary Conditions

BNDFIX	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
BNDFIX1	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
BNDFREE	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
BNDFREE1	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
BSET	Defines the analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
BSET1	Defines the analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
CSET	Defines the analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
CSET1	Defines the analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
QSET	Defines the generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
QSET1	Defines the generalized degrees-of-freedom (q-set) to be used for component mode synthesis.

SEBSET	Defines the boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
SEBSET1	Defines the fixed boundary points for a superelement.
SECSET	Defines the boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SECSET1	Defines the boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SENQSET	Defines the number of internally generated scalar points for superelement dynamic reduction.
SEQSET	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SEQSET1	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SESUP	Defines the determinate reaction superelement degrees-of-freedom in a free-body analysis.

User Sets

DEFUSET	Defines the new names for degree-of-freedom sets.
SEUSET	Defines a degree-of-freedom set for a superelement.
SEUSET1	Defines a degree-of-freedom set for a superelement.
USET	Defines a degree-of-freedom set.
USET1	Defines a degrees-of-freedom set.

Loads

Static Loads

ACCEL	Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based on the tabular input defined on this Bulk Data entry.
ACCEL1	Defines static acceleration loads at individual grid points.
BOLTFOR	Defines preload force for a set of bolts.

BOLTLD	Combines sets of bolts defined by BOLTFOR bulk entries and optionally scales the corresponding bolt preload forces.
CLOAD	Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).
DEFORM	Defines the enforced axial deformation for one-dimensional elements for use in statics problems.
FORCE	Defines a static concentrated force at a grid point by specifying a vector.
FORCEi	Defines a concentrated load at grid point.
GRAV	Defines acceleration vectors for gravity or other acceleration loading.
LOAD	Defines a static load as a linear combination of load sets..
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LOADCYT	Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = “AXI” on the CYSYM entry.
MOMENT	Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.
MOMENTi	Defines moment at grid point.
PLOAD	Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.
PLOAD1	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.
PLOAD2	Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.

PLOAD4	Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.
PLOADE1	Defines a surface traction acting on an edge of a CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, or CPLSTS8 element.
PLOADX1	Defines a surface traction to be used with the CQUADX4, CQUADX8, CTRAX3, and CTRAX6 axisymmetric elements.
RFORCE	Defines a static loading condition due to an angular velocity and/or acceleration for an entire model.
RFORCE1	Defines a static loading condition due to an angular velocity and/or acceleration for a portion of a model.
RFORCE2	Defines a static loading condition due to an angular velocity and/or acceleration for SOL 101 maneuver load rotor dynamic analysis.
RVDOF	Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.
RVDOF1	Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.
RVEL	Specifies CBUSH, CBUSH1D, CDAMPi, CELASi, and CVISC elements for which to obtain static solutions for use in residual vector computations.
SELOAD	Maps loads from an external superelement to a specified load set for the residual structure.
SLOAD	Defines concentrated static loads on scalar or grid points.
SPCD	Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPEX	Defines a time independent temperature set using a .bun file for SOL 401 only.

TEMPPi	Defines a temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Dynamic Loads

ACSRCE	Defines the power versus frequency curve for a simple acoustic source.
DAREA	Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.
DELAY	Defines the time delay term τ in the equations of the dynamic loading function.
DLOAD	Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.
DPHASE	Defines the phase lead term θ in the equation of the dynamic loading function.
DTEMP	Defines a time dependent temperature set for SOL 401 only.
DTEMPEX	Defines a time dependent temperature set using a .bun file for SOL 401 only.
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LSEQ	Defines a sequence of static load sets.
NLRGAP	Defines a nonlinear transient radial (circular) gap.
NOLINi	Nonlinear transient load definition.
RLOADi	Frequency dependent excitation definition.
TLOADi	Time dependent excitation definition.
TABLEDi	Tabular functions for generating dynamic loads.

Heat Transfer Loads

CONV	Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
CONVM	Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
PCONV	Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
PCONVM	Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
QBDY1	Defines a uniform heat flux into CHBDYj elements.
QBDY2	Defines grid point heat flux into CHBDYj elements.
QBDY3	Defines a uniform heat flux load for a boundary surface.
QHBDY	Defines a uniform heat flux into a set of grid points.
QVECT	Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.
QVOL	Defines a rate of volumetric heat addition in a conduction element.
RADBC	Specifies a CHBDYi element face for the application of radiation boundary conditions.
RADBND	Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.
RADCAV	Identifies the characteristics of each radiant enclosure.
RADLST	Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.
RADMTX	Provides the $F_{ji} = A_j F_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.
RADSET	Specifies which radiation cavities are to be included for radiation enclosure analysis.

SLOAD	Defines concentrated static loads on scalar or grid points.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPBC	Defines the temperature boundary conditions for heat transfer analysis. This applies to steady-state and transient conditions.
TEMPD	Specifies default initial temperature at grid points.
VIEW	Defines radiation cavity and shadowing for radiation view factor calculations.
VIEW3D	Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

Solution Control

Buckling Analysis

EIGB	Defines data needed to perform buckling analysis.
EIGRL	Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Eigenvalue Analysis

EIGC	Defines data needed to perform complex eigenvalue analysis.
EIGP	Defines poles that are used in complex eigenvalue extraction by the Determinant method.
EIGR	Defines data needed to perform real eigenvalue analysis.
EIGRL	Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Cyclic Symmetry

CYSYM	Defines parameters for cyclic symmetry analysis.

Frequency Response

FREQ	Defines a set of frequencies to be used in the solution of frequency response problems.
FREQi	Defines a set of frequencies for problem solution.
TABDMP1	Defines modal damping as a tabular function of natural frequency.

Random Response

RANDPS	Defines load set power spectral density factors for use in random analysis.
RANDT1	Defines time lag constants for use in random analysis autocorrelation function calculation.
RCROSS	Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.
RCROSSC	Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis for composites.
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. This is referenced by the RANDPS entry.

Rotor Dynamics

ROTORD	Defines rotor dynamics solution options.
ROTPARM	Defines solution control parameters for complex eigenvalue rotor dynamic analysis. Valid for SOLs 107 and 110 only.

Transient Response

TIC	Defines values for the initial conditions of variables used in structural transient analysis.
TSTEP	Defines time step intervals at which a solution will be generated and output in transient analysis.
TSTEP1	Defines the time step intervals at which a solution will be generated and output in SOL 401.

TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, and 99.

Nonlinear Static Analysis

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
MATOVR	Globally turns off creep and/or plasticity effects in groups of elements. Applicable to SOL 401 only.
NLCNTL	Defines solution control parameters for SOL 401.
NLPARM	Defines a set of parameters for nonlinear static analysis iteration strategy.
NLPCI	Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106).

Element Birth/Death

EBDADD	Defines an element birth/death set as a union of element birth/death sets defined on EBDSET entries.
EBDSET	Defines element birth and death times for a set of elements.

Optimization (SOL 200 Only)

BNDGRID	Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).
DCONADD	Defines the design constraints for a subcase as a union of DCONSTR entries.
DCONSTR	Defines design constraints.
DDVAL	Defines real, discrete design variable values for discrete variable optimization.
DEQATN	Defines a design variable for design optimization.
DESVAR	Defines a design variable for design optimization.
DLINK	Relates one design variable to one or more other design variables.

DOPTPRM	Overrides default values of parameters used in design optimization.
DRESP1	Defines a set of structural responses that is used in the design either as constraints or as an objective.
DRESP2	Defines equation responses that are used in the design, either as constraints or as an objective.
DRESP3	Defines SOL 200 design responses to be evaluated in an external user-supplied program, that are used in the design either as constraints or as an objective.
DSCREEN	Defines screening data for constraint deletion.
DTABLE	Defines a table of real constants that are used in equations (see the DEQATN entry).
DTI,DFRFNC	Frequency function input for use in DRESP2 & DRESP3.
DVBSHAP	Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.
DVCREL1	Defines the relation between a connectivity property and design variables.
DVCREL2	Defines the relation between a connectivity property and design variables with a user-supplied equation.
DVGRID	Defines the relationship between design variables and grid point locations.
DVMREL1	Defines the relation between a material property and design variables.
DVMREL2	Defines the relation between a material property and design variables with a user-supplied equation.
DVPREL1	Defines the relation between an analysis model property and design variables.
DVPREL2	Defines the relation between an analysis model property and design variables with a user-supplied equation.
DVSHAP	Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.
MODTRAK	Specifies parameters for mode tracking in design optimization (SOL 200).

Aerodynamic Matrix Generation

MKAERO1	Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
MKAERO2	Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Aeroelastic Stability Analysis

DIVERG	Defines Mach numbers (m) for a divergence analysis in SOLs 144 and 200.
FLFACT	Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
FLUTTER	Defines data needed to perform flutter analysis.

Aeroelastic Response Analysis

AEDW	Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.
AEFORCE	Defines a vector of absolute forces (it will **not** be scaled by dynamic pressure) associated with a particular control vector.
AEPARM	Defines a general aerodynamic trim variable degree of freedom (aerodynamic extra point).
AEPRESS	Defines a vector of pressure/unit dynamic pressure associated with a particular control vector.
GUST	Defines a stationary vertical gust for use in aeroelastic response analysis.
TABRNDG	Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.
TRIM	Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.

Aerodynamic Parameters

AERO	Gives basic aerodynamic parameters for unsteady aerodynamics.

AEROS	Defines basic parameters for static aeroelasticity.

Miscellaneous

Comments

$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Delete

/	Removes entries on restart.

Parameters

PARAM	Specifies values for parameters used in solution sequences or user-written DMAP programs.

Groups and Sets

GROUP	Defines a group of grid points, elements, or physical properties.
SET1	Defines a list of structural grid points, elements, or DRESPi (i=1,2,3) responses. SET1 entries are referenced by SPLINEi entries, PANEL entries, XYOUTPUT, and the P2RSET option on DOPTPRM entries.
SET2	Defines a list of structural grid points in terms of aerodynamic macro elements. SET2 entries are referenced by SPLINEi entries.
SET3	Defines a list of structural grid points, elements, or physical properties. SET3 entries are referenced by ACMODL entries and PANEL entries.

Direct Matrix Input

CONM1	Defines a 6x6 mass matrix at a geometric grid point.
DMI	Defines matrix data blocks.
DMIG	Defines direct input matrices related to grid, extra, and/or scalar points.

DMIG,UACCEL	Defines rigid body accelerations in the basic coordinate system.
TF	Defines a dynamic transfer function.

Direct Matrix Input for Aeroelasticity

DMIJ	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries.
DMIJI	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2.
DMIK	Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points.

Tabular Input

DTI	Defines table data blocks.
DTI,DFRFNC	Frequency function input for use in DRESP2 & DRESP3. SOL 200 only.
DTI,ESTDATA	Provides override data for time and space estimation for superelement processing operations.
DTI,INDTA	Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.
DTI,SELOAD	Usage overrides automatic generation of DTI,SELOAD definitions created through the use of the EXTSEOUT case control command.
DTI,SETREE	Defines a superelement tree that determines the superelement processing order.
DTI,SPECSEL	Correlates spectra lines specified on TABLED1 entries with damping values.
DTI,SPSEL	Correlates output requests with frequency and damping ranges.
DTI,UNITS	Specifies the system of units.
TABDMP1	Defines modal damping as a tabular function of natural frequency.
TABLEDi	Tabular functions for generating dynamic loads.

TABLEMi	Tabular functions for generating temperature-dependent material properties.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Output Control

BOUTPUT	Defines slave nodes at which output is requested.
CBARAO	Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.
ECHOOFF	Marks the point or points in the input file to deactivate printed echo of the Bulk Data.
ECHOON	Marks the point or points in the input file to activate printed echo of the Bulk Data.
PLOTEL	Defines a one-dimensional dummy element for use in plotting.
TSTEP	Specifies time step intervals for data recovery in transient response.
TSTEPNL	Specifies time step intervals for data recovery in nonlinear transient response.

Solution Control

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
NXSTRAT	Defines parameters for solution control and strategy in advanced nonlinear structural analysis.

End of Input

ENDDATA	Designates the end of the Bulk Data Section.

Include File

INCLUDE	Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

9.5 Element Utilization Summary

Use these tables to identify which combinations of element, property, and material bulk entries you can use to obtain a particular material behavior in a given solution.

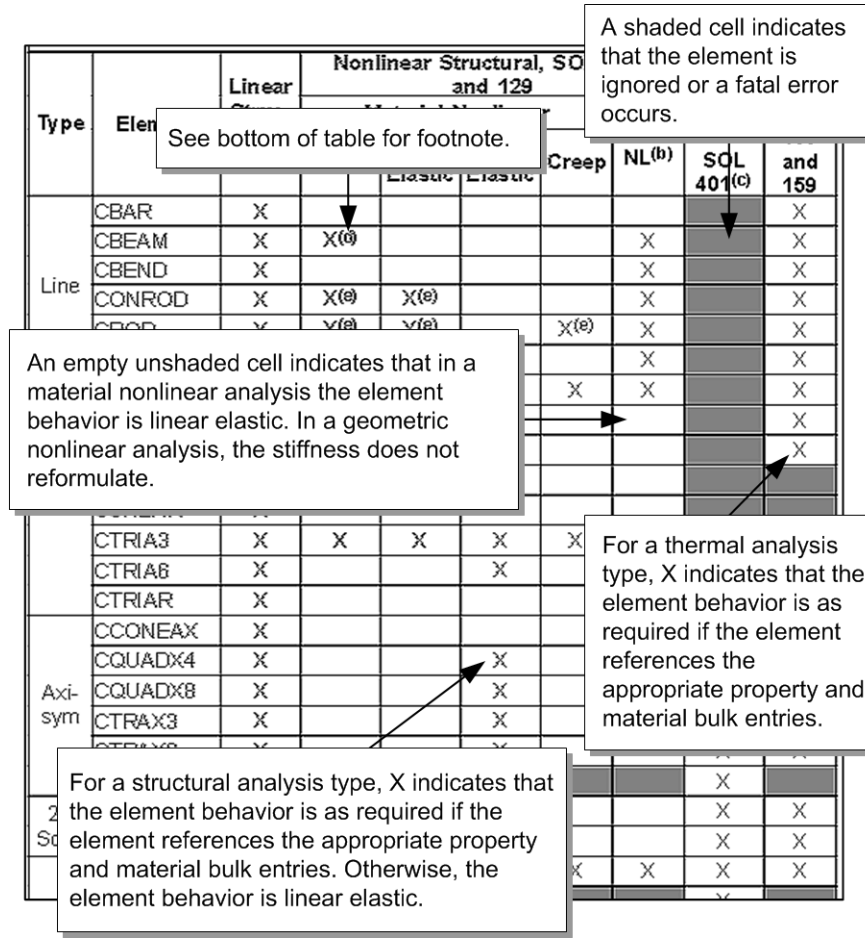
1. The Element Applicability Table shows which element bulk entries are valid in a given solution.

   For information on how to interpret this table, see Key to Element Applicability Table.

2. The Element-Property-Material Cross-Reference Tables show which combinations of material bulk entries and property bulk entries are valid for a given element bulk entry.

For example, suppose you want to perform a SOL 106 plasticity analysis on a thin-walled structure. The *Element Applicability for Solutions 101–401 Table* indicates that the CQUAD4 and CTRIA3 shell elements can be used to model plasticity in SOL 106. Because you know that MATS1 bulk entries are required to model plasticity, the table *Shell Elements: Supplemental Materials by Primary Material, Property, and Element* indicates that the CQUAD4 and CTRIA3 elements must reference PSHELL bulk entries that in turn reference MAT1, MAT2, or MAT8 bulk entries.

Key to Element Applicability Table



Element Applicability Table

Table 9-1. Element Applicability for Solutions 101–401

Type	Element	Linear Structural SOLs[a]	Nonlinear Structural, SOLs 106 and 129					Multi-step Structural SOL 401[c]	Heat Trans SOLs 153 and 159
			Material Nonlinear				Geom NL[b]		
			Plastic	NL Elastic	Hyper-Elastic	Creep			
Line	CBAR	X							X
	CBEAM	X	X[d]				X		X
	CBEND	X					X		X
	CONROD	X	X[e]	X[e]			X		X
	CROD	X	X[e]	X[e]		X[e]	X		X
	CTUBE	X	X[e]	X[e]			X		X
Shell	CQUAD4	X	X	X	X	X	X		X
	CQUAD8	X			X				X
	CQUADR	X							X
	CRAC2D	X							
	CSHEAR	X							
	CTRIA3	X	X	X	X	X	X		X
	CTRIA6	X			X				X
	CTRIAR	X							X
Axi-sym	CQUADX4	X			X			X	X
	CQUADX8	X			X			X	X
	CTRAX3	X			X			X	X
	CTRAX6	X			X			X	X
	CCHOCKi							X	
2D Solid	CPLSTNi	X[f]						X	X
	CPLSTSi	X[f]						X	X
3D Solid	CHEXA	X	X	X	X	X	X	X	X
	CHEXCZ							X	
	CPENTA	X	X	X	X	X	X	X	X
	CPENTCZ							X	
	CPYRAM	X	X	X		X	X	X	X
	CRAC3D	X							
	CTETRA	X	X	X	X	X	X	X	X

Table 9-1. Element Applicability for Solutions 101–401

Type	Element	Linear Structural SOLs[a]	Nonlinear Structural, SOLs 106 and 129					Multi-step Structural SOL 401[c]	Heat Trans SOLs 153 and 159
			Material Nonlinear				Geom NL[b]		
			Plastic	NL Elastic	Hyper-Elastic	Creep			

[a] Includes SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 187, and 200.
[b] Small strain only. For large strain, refer to the hyperelasticity column.
[c] SOL 401 supports linear, plasticity, creep, and geometric nonlinear structural analyses.
[d] Elastic-perfectly plastic material behavior used to model the ends of beams as plastic hinges.
[e] Nonlinear capability for axial deflections only.
[f] Not supported for use in SOLs 114, 115, 116, 118, 144, 145, and 146.

Element-Property-Material Cross-Reference Tables

Table 9-2. Element-Property-Material Cross-Reference Tables

Element Type	Elements	Table
Line	CBAR, CBEAM, CBEND, CONROD, CROD, CTUBE	9-3
Shell	CQUAD4, CQUAD8, CQUADR, CRAC2D, CSHEAR, CTRIA3, CTRIA6, CTRIAR	9-4
Axisymmetric	CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8, CQUADX4, CQUADX8, CTRAX3, CTRAX6	9-5
2D Solid	CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8	9-6
3D Solid	CHEXA, CHEXCZ, CPENTA, CPENTCZ, CPYRAM, CRAC3D, CTETRA	9-7
Special Purpose	CAABSF, CAERO1, CAERO2, CAERO3, CAERO4, CAERO5, CBEAR, CBUSH, CBUSH1D, CDAMP1, CDAMP2, CDAMP3, CDAMP4, CDAMP5, CDUM1, CDUM2, CDUM3, CDUM4, CDUM5, CDUM6, CDUM7, CDUM8, CDUM9, CELAS1, CELAS2, CELAS3, CELAS4, CFAST, CGAP, CHACAB, CHACBR, CHBDYE, CHBDYG, CHBDYP, CMASS1, CMASS2, CMASS3, CMASS4, CONM1, CONM2, CONV, CONVM, CVISC, CWELD, RBAR, RBE1, RBE2, RBE3	9-8

Table 9-3. Line Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CBAR	PBAR	MAT1					X								
		MAT4								X					
		MAT5									X				
	PBARL	MAT1					X								
		MAT4								X					
		MAT5									X				

9
BULK

Table 9-3. Line Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CBEAM	PBEAM	MAT1				X(1)	X								
		MAT4								X					
		MAT5									X				
	PBEAML	MAT1					X								
		MAT4								X					
		MAT5									X				
	PBCOMP	MAT1				X(1)	X								
		MAT4								X					
		MAT5									X				
CBEND	PBEND	MAT1					X								
		MAT4								X					
		MAT5									X				
CONROD	N/A	MAT1				X(2)	X								
		MAT4								X					
		MAT5									X				
CROD	PROD	MAT1				X(2)	X								X(2)
		MAT4								X					
		MAT5									X				
CTUBE	PTUBE	MAT1				X(2)	X								
		MAT4								X					
		MAT5									X				

(1) For material nonlinear analysis, elastic-perfectly plastic material behavior only. Strain hardening is not supported. The ends of the element are modeled as plastic hinges.
(2) Nonlinear capability for axial deflections only.

Table 9-4. Shell Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CQUAD4 CTRIA3	PSHELL	MAT1				X	X								X
		MAT2				X		X							X
		MAT4								X					
		MAT5									X				
		MAT8				X						X			X
	PCOMP	MAT1					X								
		MAT2						X							
		MAT8										X			
	PCOMPG	MAT1					X								
		MAT2						X							
		MAT8										X			
	PLPLANE	MATHP													
	N/A	MFLUID													
CQUAD8 CTRIA6	PSHELL	MAT1					X								
		MAT2						X							
		MAT4								X					
		MAT5									X				
		MAT8										X			
	PCOMP	MAT1					X								
		MAT2						X							
		MAT8										X			
	PCOMPG	MAT1					X								
		MAT2						X							
		MAT8										X			
	PLPLANE	MATHP													

Table 9-4. Shell Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CQUADR CTRIAR	PSHELL	MAT1					X								
		MAT2						X							
		MAT4								X					
		MAT5									X				
		MAT8										X			
	PCOMP	MAT1					X								
		MAT2						X							
		MAT8										X			
	PCOMPG	MAT1					X								
		MAT2						X							
		MAT8										X			
CRAC2D	PRAC2D	MAT1					X								
		MAT2						X							
		MAT8										X			
CSHEAR	PSHEAR	MAT1													

Table 9-5. Axisymmetric Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CCHOCK3 CCHOCK4 CCHOCK6 CCHOCK8	PCHOCK	MAT1	X			X	X								X
		MAT11	X			X								X	X

Table 9-5. Axisymmetric Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CQUADX4 CQUADX8 CTRAX3 CTRAX6	PSOLID	MAT1	X				X								
		MAT3							X						
		MAT4								X					
		MAT5									X				
		MAT9	X										X		
		MAT11	X											X	
	PLSOLID	MATHP													

Table 9-6. 2D Solid Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CPLSTN3 CPLSTN4 CPLSTN6 CPLSTN8	PPLANE	MAT1	X				X								
		MAT3							X						
		MAT4								X					
		MAT5									X				
		MAT11	X											X	
	PGPLSN	MAT1	X				X								
		MAT11	X											X	
CPLSTS3 CPLSTS4 CPLSTS6 CPLSTS8	PPLANE	MAT1	X				X								
		MAT3							X						
		MAT4								X					
		MAT5									X				
		MAT11	X											X	

Table 9-7. 3D Solid Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CHEXA CPENTA	PSOLID	MAT1	X			X	X								X
		MAT4								X					
		MAT5									X				
		MAT9	X			X							X		X
		MAT10													
		MAT11	X			X								X	X
		MATPOR													
	PLSOLID	MATHP													
	PCOMPS	MAT1	X	X	X		X								
		MAT9	X		X								X		
		MAT11	X	X	X									X	

9
BULK

Table 9-7. 3D Solid Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CHEXCZ CPENTCZ	PSOLCZ	MAT1					X								
		MAT11												X	
		MATCZ													
CPYRAM	PSOLID	MAT1	X			X	X								X
		MAT4								X					
		MAT5									X				
		MAT9	X			X							X		X
		MAT10													
		MAT11	X			X								X	X
		MATPOR													
CRAC3D	PRAC3D	MAT1					X								
		MAT9											X		
CTETRA	PSOLID	MAT1	X			X	X								X
		MAT4								X					
		MAT5									X				
		MAT9	X			X							X		X
		MAT10													
		MAT11	X			X								X	X
		MATPOR													
	PLSOLID	MATHP													

Table 9-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CAABSF	PAABSF	N/A													
CAEROi (1)	PAEROi (1)	N/A													
CBEAR	PBEAR	N/A													
CBUSH	PBUSH (2)	N/A													
CBUSH1D	PBUSH1D	N/A													
CDAMP1	PDAMP	N/A													
CDAMP2	N/A	N/A													
CDAMP3	PDAMP	N/A													
CDAMP4	N/A	N/A													
CDAMP5	PDAMP5	MAT4								X					
		MAT5									X				
CDUMi (3)	PDUMi (3)	(4)													
CELAS1	PELAS	N/A													
CELAS2	N/A	N/A													
CELAS3	PELAS	N/A													
CELAS4	N/A	N/A													
CFAST	PFAST	N/A													
CGAP	PGAP	N/A													
CHACAB	PACABS	N/A													
CHACBR	PACBAR	N/A													
CHBDYE	N/A	N/A													
CHBDYG	N/A	N/A													
CHBDYP	PHBDY	N/A													
CMASS1	PMASS	N/A													
CMASS2	N/A	N/A													
CMASS3	PMASS	N/A													
CMASS4	N/A	N/A													
CONMi (5)	N/A	N/A													
CONV	PCONV	MAT4								X					

Table 9-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element

Element	Property	Primary Material	Supplemental Material												
			MATi												CREEP
			CRP	DMG	FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CONVM	PCONVM	MAT4								X					
CVISC	PVISC	N/A													
CWELD	PWELD	MAT1													
RBAR	N/A	N/A													
RBEi (6)	N/A	N/A													

(1) i = 1 through 5
(2) Use a PBUSHT entry with the same PID as a PBUSH entry to define frequency dependent or stress dependent properties.
(3) i = 1 through 9
(4) Per user definition.
(5) i = 1 and 2
(6) i = 1, 2, and 3

$

Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

FORMAT:

$ followed by any characters out to column 80.

EXAMPLE:

```
$ TEST FIXTURE-THIRD MODE
```

REMARKS:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

/

Delete

Removes entries on restart.

FORMAT:

1	2	3	4	5	6	7	8	9	10
/	K1	K2							

EXAMPLE:

/	4								

Field	Contents
K1	Sorted sequence number of first entry in sequence to be removed. (Integer > 0)
K2	Sorted sequence number of last entry in sequence to be removed. (Integer > 0; Default = K1)

REMARKS:

1. This entry causes Bulk Data entries having sort sequence numbers K1 through K2 to be removed from the Bulk Data. The sort sequence numbers appear in the output of the previous run under the sorted Bulk Data echo.
2. If K2 is blank, only entry K1 is removed from the Bulk Data.
3. If the current execution is not a restart, the entries marked for deletion are ignored.
4. This entry can only be used in the rigid formats and the structured solution sequences. See “Executive Control Statements”, under the SOL statement.
5. K2 may be specified as larger than the actual sequence number of the last entry. This is convenient when deleting entries to the end of the Bulk Data Section.

/

Chapter 10: Bulk Data Entries A—B

ACCEL

Define Static Acceleration Loads

Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based on the tabular input defined on this Bulk Data entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ACCEL	SID	CID	N1	N2	N3	DIR			
	LOC1	VAL1	LOC2	VAL2	Continues in groups of 2				

EXAMPLE:

ACCEL	100	2	0.0	1.0	0.0	X			
	0.0	1.0	33.1	2.0					

FIELDS:

Field	Contents
SID	Load set identification number. (Integer>0)
CID	Coordinate system identification number. (Integer>0: Default=0)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
DIR	Component direction of acceleration variation. (Character; one of X, Y or Z)
LOCi	Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)
VALi	The load scale factor associated with location LOCi. (Real)

REMARKS:

1. For all grids of the model, the acceleration vector is defined by **a** = *VAL* · **N** where **N** is the vector with components N1, N2, and N3. The magnitude of **a** is equal to *VAL* times the magnitude of **N**. The scale factor *VAL* for each grid is found by linearly interpolating the DIR coordinate of the grid between table values *LOCi/VALi*. If the GRID point coordinate in coordinate system CID is outside the range of the table, *VAL* is determined from the closer of *VAL1* or *VALn*. (See the following figure.)

2. This type of acceleration load may be combined with other loads such as FORCE, MOMENT, GRAV and ACCEL1 loads only by specification on a LOAD bulk data entry. That is, the SID on an ACCEL entry may not be the same as that on any other load entry.

3. This acceleration load does not include effects due to mass on scalar points.

4. A blank CID entry or a CID of zero references the basic coordinate system.

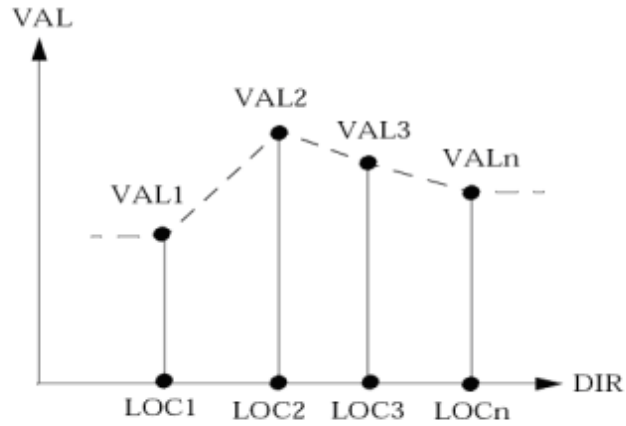
5. The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.

6. A minimum of two pairs of {LOCi, VALi} data must be specified.

7. When applying a load with a GRAV, ACCEL, or ACCEL1 entry to axisymmetric elements, any component in the radial direction is treated as a radial acceleration load.

8. The software treats cylindrical and spherical coordinate systems selected in the CID field on the GRAV, ACCEL, or ACCEL1 entries as a Cartesian coordinate system. For example, if you define an acceleration vector as 1.0, 0.0, 0.0 and a cylindrical or spherical coordinate system is selected in the CID field, the software will not vary the load vector radially at the grid points.

9. The ACCEL bulk entry is not supported for non-partitioned superelements.



Definition of Load Scale Factor vs Location

ACCEL1

Define Static Acceleration Loads

Defines static acceleration loads at individual grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ACCEL1	SID	CID	A	N1	N2	N3			
	GRIDID1	GRIDID2	etc						

EXAMPLE:

ACCEL1	100	2	100.0	0.0	1.0	0.0			
	1	2	3	4	5	6	7	8	
	9	10	THRU	100	BY	5	200	250	

FIELDS:

Field	Contents
SID	Load set identification number. (Integer>0)
CID	Coordinate system identification number. (Integer>0: Default=0)
A	Acceleration vector scale factor. (Real)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
GRIDIDi	List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer>0)

REMARKS:

1. The acceleration vector is defined by **a** = *A* · **N** where **N** is the vector with components N1, N2, and N3. The magnitude of **a** is equal to *A* times the magnitude of **N**.

2. The SID on an ACCEL1 entry can be the same as the SID defined on other ACCEL1 entries. Although, the SID on an ACCEL1 entry cannot be the same as the SID defined on other types of load entries. For example, you cannot use the same SID on ACCEL1 and FORCE entries. The LOAD bulk entry is required to combine these loads.

3. This acceleration load does not include effects due to mass on scalar points.

4. A blank CID entry or a CID of zero references the basic coordinate system.

5. The ACCEL1 card must contain at least one GRIDID.

6. The ACCEL1 bulk entry is not supported for non-partitioned superelements.

7. When applying a load with a GRAV, ACCEL, or ACCEL1 entry to axisymmetric elements, any component in the radial direction is treated as a radial acceleration load.

8. The software treats cylindrical and spherical coordinate systems selected in the CID field on the GRAV, ACCEL, or ACCEL1 entries as a Cartesian coordinate system. For example, if you define an acceleration vector as 1.0, 0.0, 0.0 and a cylindrical or spherical coordinate system is selected in the CID field, the software will not vary the load vector radially at the grid points.

ACMODL

Fluid-Structure Interface Modeling Parameters

Defines modeling parameters for the interface between the fluid and the structure.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ACMODL		INFOR	FSET	SSET	NORMAL		OVLPANG	SRCHUNIT	
	INTOL	AREAOP			CTYPE				

EXAMPLE:

ACMODL					0.25		15.0		
					WEAK				

FIELDS:

Field	Contents
INFOR	Defines the meaning of the SID entered on the FSET and SSET fields. (Character; ELEMENTS, PID or SET3) See Remark 3.
FSET	Selects the ID of a SET1 or SET3 entry to define the fluid elements for the interface. (Integer or blank) See Remark 3. If the ID is entered, the corresponding fluid elements are considered. If a negative sign is included in front of the ID, the corresponding fluid elements are excluded. If blank, all fluid elements are considered.

Field	Contents
SSET	Selects the ID of a SET1 or SET3 entry to define the structural elements for the interface. (Integer or blank) See Remark 3. If the ID is entered, the corresponding structural elements are considered. If a negative sign is included in front of the ID, the corresponding structural elements are excluded. If blank, all structural elements are considered.
NORMAL	Outward normal search distance to detect fluid-structure interface. (Real > 0.0; Default = 0.5) See Remark 4. If SRCHUNIT = REL, then NORMAL is a ratio of the height of the fluid box in the outward normal direction to the fluid surface to the maximum edge length of the fluid free face. If SRCHUNIT = ABS, then NORMAL is the outward search distance in the model/absolute units.
OVLPANG	Angular tolerance in degrees used to decide whether a fluid free face and a structural face can be considered as overlapping. If the angle between the normals of the fluid and structural faces exceeds this value, they cannot be coupled. (Real > 0.0; Default = 60.0)
SRCHUNIT	Search units. (Character; Default = REL) See Remark 4. If SRCHUNIT = ABS, then the model units are absolute. If SRCHUNIT = REL, then the relative model units are based on element size.
INTOL	Inward normal search distance to detect fluid-structure interface. (Real > 0.0; Default = 0.20) See Remark 4. If SRCHUNIT = REL, then INTOL is a ratio of the height of the fluid box in the inward normal direction to the fluid surface to the maximum edge length of the fluid free face. If SRCHUNIT = ABS, then INTOL is the inward search distance in the model/absolute units.

Field	Contents
AREAOP	Alternative fluid-structure coupling method selection. (Integer; Default=0) See Remark 5. 0 = The recommended method is used. 1 = The RBE3 method is used.
CTYPE	Fluid-structure coupling type (only supported by new acoustics method introduced in NX Nastran 11). (Character; Default = STRONG) See Remark 6. If CTYPE = STRONG, two-way coupling is turned on. If CTYPE = WEAK, one-way coupling is turned on. Here, the effect of the fluid on the structure is assumed to be negligible.

REMARKS:

1. Only one ACMODL entry is allowed. If this entry is not present, defaults will be used.

2. The ACMODL entry is supported in solutions 103, 107-112, and 200.

3. If you enter the IDs of SET1 entries on the FSET and SSET fields, you must enter either ELEMENTS or PID on the INFOR field to define how the software should interpret the items selected by the SET1 entries.

   If you enter the IDs of SET3 entries on the FSET and SSET fields, you must enter SET3 on the INFOR field. The TYPE field on the SET3 entries defines how the software should interpret the items selected by the SET3 entries.

   The SET1 entry can list element IDs or physical property IDs. The SET3 entry can list element IDs, physical property IDs, or grid IDs. If property IDs or grid IDs are selected, the software will determine the associated elements. The PSOLID property ID is supported for selecting fluid elements. The PSOLID, PSHELL, PCOMP, and PCOMPG property IDs are supported for selecting structural elements. Solid composite elements using the PCOMPS property cannot be coupled to the fluid.

4. The fields NORMAL and INTOL are interpreted as follows:

   If SRCHUNIT = REL, then NORMAL is a ratio of the outward height of the bounding box to the maximum edge length of the fluid free face. That is, if L is

the largest edge of the fluid free face, the height H of the bounding box used to search for structural faces will be NORMAL * L. INTOL applies similarly, but the inward direction.

If SRCHUNIT = ABS, then NORMAL defines the outward height of the fluid bounding box in the model/absolute units. INTOL defines the inward height of the fluid bounding box in the model/absolute units.

If SRCHUNIT = ABS and NORMAL or INTOL are blank, then SRCHUNIT will be reset to REL and their corresponding default value is used.

5. AREAOP = 0 selects the default, recommended NX Nastran coupling option.

   AREAOP = 1 selects an alternate option, which applies an area correction and removes parallel disconnected faces from the coupling.

6. The two-way coupled formulation (that is CTYPE = STRONG) of NX Nastran leads to the following:

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & A \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

   where *s* indicates structure

   and *f* indicates fluid

   For a one-way (fully decoupled) solution (that is CTYPE = WEAK), the *A* term is ignored and the formulation becomes:

$$\begin{bmatrix} M_s & 0 \\ -A^T & M_f \end{bmatrix} \begin{Bmatrix} \ddot{u}_s \\ \ddot{p} \end{Bmatrix} + \begin{bmatrix} B_s & 0 \\ 0 & B_f \end{bmatrix} \begin{Bmatrix} \dot{u}_s \\ \dot{p} \end{Bmatrix} + \begin{bmatrix} K_s & 0 \\ 0 & K_f \end{bmatrix} \begin{Bmatrix} u_s \\ p \end{Bmatrix} = \begin{Bmatrix} P_s \\ P_f \end{Bmatrix}$$

ACSRCE

Acoustic Source Specification

Defines source strength as a function of frequency for a simple acoustic source of the form:

$$\text{Source Strength} = Q(f) = A\left[\frac{1}{2\pi f}\sqrt{\frac{8\pi CP(f)}{\rho}}\right]e^{i(\theta + 2\pi f\tau)}$$

$$C^2 = B/\rho$$

FORMAT:

1	2	3	4	5	6	7	8	9	10
ACSRCE	SID	EXCITEID	DELAY	DPHASE	TP	RHO	B		

EXAMPLES:

ACSRCE	103	11	0.001		12	1.0	15.0		

ACSRCE	103	11	104	45.0	12	1.0	15.0		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of a DAREA or SLOAD entry that lists each degree of freedom to apply the excitation and the corresponding scale factor, *A*, for the excitation. (Integer > 0)
DELAY	Time delay, τ. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 6) If real entry, value of τ for all degrees of freedom in EXCITEID entry.

Field	Contents
	If integer entry, identification number of a DELAY entry that contains values of τ for all degrees of freedom in EXCITEID entry. See Remark 7.
DPHASE	Phase angle, θ, in degrees. See Remark 8. (Real or Integer > 0 or blank; for default behavior, see Remark 6) If real entry, value of θ for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a DPHASE entry that contains values of θ for all degrees of freedom in EXCITEID entry. See Remark 7.
TP	Power as a function of frequency, *P(f)*. (Real > 0.0 or Integer > 0) If real entry, value of *P(f)* used over all frequencies for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a TABLEDi entry that defines *P(f)* for all degrees of freedom in EXCITEID entry.
RHO	Density of the fluid. (Real > 0.0)
B	Bulk modulus of the fluid. (Real > 0.0)

REMARKS:

1. Acoustic sources must be selected with DLOAD = SID in the case control section.
2. SID must be unique for all ACSRCE, RLOADi, TLOADi, and SELOAD entries.
3. If ACSRCE and RLOADi entries are combined with a DLOAD bulk entry, the identification numbers of the TABLEDi selected with the TP field on the ACSRCE entries must be different from the identification numbers of the TABLEDi selected with the TC and TD fields on RLOAD1 entries, and the TB and TP fields on RLOAD2 entries.
4. For additional remarks, see the RLOAD1 entry description.
5. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.

6. If any of DELAY and DPHASE fields are blank or zero (either integer zero or real zero), the corresponding value for τ and θ used by the software is real zero.

7. For degrees of freedom in the EXCITEID entry that are not specified on the DELAY entry, the software uses real zero as the value for τ. For degrees of freedom in the EXCITEID entry that are not specified on the DPHASE entry, the software uses real zero as the value for θ.

8. The software converts the phase angle, θ, to radians.

ADUMi

Dummy Element Attributes

Defines attributes of the dummy elements (1 ≤ i ≤ 9).

FORMAT:

1	2	3	4	5	6	7	8	9	10
ADUMi	NG	NC	NP	ND	ELNM				

EXAMPLE:

ADUM2	8	2	1	3	CTRIM6				

FIELDS:

Field	Contents
NG	Number of grid points connected by a DUMi dummy element. (Integer > 0)
NC	Number of additional fields (Ai) on the CDUMi connection entry. (Integer ≥ 0)
NP	Number of additional fields (Ai) on the PDUMi property entry. (24 ≥ Integer ≥ 0)
ND	Number of displacement components at each grid point used in the generation of the differential stiffness matrix. Zero implies no differential stiffness. (Integer 3 or 6)
ELNM	The name of the element connection and property entry. In the example above, the connection entry is named “CTRIM6” and the property entry is named “PTRIM6”.

AECOMP

Component for an Integrated Load Monitor Point

Defines a component for use in aeroelastic monitor point definition.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AECOMP	NAME	LISTTYPE	LISTID1	LISTID2	LISTID3	LISTID4	LISTID5	LISTID6	
	LISTID7	-etc.-							

EXAMPLE:

AECOMP	WING	AELIST	1001	1002					

FIELDS:

Field	Contents
NAME	A character string of up to eight characters identifying the component. (Character)
LISTTYPE	One of CAERO or AELIST for aerodynamic components and SET1 for structural components. Aerodynamic components are defined on the aerodynamic ks-set mesh while the structural components are defined on the g-set mesh. See Remarks 2 and 3.
LISTIDi	The identification number of either SET1, AELIST or CAEROi entries that define the set of grid points that comprise the component. See Remarks 2 and 3.

REMARKS:

1. The identification name must be unique among all AECOMP and AECOMPL entries.

2. If the component is defined on the structure, the LISTIDs must refer to SET1 entry(ies) that define the list of associated GRID points. For the AELIST or CAERO option, the LISTID's must refer to AELIST or CAERO i entries, respectively. Note that for DLM models (CAERO1/2), the set of points defined by the AELIST are the box identification numbers. For example, if the control surface's grids are desired, the same AELIST used for the AESURF can be referred to here. An AECOMP component must be defined as either an aerodynamic mesh component or a structural component. The two mesh classes cannot be combined into a single component.

3. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

AECOMPL

Component for an Integrated Load Monitor Point

Defines a component for use in aeroelastic monitor point definition as a union of other components.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AECOMPL	NAME	LABEL1	LABEL2	LABEL3	LABEL4	LABEL5	LABEL6	LABEL7	
	LABEL8	-etc.-							

EXAMPLE:

AECOMPL	HORIZ	STAB	ELEV	BALANCE					

FIELDS:

Field	Contents
NAME	A character string of up to eight characters identifying the component. (Character)
LABELi	A string of 8 characters referring to the names of other components defined by either AECOMP or other AECOMPL entries. (Character. See Bulk Data Syntax Rules.)

REMARKS:

1. The identification name must be unique among all AECOMP and AECOMPL entries.
2. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

AEDW

Parametric Normal Wash Loading for Aerodynamics

Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEDW	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

EXAMPLE:

AEDW	0.90	SYMM	ASYMM	101	ALP1				

FIELDS:

Field	Contents
MACH	The Mach number for this force. See Remark 2 (Real ≥ 0.0, ≠ 1.0).
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
DMIJ	The name of a DMI or DMIJ entry that defines the downwash.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element downwashes.

REMARKS:

1. AEDW, AEFORCE and AEPRESS are associated with the current AECONFIG using either Case Control (if in the main Bulk Data Section) or using the BEGIN AECONFIG=<config> if in a partition of the Bulk Data.

2. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero downwash (j-set) input is needed on the interference body elements.

3. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

AEFACT

Aerodynamic Lists

Defines real numbers for aeroelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEFACT	SID	D1	D2	D3	D4	D5	D6	D7	
	D8	D9	-etc.-						

EXAMPLE:

AEFACT	97	.3	.7	1.0					

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer > 0)
Di	Number. (Real)

REMARKS:

1. AEFACT entries must be selected by a CAEROi or PAEROi entry.
2. Embedded blank fields are not allowed.
3. To specify division points, there must be one more division point than the number of divisions.
4. When referenced by the CAERO3 entry, AEFACT defines the aerodynamic grid points. The ID number of the first point defined by each AEFACT entry is

the value of the CAERO3 ID that selected the AEFACT entry. The ID of each following point defined on the AEFACT is incremented by 1.

AEFORCE

Parametric Force for Aerodynamics

Defines a vector of absolute forces (it will **not** be scaled by dynamic pressure) associated with a particular control vector. This force vector may be defined on either the aerodynamic mesh (ks-set) or the structural mesh (g-set). The force vector will be used in nonlinear static aeroelastic trim.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEFORCE	MACH	SYMXZ	SYMXY	UXID	MESH	FORCE	DMIK		

EXAMPLE:

AEFORCE	0.90	SYMM	ASYMM	101	AERO		BETA		

FIELDS:

Field	Contents
MACH	The Mach number for this force. See Remark 2 (Real ≥ 0.0, ≠ 1.0).
SYMXZ, SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
MESH	One of AERO or STRUCT that declares whether the force vector is defined on the aerodynamic ks-set mesh or the structural g-set mesh. See Remark 3.
FORCE	The ID of a FORCE/MOMENT set that defines the vector. See Remark 3 (Integer > 0 if MESH=STRUCT).
DMIK	The name of a DMIK entry that defines the aerodynamic force vector. See Remark 3 (Character; required if MESH=AERO).

REMARKS:

1. AEDW, AEFORCE and AEPRESS are associated with the current AECONFIG.

2. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

3. If the vector is defined on the structure, normal FORCEi and MOMENTi data are used. They will be subject to normal reduction to the solution set. For the AERO mesh option, the DMIK Bulk Data are used. Any forces associated with the permanently SPC'd degrees-of-freedom (which are dependent on the type of aerodynamic model being used) will be ignored.

AELINK

Links Aeroelastic Variables

Defines relationships between or among AESTAT and AESURF entries, such that:

$$u^D + \sum_{i=1}^{n} \mathrm{Ci}\ u_i^I = 0.0$$

u^D = dependent variable

and

u^I_i = independent variable

FORMAT:

1	2	3	4	5	6	7	8	9	10
AELINK	ID	LABLD	LABL1	C1	LABL2	C2	LABL3	C3	
	LABL4	C4	-etc.-						

EXAMPLE:

AELINK	10	INBDA	OTBDA	-2.0					

FIELDS:

Field	Contents
ID	Trim set identification number. (Integer > 0)
LABLD	Character string to identify the dependent aerodynamic variable. (Character)
LABLi	Character string to identify the i-th independent aerodynamic variable. (Character)

Field	Contents
Ci	Linking coefficient for the i-th variable. (Real)

REMARKS:

1. The AELINK entry (or entries) is selected by the TRIM=ID in Case Control.
2. This entry constrains the dependent variable to be a linear combination of the independent variables.
3. LABLD data must be unique for a given ID (i.e., the variable cannot be constrained twice).
4. LABLD and LABLi data refer to AESURF or AESTAT Bulk Data entries.

AELIST

Aerodynamic Elements List

Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AELIST	SID	E1	E2	E3	E4	E5	E6	E7	
	E8	-etc.-							

EXAMPLE:

AELIST	75	1001	THRU	1075	1101	THRU	1109	1201	
	1202								

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
Ei	List of aerodynamic boxes generated by CAERO1 entries to define a surface. (Integer > 0 or “THRU”)

REMARKS:

1. These entries are referenced by the AESURF entry.
2. When the “THRU” option is used, all intermediate grid points must exist. The word “THRU” may not appear in field 3 or 9 (2 or 9 for continuations).
3. Intervening blank fields are not allowed.

AEPARM

General Controller for Use in Trim

Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point). The forces associated with this controller will be derived from AEDW, AEFORCE and AEPRESS input data.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEPARM	ID	LABEL	UNITS						

EXAMPLE:

AEPARM	5	THRUST	LBS						

FIELDS:

Field	Contents
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. See Remark 1 (Character).
UNITS	Label used to describe the units of the controller values. (Character)

REMARKS:

1. Controller LABELs that comprise the unique set relative to all the AESURF, AESTAT and AEPARM entries will define the set of trim variable degrees-of-freedom for the aeroelastic model.

2. Unit labels are optional and are only used to label outputs. No units will be associated with the controller if left blank.

AEPRESS

Parametric Pressure Loading for Aerodynamics

Defines a vector of pressure/unit dynamic pressure associated with a particular control vector. From this pressure vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEPRESS	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

EXAMPLE:

AEPRESS	0.90	SYMM	ASYMM	101	ALP1				

FIELDS:

Field	Contents
MACH	The Mach number for this force. See Remark 2. (Real ≥ 0.0, ≠ 1.0)
SYMXZ, SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
DMIJ	The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element downwashes.

REMARKS:

1. AEDW, AEFORCE, and AEPRESS are associated with the current AECONFIG using Case Control.

2. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

3. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero downwash (j-set) input is needed on the interference body elements.

AERO

Aerodynamic Physical Data

Gives basic aerodynamic parameters for unsteady aerodynamics.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AERO	ACSID	VELOCITY	REFC	RHOREF	SYMXZ	SYMXY			

EXAMPLE:

AERO	3	1.3+4	100.	1.-5	1	-1			

FIELDS:

Field	Contents
ACSID	Aerodynamic coordinate system identification. See Remark 2 . (Integer ≥ 0; Default is the basic coordinate system.)
VELOCITY	Velocity for aerodynamic force data recovery and calculation of reduced frequency. See Remark 5 . (Real)
REFC	Reference length for reduced frequency. (Real)
RHOREF	Reference density. (Real)
SYMXZ	The symmetry key for the aero coordinate x-z plane. See Remark 6 (Integer=+1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default=0).
SYMXY	The symmetry key for the aero coordinate x-y plane that can be used to simulate ground effect. (Integer=-1 for symmetry, 0 for no symmetry, and +1 for antisymmetry; Default=0)

REMARKS:

1. This entry is required for aerodynamic problems. Only one AERO entry is allowed.

2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction.

3. Set SYMXY=-1 to simulate ground effect.

4. PARAM,WTMASS does not affect aerodynamic matrices. RHOREF must be input in mass units.

5. VELOCITY is used only in aeroelastic response analysis, and it must be equal to V on the GUST Bulk Data entry.

6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AEROS

Static Aeroelasticity Physical Data

Defines basic parameters for static aeroelasticity.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AEROS	ACSID	RCSID	REFC	REFB	REFS	SYMXZ	SYMXY		

EXAMPLE:

AEROS	10	20	10.	100.	1000.	1			

FIELDS:

Field	Contents
ACSID	Aerodynamic coordinate system identification. See Remark 2 . (Integer ≥ 0; Default is the basic coordinate system).
RCSID	Reference coordinate system identification for rigid body motions. (Integer ≥ 0; Default is the basic coordinate system.)
REFC	Reference chord length. (Real > 0.0)
REFB	Reference span. (Real > 0.0)
REFS	Reference wing area. (Real > 0.0)
SYMXZ	The symmetry key for the aero coordinate x-z plane. See Remark 6 . (Integer=+1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default=0).
SYMXY	The symmetry key for the aero coordinate x-y plane that can be used to simulate ground effects. (Integer=+1 for antisymmetry, 0 for no symmetry, and -1 for symmetry; Default=0)

REMARKS:

1. This entry is required for static aeroelasticity problems. Only one AEROS entry is allowed.

2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction (T1).

3. The RCSID must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system.

4. REFB should be full span, even on half-span models.

5. REFS should be half area on half-span models.

6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AESTAT

Static Aeroelasticity Trim Variables

Specifies rigid body motions to be used as trim variables in static aeroelasticity.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AESTAT	ID	LABEL							

EXAMPLE:

AESTAT	5001	ANGLEA							

FIELDS:

Field	Contents
ID	Identification number of an aerodynamic trim variable degree-of-freedom. See Remark 1 . (Integer > 0).
LABEL	An alphanumeric string of up to eight characters used to identify the degree-of-freedom. See Remark 1 (Character. See Bulk Data Syntax Rules.)

REMARKS:

1. The degrees-of-freedom defined with this entry represent rigid body motion in the reference coordinate system defined on the AEROS entry. The standard labels that define the various rigid body motions are given below.

Table 10-1. Standard Labels Defining Rigid Body Motions		
LABEL	**Degree-of-Freedom Motion**	**Description**
ANGLEA	u_r (R2)	Angle of Attack
SIDES	u_r (R3)	Angle of Sideslip
ROLL	$\mathring{u}_r$ (R1)	Roll Rate
PITCH	$\mathring{u}_r$ (R2)	Pitch Rate
YAW	u_r (R3)	Yaw Rate
URDD1	$\ddot{u}_r$ (T1)	Longitudinal (See Remark 3 .)
URDD2	$\ddot{u}_r$ (T2)	Lateral
URDD3	$\ddot{u}_r$ (T3)	Vertical
URDD4	$\ddot{u}_r$ (R1)	Roll
URDD5	$\ddot{u}_r$ (R2)	Pitch
URDD6	$\ddot{u}_r$ (R3)	Yaw

These reserved names may be defined on the AEPARM entry instead. See the AEPARM, AEPRESS, and AEFORCE entries.

2. The degrees-of-freedom defined with this entry are variables in the static aeroelastic trim solution, unless they are constrained by referencing them with a TRIM Bulk Data entry.

3. If a label other than those above is specified, then the user must either generate the corresponding forces with an AELINK or via a DMI Bulk Data entry along with a DMAP alter that includes the DMIIN module and additional statements to merge into the appropriate matrices. Or, using AEPARM and AEDW, AEPRESS, and/or AEFORCE, you can accomplish this purpose without the need for any alters.

AESURF

Aerodynamic Control Surface

Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points. The forces associated with this controller will be derived from rigid rotation of the aerodynamic model about the hinge line(s) and from AEDW, AEFORCE and AEPRESS input data. The mass properties of the control surface can be specified using an AESURFS entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AESURF	ID	LABEL	CID1	ALID1	CID2	ALID2	EFF	LDW	
	CREFC	CREFS	PLLIM	PULIM	HMLLIM	HMULIM	TQLLIM	TQULIM	

EXAMPLE:

AESURF	6001	ELEV	100	100	200	200			
	10.0	180.0			-1.4E4	1.2E4	20	30	

FIELDS:

Field	Contents
ID	Controller identification number. (Integer > 0)
LABEL	Controller name, see Remark 1 (Character. See Bulk Data Syntax Rules.)
CIDi	Identification number of a rectangular coordinate system with a y-axis that defines the hinge line of the control surface component. (Integer > 0)
ALIDi	Identification of an AELIST Bulk Data entry that identifies all aerodynamic elements that make up the control surface component. (Integer > 0)

Field	Contents
EFF	Control surface effectiveness. See Remark 4 (Real ≠ 0.0; Default=1.0).
LDW	Linear downwash flag. See Remark 2 (Character, one of LDW or NOLDW; Default=LDW).
CREFC	Reference chord length for the control surface. (Real > 0.0; Default=1.0)
CREFS	Reference surface area for the control surface. (Real > 0.0; Default=1.0)
PLLIM, PULIM	Lower and upper deflection limits for the control surface in radians. (Real; Default=± π/2)
HMLLIM, HMULIM	Lower and upper hinge moment limits for the control surface in force-length units. (Real, Default=no limit)
TQLLIM, TQULIM	Set identification numbers of TABLEDi entries that provide the lower and upper deflection limits for the control surface as a function of the dynamic pressure. (Integer > 0, Default=no limit)

REMARKS:

1. The IDs on AESURF, AESTAT, and AEPARM entries are ignored. AESURFS can be used to define mass properties of the control surface.

2. The degrees-of-freedom defined on this entry represent a rigid body rotation of the control surface components about their hinge lines. In the default LDW (Linear DownWash) case, the downwash due to a unit perturbation of the control surface will be computed as part of the database. In the NOLDW case, the user must prescribe the controller's effects by direct definition of the induced forces using the AEPRESS, AEDW and/or AEFORCE entries.

3. Either one or two control surface components may be defined.

4. If EFF is specified, then the forces produced by this surface are modified by EFF (e.g., to achieve a 40% reduction, specify EFF=0.60).

5. The continuation is not required.

6. The CREFC and CREFS values are only used in computing the nondimensional hinge moment coefficients.

7. Position limits may be specified using either PiLIM or TQiLIM, but not both.
8. Position and hinge moment limits are not required.

AESURFS

Structural Grids on an Aerodynamic Control Surface

Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry. The mass associated with these structural nodes define the control surface moment(s) of inertia about the hinge line(s).

FORMAT:

1	2	3	4	5	6	7	8	9	10
AESURFS	ID	LABEL		LIST1		LIST2			

EXAMPLE:

AESURFS	6001	ELEV		6002		6003			

FIELDS:

Field	Contents
ID	Controller identification number. See Remark 1 (Integer > 0).
LABEL	Controller name. See Remark 1 (Character. See Bulk Data Syntax Rules.)
LISTi	Identification number of a SET1 entry that lists the structural grid points that are associated with this component of this control surface. (Integer > 0)

REMARKS:

1. The LABEL on the AESURFS entry must match one on an AESURF entry. The ID is ignored.

2. The mass of the GRID points listed on the SETi entries is used to compute the mass moment of inertia of the control surface about its i'th hinge line. The presence of these data will allow the hinge moments to include the inertial forces in the computations. These data are optional, and, if omitted, result in hinge moments which include only the applied, aeroelastically corrected, forces.

3. These data will be associated with a structural superelement by grid list or partitioned SUPER=<seid> if the AESURFS is defined in the main bulk data section.

AMLREG

Automatically Matched Layer (AML) Region Definition

Defines automatically matched layer region for acoustic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
AMLREG	RID	SID	Name/Descriptor						
	NL	RADSURF	INFID1	INFID2	INFID3				

EXAMPLE:

AMLREG	1	10	AML REGION 1						
	7	AML	12	13	14				

FIELDS:

Field	Contents
RID	AML region identification number. (Integer > 0) See Remark 1.
SID	Surface identification number. (Integer > 0) See Remark 2.
Name/ Descriptor	The Name/Descriptor is an optional character string. (Character; 48 characters maximum; See Bulk Data Syntax Rules) See Remark 3.
NL	Number of layers of extrusion to be formed by solver. (Integer > 0; Default = 5)

Field	Contents
RADSURF	Radiation surface type. (Character; Default = AML) If RADSURF = AML, the pressure and velocities on the AML boundary are used to compute results in the far field. If RADSURF = PHYB, the pressure and velocities on the physical boundary (that is, all free fluid faces with the exception of faces on the AML and the infinite planes) are used to compute results in the far field. If RADSURF = NONE, the region does not radiate.
INFIDi	Identification number of an infinite plane. Up to three (3) infinite planes are considered for an AML region. The infinite planes are used when acoustic results are to be computed exterior to the AML region. (Integer ≥ 0; Default = 0) See Remark 4.

REMARKS:

1. RID must be unique with respect to all other AMLREG bulk entries. AMLREG definitions are only valid for analyses with AML method.

2. The SID references a BSURFS bulk entry to define the fluid element faces used to describe the AML region. The element faces on the BSURFS entry **must** all be on fluid elements.

3. Because of the name/descriptor field, free-field and large field data formats are not supported for this bulk entry.

4. If INFIDi = 0, there is no infinite plane for this AML region. If INFIDi > 0, INFIDi references an IPLANE bulk entry to define the infinite plane. If infinite planes exist, then every AML region free edge must belong to an infinite plane. Each and every AML region free edge must belong to one and only one infinite plane. No other element edges in the AML region except free edges can belong to an infinite plane.

ASET

Degrees-of-freedom for the a-set

Defines degrees-of-freedom in the analysis set (a-set).

FORMAT:

1	2	3	4	5	6	7	8	9	10
ASET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

ASET	16	2	23	3516	1	4			

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive a-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

3. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry, is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

ASET1

Degrees-of-freedom for the a-set, Alternate Form of ASET Entry

Defines degrees-of-freedom in the analysis set (a-set).

FORMAT:

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

EXAMPLE:

ASET1	345	2	1	3	10	9	6	5	
	7	8							

ALTERNATE FORMAT AND EXAMPLE:

ASET1	C	ID1	“THRU”	ID2					
ASET1	123456	7	THRU	109					

FIELDS:

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; for THRU option, ID1 < ID2)

REMARKS:

1. Degrees-of-freedom specified on this entry form members of the a-set that are exclusive from other sets defined by Bulk Data entries. See “Degree-of-Freedom Sets” for a list of these entries.

2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

3. If the alternate format is used, all points in the sequence ID1 through ID2 are not required to exist, but there must be at least one degree-of-freedom in the a-set for the model, or a fatal error will result. Any points implied in the THRU that do not exist will collectively produce a warning message but will otherwise be ignored.

4. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified, then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

BAROR

CBAR Entry Default Values

Defines default values for field 3 and fields 6 through 8 of the CBAR entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3		

EXAMPLE:

BAROR		39			0.6	2.9	-5.87		

ALTERNATE FORMAT AND EXAMPLE

BAROR		PID			G0				

BAROR		39			18				

FIELDS:

Field	Contents
PID	Property identification number of the PBAR entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$, using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)

REMARKS:

1. The contents of fields on this entry will be assumed for any CBAR entry whose corresponding fields are blank.

2. Only one BAROR entry is allowed.

3. For an explanation of bar element geometry, see “CBAR Element”in the *NX Nastran Element Library*.

4. If field 6 is an integer, then G0 is used to define the orientation vector and X2 and X3 must be blank. If field 6 is real or blank, then X1, X2, and X3 are used.

BCONP

Slideline Contact Parameters

Defines the parameters for a slideline contact region and its properties.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCONP	ID	SLAVE	MASTER		SFAC	FRICID	PTYPE	CID	

EXAMPLE:

BCONP	95	10	15		1.0	33	1		

FIELDS:

Field	Contents
ID	Contact region identification number. See Remark 1 (Integer > 0).
SLAVE	Slave region identification number. See Remark 2 (Integer > 0).
MASTER	Master region identification number. See Remark 3 (Integer > 0).
SFAC	Stiffness scaling factor. SFAC is used to scale the penalty values automatically calculated by the program. See Remark 4 (Real > 0.0, Default=1.0).
FRICID	Contact friction identification number. See Remark 5 (Integer > 0 or blank).
PTYPE	Penetration type. See Remark 6 . (Integer 1 or 2, Default=1) 1: unsymmetrical (slave penetration only—Default) 2: symmetrical

Field	Contents
CID	Coordinate system identification number to define the slideline plane vector and the slideline plane of contact. See Remark 7 . (Integer > 0, Default=0, which means the basic coordinate system.)

REMARKS:

1. The ID field must be unique with respect to all other BCONP identification numbers.

2. The referenced SLAVE is the identification number in the BLSEG Bulk Data entry. This is the slave line. The width of each slave segment must also be defined to get proper contact stresses. See the Bulk Data entry, “BWIDTH” for details on specifying widths.

3. The referenced MASTER is the identification number in the BLSEG Bulk Data entry. This is the master line. For symmetrical penetration, the width of each master segment must also be defined. See the Bulk Data entry, “BWIDTH” for details on specifying widths.

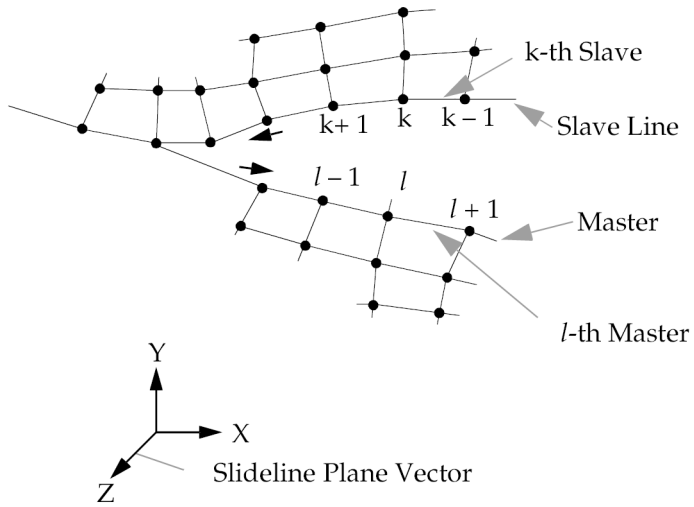
4. SFAC may be used to scale the penalty values automatically calculated by the program. The program calculates the penalty value as a function of the diagonal stiffness matrix coefficients that are in the contact region. In addition to SFAC, penalty values calculated by the program may be further scaled by the ADPCON parameter. (See the ADPCON parameter description for more details). The penalty value is then equal to k*SFAC*|ADPCON|, where k is a function of the local stiffness. It should be noted that the value in SFAC applies to only one contact region, whereas the ADPCON parameter applies to all the contact regions in the model.

5. The referenced FRLCLD is the identification number of the BFRLC Bulk Data entry. The BFRLC defines the frictional properties for the contact region.

6. In an unsymmetrical contact algorithm, only slave nodes are checked for penetration into master segments. This may result in master nodes penetrating the slave line. However, the error depends only on the mesh discretization. In symmetric penetration, both slave and master nodes are checked for penetration. Thus, no distinction is made between slave and master. Symmetric penetration may be up to thirty percent more expensive than the unsymmetric penetration.

7. In Figure 10-1, the unit vector in the Z-axis of the coordinate system defines the slideline plane vector. The slideline plane vector is normal to the slideline plane. Relative motions outside the slideline plane are ignored, and therefore

must be small compared to a typical master segment. In a master segment, the direction from master node 1 to master node 2 gives the tangential direction (t). The normal direction for a master segment is obtained by the cross product of the slideline plane vector with the unit tangent vector (i.e., n=z x t). The definition of the coordinate system should be such that the normal direction must point toward the slave region. For symmetric penetration, the normals of master segments and slave segments must face each other. This is generally accomplished by traversing from master line to slave line in a counterclockwise or clockwise fashion depending on whether the slideline plane vector forms a right-hand or left-hand coordinate system with the slideline plane.



- The X-Y plane is the slideline plane. Unit normal in the Z-direction is the slideline plane vector.
- Arrows show positive direction for ordering nodes. Counterclockwise from master line to slave line.

Figure 10-1. A Typical Finite Element Slideline Contact Region

BCPROP

Surface-to-Surface Contact or Glue Region Definition by Property ID

Defines a surface-to-surface contact or glue region by property IDs of shell elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	IP10	-etc-					

EXAMPLE:

BCPROP	3	1	5	8	3	22			

FIELDS:

Field	Contents
ID	Identification number of a contact or glue region. See Remark 2 and 3. (Integer > 0)
IPi	ID of a PSHELL, PCOMP, or PCOMPG entry (Integer > 0)

REMARKS:

1. The continuation field is optional.
2. BCPROP is a collection of one or more shell property IDs. BCPROP defines a contact or glue region which may act as a source or target.
3. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.

REMARKS RELATED TO SOLS 601 AND 701:

1. ID must be less than 100000. Otherwise, the program will issue an error.

BCPROPS

Contact or glue region definition by PSOLID or PCOMPS property ID

Defines a surface-to-surface contact or glue region by PSOLID or PCOMPS property ID.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCPROPS	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	IP10	-etc-					

EXAMPLE:

BCPROPS	3	1	5	8	3	22			

FIELDS:

Field	Contents
ID	Identification number of a contact or glue region. See Remark 2 and 3. (Integer > 0)
IPi	PSOLID or PCOMPS entry ID. (Integer > 0)

REMARKS:

1. The continuation field is optional.
2. BCPROPS is a collection of one or more solid property IDs. BCPROPS defines a contact or glue region formed by the free faces of the solid elements (CHEXA, CPENTA, CPYRAM, or CTETRA) and may act as a source or target.
3. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.

4. Only the solid element free faces within the given set of PIDs are considered in the contact and/or glue algorithm. These free faces are automatically determined by the software.

5. The collapsed CHEXA element used in a SOL 401 crack simulation is not supported in a glue or contact region.

REMARKS RELATED TO SOLS 601 AND 701:

1. ID must be less than 100000. Otherwise, the program will issue an error.

BCRPARA

Contact Face and Edge Region Parameters

Defines parameters for a contact face or edge region.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCRPARA	CRID	SURF	OFFSET	TYPE	MGP				

EXAMPLE:

BCRPARA	1	TOP	0.02						

FIELDS:

Field	Contents
CRID	Contact region ID. (Integer > 0)
SURF	Indicates the contact side of shell elements. See Remark 1 and SOL 601 and 701 Remarks 1 and 2. (Character=“TOP” or “BOT”; Default=“TOP”)
OFFSET	Offset distance for an edge or surface contact region. For SOLs 101, 103, 111, 112, and 401: Real, Default = 0.0 For SOLs 601 and 701: Real >= 0.0, Default = OFFSET value in BCTPARA entry. See SOL 601 and 701 Remark 3.
TYPE	Indicates whether a 3D contact region is a rigid (target) surface or is a shell coating on solid elements. See Remarks 4 to 8 in the SOL 601 and 701 remarks. (Character=“FLEX”, “RIGID” or “COATING”. Default=“FLEX”). This is only supported by SOLs 601 and 701.
MGP	Master grid point for a target contact region with TYPE=RIGID or when the rigid-target algorithm is used. The master grid point may be used to control the motion of a rigid surface. (Integer ≥ 0,; Default=0) This is only supported by SOLs 601 and 701.

REMARKS:

1. The BCRPARA bulk entry is not required. When it is not present, the default values are used.

2. SURF is used to define the contact side of shell element regions. When SURF is "TOP", the contact side is consistent with the shell element normal and when "BOT" the opposite. In a SOL 101 (including consecutive solutions 103, 111 and 112), SURF must be defined so that source and target contact sides either face one another to represent a separation condition, or oppose one another to represent an interference condition.

REMARKS RELATED TO SOL 601 AND 701:

1. For a 3D contact region, SURF is applicable only for a single-sided contact region defined on shell elements (i.e., using BSURF or BCPROP) and is used as a target region. For a contact region defined on 3D solid element faces (i.e., using BSURFS), the contact side is automatically determined by the program. For a contact region used as a source region (contactor), it does not matter which is the contact side.

2. For a 2D contact region, SURF is only applicable to a rigid target region defined by grid points that are not attached to any elements. For a 2D contact region with underlying 2D elements, the contact side is automatically determined by the program.

3. OFFSET is only applicable when the rigid target algorithm is not selected, (i.e., TYPE or XTYPE=0 or 1 in the BCTPARA entry).

4. TYPE=“RIGID” is ignored if the contact region is used as a source region.

5. If the rigid target algorithm is selected (i.e., TYPE=2 (SOL 601) or XTYPE= 3 (SOL 701) in BCTPARA command), the target region must be attached to shell elements only (i.e., using BSURF or BCPROP) and it is automatically set as rigid.

6. TYPE and MGP are interpreted as follows:

   - 2D target regions using grids only (not attached to underlying elements) are always rigid. MGP>0 can be specified without TYPE=RIGID.

   - 2D target regions with underlying elements will behave as rigid only when both TYPE=RIGID and MGP>0 are specified. Otherwise, the target region is flexible and MGP is ignored.

- 3D target regions that are attached to solid elements will behave as rigid only when both TYPE=RIGID and MGP>0 are specified. Otherwise, the target region is flexible and MGP is ignored.

- 3D target regions that are attached to shell elements (i.e., using BSURF or BCPROP), only TYPE=RIGID is required for the region to behave as rigid. MGP is ignored when TYPE=FLEX.

7. TYPE="COATING" should only be specified for a contact region defined on shell elements (i.e., using BSURF or BCPROP), coated on solid elements. The contact region will be transferred onto the solid element faces and the shell elements will be deleted. If there are no underlying solid elements, an error message will be issued.

8. A contact region is specified as a target or source region in the BCTSET entry.

9. When TYPE=COATING, the shell elements should match the solid element faces, i.e. element edges should match, linear shell on linear solid face, and parabolic shell on parabolic solid face.

BCTADD

Surface-to-Surface Contact Set Combination (SOLs 101, 103, 111, 112, 401, 601 and 701)

Defines a surface-to-surface contact set as a union of contact sets defined on BCTSET entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCTADD	CSID	SI	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc-						

EXAMPLE:

BCTADD	12	1	3	6	5	9			

FIELDS:

Field	Contents
CSID	Contact set identification number. (Integer > 0)
Si	Identification numbers of contact sets defined via BCTSET entries. (Integer > 0)

REMARKS:

1. Multiple contact sets (BCTSETs) with unique CSIDs can be combined by including their CSIDs on a BCTADD entry. The BCTADD entry has its own unique CSID which is used on the BCSET case control command. Multiple contact sets with their own CSID are necessary when different parameters (BCTPARA or BCTPARM) are needed for different contact sets.

2. To include several contact sets defined via BCTSET entries in a model, BCTADD must be used to combine the contact sets. CSID in BCTADD is then selected with the Case Control command BCSET.

3. Si must be unique and may not be the identification of this or any other BCTADD entry.

BCTPARA

SOLs 601 and 701 Contact Set Parameters

Defines parameters for a SOL 601 or 701 contact set.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCTPARA	CSID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	-etc-				

EXAMPLE:

BCTPARA	2	EPSN	1.0E-10	TZPENE	0.1	CSTIFF	1		
	OFFTYPE	2	INIPENE	3					

FIELDS:

Field	Contents
CSID	Contact set ID. Parameters defined in this command apply to contact set CSID defined by a BCTSET entry. (Integer > 0)
PARAMi	Name of the BCTPARA parameter. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. See Table 10-2 for parameter listing. (Real or Integer)

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
General Parameters	

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
TYPE	Selects the type of contact algorithm. TYPE = 2 is not applicable for 2D contact. (Integer; Default=0) 0 – Constraint function algorithm is used. 1 – Segment method algorithm is used. 2 – Rigid target algorithm is used.
NSIDE	Flags single or double-sided contact. NSIDE is not applicable for 2D contact. (Integer; Default=1) 1 – Contact surfaces are single-sided. 2 – Contact surfaces are double-sided.
TBIRTH	Birth time for contact set. (Real; Default=0.0; Unit = Time)
TDEATH	Death time for contact set. (Real; Default=0.0; Unit = Time) If TDEATH ≤ TBIRTH, it is ignored.
INIPENE	Flags how initial penetrations or gaps are handled. See Remark 3. (Integer; Default=0) 0 – Initial penetrations are eliminated. 1 – Initial penetrations are eliminated and the list of penetrating nodes is printed. 2 – Initial penetrations are ignored. In successive steps, each contractor node is allowed to penetrate the target up to its initial penetration. 3 – Initial penetrations or gaps are overridden by GAPVAL. This option is not available for rigid target algorithm (TYPE = 2).
GAPVAL	Specifies a constant gap distance between the source region (contactor) and the target region when INIPENE = 3. Negative GAPVAL means initial penetrations which will be eliminated. (Real, Default = 0.0; Unit = Length)

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
PDEPTH	Penetration depth used to limit contact detection for single-side target contact surface (i.e., NSIDE=1). (Real >= 0.0; Default=0.0; Unit = Length) If PDEPTH = 0.0, then contact is detected whenever a node on the contactor surface is below the target surface, i.e., any amount of penetration. If PDEPTH > 0.0, then contact is detected only if the penetration is less than or equal to PDEPTH.
SEGNORM	Indicates whether a continuous (interpolated) contact segment normal is used for the contact surfaces. (Integer; Default=0) 0 – SEGNORM=1 if NSIDE=1, SEGNORM=-1 if NSIDE=2. 1 – Continuous segment normal is used. -1 – Continuous segment normal is not used.
OFFTYPE	Type of offset for contact regions. OFFTYPE is not applicable for 2D contact. (Integer; Default=0) 0 – Use offset value specified by OFFSET for single-sided contact only. (i.e., NSIDE=1). Use offset of 0.001 for double-sided contact (i.e., NSIDE=2) 1 – Use specified offset value for either single- or double-sided contact. 2 – Half the shell thickness is used for contact regions on shell elements and no offset is used otherwise.
OFFSET	Default offset distance for contact regions. (Real ≥ 0.0; Default=0.0; Unit = Length) Note: For contact algorithm TYPE=0 or 1, individual offset distances can be specified for each contact region using the BCRPARA entry to override the default offset distance specified here.
Standard contact algorithm (TYPE=0 or 1)	

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
DISP	Selects the displacement formulation used for this contact set. (Integer; Default = 0) 0 – Use the formulation selected by CTDISP in NXSTRAT entry (default) 1 – Use small displacement formulation (contact conditions are not updated) 2 – Use large displacement formulation (contact conditions are updated)
TZPENE	Time to eliminate initial penetrations. (Real ≥0.0; Default=0.0; Unit = Time) If TZPENE=0.0 and INIPENE=0 or 1, then the initial penetrations are eliminated in the first time step. This may cause convergence difficulties for certain problems. By using TZPENE > 0.0, the initial penetrations are eliminated gradually over time TZPENE.
CSTIFF	Indicates whether consistent contact stiffness is used. (Integer; Default=0) 0 – Consistent contact stiffness is not used 1 – Consistent contact stiffness is used
TIED	Indicates whether contact regions in each contact pair are tied together. Currently, tied contact option assumes small rotations of the contact regions. (Integer; Default=0) 0 – Not tied 1 – Tied
TIEDTOL	Tolerance used to determine whether contactor nodes are tied to the target region when TIED=1 is specified. A contactor node is tied to its target region if the distance between them is less than or equal to TIEDTOL. (Real; Default=0.0; Unit = Length)

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
EXTFAC	Factor for extending contact surfaces beyond their boundaries. The amount of extension is given by this factor multiplied by the length of the contact segments. (1.0E-6 ≤ Real ≤ 0.1; Default=0.001; Unitless)
FRICMOD	Type of friction model (0 ≤ Integer ≤ 13; Default=0). See Remark 1. 0 – Constant coefficient of friction specified for each contact pair, i.e. FRICi in the BCTSET entry. 1 – Constant coefficient of friction specified by FPARA1. 2 – Friction model 1; uses FPARA1 and FPARA2. 3 – Friction model 2; uses FPARA1, FPARA2, and FPARA3. 4 – Use different static and dynamic friction coefficients; uses FPARA1, FPARA2, and FPARA3. 5 – Friction coefficient varies with sliding velocity; uses FPARA1, FPARA2 and FPARA3. 6 – Anisotropic friction model; uses FPARA1, FPARA2, FPARA3, FPARA4, and FPARA5. 7 – Friction coefficient varies with consistent contact force; uses FPARA1 and FPARA2. 8 – Friction coefficient varies with time; uses FPARA1, FPARA2 and FPARA3. 9 – Friction coefficient varies with coordinate values; uses FPARA1, FPARA2, FPARA3, FPARA4, and FPARA5. 12 – Modified friction model 1; uses FPARA1 and FPARA2. 13 – Modified friction model 2; uses FPARA1, FPARA2, and FPARA3. See Section 4.4 of the Advanced Nonlinear Theory and Modeling Guide for a description of friction models.
FPARA1	Friction parameter A1

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
FPARA2	Friction parameter A2
FPARA3	Friction parameter A3
FPARA4	Friction parameter A4
FPARA5	Friction parameter A5
FRICDLY	Indicates whether the application of friction is delayed, i.e., friction is applied on a node one time step after the node comes into contact. Delay of friction application may improve the convergence of the solution. (Integer; Default = 0) 0 – No delay 1 – Delay
	Constraint Function Contact Algorithm (Type=0)
EPSN	Parameter for normal constraint function, w. (Real; Default=1.0E-12; Unitless)
EPST	Parameter for frictional constraint function, v. (Real > 0.0; Default=0.001 Unit = Length/Time)
CFACTOR1	Compliance factor. (Real; Default=0.0; Unit = $Length^3$/Force)
	Thermo-Mechanical Coupling (for TYPE=0 only)
TMCHHAT	Contact heat transfer coefficient used for calculating the amount of heat transfer between bodies in contact. (Real ≥ 0.0, Default = 0.0; Unit = Watt/($Length^2$*Temperature))
TMCFC	Coefficient that specifies the proportion of heat generated due to frictional contact going to the contactor body. (0.0 ≤ Real ≤ 1.0, Default = 0.5, Unitless; TMCFC + TMCFT ≤ 1.0)
TMCFT	Coefficient that specifies the proportion of heat generated due to frictional contact going to the target body. (0.0 ≤ Real ≤ 1.0, Default = 0.5, Unitless; TMCFC + TMCFT ≤ 1.0)

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
	Rigid Target Algorithm (TYPE=2) (3D contact only)
NCMOD	Normal contact modulus. (Real; Default=1.0E11; Unit = Force/Length)
TFORCE	The maximum tensile contact force allowed for a converged solution. (Real ≥ 0.0, Default = 0.001; Unit = Force)
SLIDVEL	The maximum sliding velocity used in modeling sticking friction. When the velocity is smaller than SLIDVEL, sticking is assumed; when the velocity is larger than SLIDVEL, sliding is assumed. (Real > 0.0, Default = 1.0E-10; Unit = Length/Time)
OCHECK	Specifies whether oscillation checking is performed and when it is done. (Integer >= 0; Default = 5) If OCHECK=0, no oscillation checking is performed. Otherwise, oscillation checking is performed after equilibrium iteration OCHECK. Oscillation checking consists of two checks: a) If a contactor node oscillates between two neighboring target segments during the equilibrium iterations, oscillation checking puts the contactor node into contact with the boundary edge between the target segments. b) In analysis with friction, if the sliding velocity of a contactor node oscillates during the equilibrium iterations, oscillation checking puts the contactor node into sticking contact.
GAPBIAS	Contact is detected when the distance between the target and contactor (accounting for any offsets) is less than GAPBIAS. (Real; Default = 0.0; Unit = Length)

Table 10-2. BCTPARA Parameters for SOL 601

Name	Description
OFFDET	Selects the implementation of offsets. (Integer; Default = 0) 0 – Program chooses the implementation based upon the shape of the target surfaces; if a target surface is flat or convex, spheres are used, otherwise, normals are used. 1 – A sphere of radius equal to the offset is placed around each contactor node, and contact is detected between the sphere and the target surface. 2 – Two surfaces are constructed for each contactor surface: an upper surface and a lower surface. These surfaces are constructed using the offsets and the averaged contactor normals. Contact is then detected between points on the constructed contactor surfaces and target surface.

Table 10-3. BCTPARA Parameters for SOL 701 (3D contact only)

Name	Description
	General Parameters
XTYPE	Selects the type of contact algorithm. (Integer; Default=0) 0 – Kinematic constraint algorithm is used. 1 – Penalty algorithm is used. 3 – Rigid target algorithm is used.
NSIDE	Flags single or double-sided contact. (Integer; Default=1) 1 – Contact surfaces are single-sided. 2 – Contact surfaces are double-sided.
TBIRTH	Birth time for contact set. (Real; Default=0.0; Unit = Time)
TDEATH	Death time for contact set. (Real; Default=0.0; Unit = Time) If TDEATH ≤ TBIRTH, it is ignored.

Table 10-3. BCTPARA Parameters for SOL 701 (3D contact only)

Name	Description
INIPENE	Flags how initial penetrations are handled. See Remark 3. (Integer; Default=0) 0 – Initial penetrations are eliminated. 1 – Initial penetrations are eliminated and the list of penetrating nodes is printed. 2 – Initial penetrations are ignored. In successive steps, each contactor node is allowed to penetrate the target up to its initial penetration.
PDEPTH	Penetration depth used to limit contact detection for single-side target contact surface (i.e., NSIDE=1). (Real >= 0.0; Default=0.0; Unit = Length) If PDEPTH = 0.0, then contact is detected whenever a node on the contactor surface is below the target surface, i.e., any amount of penetration. If PDEPTH > 0.0, then contact is detected only if the penetration is less than or equal to PDEPTH.
OFFTYPE	Type of offset for contact regions. (Integer; Default=0) 0 – Use offset value specified by OFFSET for single-sided contact only (i.e., NSIDE=1). Use offset of 0.001 for double-sided contact (i.e., NSIDE=2). 1 – Use specified offset value for either single- or double-sided contact. 2 – Half the shell thickness is used for contact regions on shell elements and no offset is used otherwise.
OFFSET	Default offset distance for contact regions. (Real; Default=0.0; Unit = Length) Note: For contact algorithm XTYPE=0 or 1, individual offset distances can be specified for each contact region using the BCRPARA entry to override the default offset distance specified here.
Contact Algorithm XTYPE=0 or 1	

Table 10-3. BCTPARA Parameters for SOL 701 (3D contact only)

Name	Description
TZPENE	Time to eliminate initial penetrations. (Real=0.0; Default=0.0; Unit = Time) If TZPENE=0.0 and INIPENE=0 or 1, then the initial penetrations are eliminated in the first time step. This may cause convergence difficulties for certain problems. By using TZPENE > 0.0, the initial penetrations are eliminated gradually over time TZPENE.
EXTFAC	Factor for extending contact surfaces beyond their boundaries. The amount of extension is given by this factor multiplied by the length of the contact segments. (1.0E-6 ≤ Real ≤ 0.1; Default=0.001; Unitless)
	Penalty Contact Algorithm (XTYPE=1)
XKNCRIT	Selects the criterion for evaluation of normal penalty stiffness. (Integer; Default=0) 0 - The program calculates the normal penalty stiffness. 1 - The user specifies the normal penalty stiffness (XKN).
XKN	Specifies the normal penalty stiffness to be used when XKNCRIT=1. (Real; Unit = Force/Length^3)
XKTCRIT	Selects the criterion for evaluation of tangential penalty stiffness. (Integer; Default=0) 0 - The program calculates the normal penalty stiffness 1 - The user specifies the normal penalty stiffness (XKT).
XKT	Specifies the tangential penalty stiffness to be used when XKTCRIT=1. (Real; Unit = Force*Time/Length^3)

Table 10-3. BCTPARA Parameters for SOL 701 (3D contact only)

Name	Description
XDAMP	Indicates whether damping is used when the penalty explicit contact algorithm is used. (Integer; Default=0) 0 – Damping is not used, i.e., the XNDAMP parameter is ignored. 1 – Damping is used and XNDAMP is a factor of the critical damping, i.e., the damping coefficient is given by XNDAMP multiplied by the critical damping. This is the recommended choice if damping is used. 2 – Damping is included and the damping coefficient is specified directly by XNDAMP.
XNDAMP	Specifies the relative or absolute damping coefficient (for normal penalty stiffness) when the penalty explicit contact algorithm is used and XDAMP=1 or 2. (Real=0.0; Default=0.1; Unit = Force*Time/Length^3)
Rigid Target Algorithm (XTYPE=3)	
SLIDVEL	The maximum sliding velocity used in modeling sticking friction. When the velocity is smaller than SLIDVEL, sticking is assumed; when the velocity is larger than SLIDVEL, sliding is assumed. (Real > 0.0, Default = 1.0E-10)
GAPBIAS	Contact is detected when the distance between the target and contactor (accounting for any offsets) is less than GAPBIAS. (Real; Default = 0.0)
OFFDET	Selects the implementation of offsets. (Integer; Default = 0) 0 – Program chooses the implementation based upon the shape of the target surfaces; if a target surface is flat or convex, spheres are used, otherwise, normals are used. 1 – A sphere of radius equal to the offset is placed around each contactor node, and contact is detected between the sphere and the target surface. 2 – Two surfaces are constructed for each contactor surface: an upper surface and a lower surface. These surfaces are constructed using the offsets and the averaged contactor normals. Contact is

Table 10-3. BCTPARA Parameters for SOL 701 (3D contact only)

Name	Description
	then detected between points on the constructed contactor surfaces and target surface.

REMARKS:

1. Field 9 on line 1 should be blank. Beginning on the continuation lines, fields 2 to 9 can be used for 4 pairs of PARAM/VALUE.

2. The BCTPARA bulk entry is not required. When it is not present, the default values are used. At least one parameter should be defined when a BCTPARA entry exists.

3. Shell element thicknesses or contact offsets defined with the OFFSET parameter can optionally be included in a contact definition with the appropriate OFFTYPE parameter setting. They are applied *before* the software removes any gaps or penetrations as a result of the INIPENE setting. You can optionally use INIPENE=3 and GAPVAL to define a consistent gap or penetration to the contact set.

BCTPARM

Surface-to-Surface Contact Parameters (SOLs 101, 103, 111, 112, and 401)

Control parameters for the surface-to-surface contact algorithm.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCTPARM	CSID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	-etc-				

EXAMPLE:

BCTPARM	1	PENN	10.0	PENT	0.5	CTOL	0.001		
	SHLTHK	1							

FIELDS:

Field	Contents
CSID	Contact set ID. Parameters defined in this command apply to contact set CSID defined by a BCTSET entry. (Integer > 0)
PARAMi	Name of the BCTPARM parameter. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. See Table 10-4 for the parameter listing. (Real or Integer)

Table 10-4. Primary parameters supported by SOLs 101, 103, 111, and 112:

Name	Description
CTOL	Contact force convergence tolerance. (Default=0.01)
MAXF	Maximum number of iterations for a force loop. (Default=10)

Table 10-4. Primary parameters supported by SOLs 101, 103, 111, and 112:

Name	Description
MAXS	Maximum number of iterations for a status loop. (Default=20)
NCHG	Allowable number of contact changes for convergence. (Default=0.02). See Remark 3.
INIPENE*	Use when the goal is for a pair of contact regions to be initially touching without interference, but due to the faceted nature of finite elements around curved geometry, some of the element edges or faces may have a slight gap or penetration. See Remarks 7 and 8. 0 or 1 - Contact is evaluated exactly as geometry is modeled. No corrections will occur for gaps or penetrations (Default). 2 - Penetrations will be reset to a new initial condition in which there is no interference. 3 - Gaps and penetrations are both reset to a new initial condition in which there is no interference.
SHLTHK*	Shell thickness offset flag. 0 - Includes half shell thickness as surface offset. (Default) 1 - Does not include thickness offset.
ZOFFSET	Determines if the shell element z-offset is included in the contact solution. 0 - Includes the shell z-offset when determining the contact surfaces (Default). 1 - Does not include the shell z-offset when determining the contact surfaces.

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
CNTCONV	Contact convergence criteria. 1 – The contact convergence criteria is based on PTOL. (Default) 2 – The convergence criteria is based on CTOL.

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
PTOL*	Contact penetration tolerance used when CNTCONV=1. If the contact penetrations exceed the penetration tolerance, an extra augmentation loop is performed. If the penetrations are below this tolerance, the augmentation loop is considered converged. In addition, if the global solution convergence criteria is satisfied, then the time step is considered converged. (REAL≠0.0; Default = 0.01 *characteristic length) A positive value is scaled by the characteristic length computed for the contact pair. If a negative value is defined, the absolute value is used, but the value is not scaled by the characteristic length.
CTOL	Contact augmentation traction convergence used when CNTCONV=2. The augmentation loop convergence criteria can be based on traction convergence. The contact force ratio FRAT is determined as: $$FRAT = \sqrt{\frac{((\lambda^k - \lambda^{k-1}) * (\lambda^k - \lambda^{k-1})}{(\lambda^k * \lambda^k)}}$$ where k is the augmentation loop id. If FRAT < CTOL, the contact augmentation loop is considered converged. (Default = 0.05)
RCTOL	Iterative contact force convergence. (Default = 0.05; Real>0.0)
MAXS	Maximum number of augmentation (outer) loops. If the augmentation loop has not converged in MAXS number of iterations, the solution will proceed to the next step if the global convergence criteria has been met. Setting MAXS=1 selects a pure penalty formulation. (Default = 20; INTEGER≥1)

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
INIPENE*	Use when the goal is for a pair of contact regions to be touching without interference, but due to the faceted nature of finite elements around curved geometry, some of the element edges or faces may have a slight gap or penetration. See Remark 8. 0 or 1 - Contact is evaluated exactly as the geometry is modeled. No corrections will occur for gaps or penetrations (Default). 2 - Penetrations will be reset to a new initial condition in which there is no interference. 3 - Gaps and penetrations are both reset to a new initial condition in which there is no interference. INIPENE is applied when contact elements are initially created, and if they are recreated as a result of large displacement effects when PARAM,LGDISP,1 is defined.
OPNSTF*	Open contact stiffness scale factor. The open contact stiffness is computed by OPNSTF * closed stiffness. (OPNSTF default = 1.0E-6)
OPNTOL*	Open gap tolerance scale factor. The open contact stiffness (OPNSTF * closed stiffness) is applied to the contact elements that have a gap value less than or equal to OPNTOL * characteristic length, but greater or equal than GAPTOL * characteristic length. The contact element stiffness is 0.0 if the gap is greater than OPNTOL * characteristic length. (OPNTOL default = 1.0)
GAPTOL*	Closed gap tolerance scale factor. The closed contact stiffness is applied to the contact elements that have a gap less than GAPTOL * characteristic length. (Default = 1.0E-10)

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
NOSEP*	No separation contact option. NOSEP=0 (default): When contact stiffness is recomputed in a consecutive nonlinear iteration, contact elements which are inactive as a result of normal tractions=0.0 and no penetration, and which have a gap greater than GAPTOL * characteristic length will remain inactive in the consecutive iteration. NOSEP=1: The open contact stiffness (OPNSTF * closed stiffness) is applied to the inactive contact elements that have a gap value less than or equal to OPNTOL * characteristic length, but greater or equal than GAPTOL * characteristic length. The contact elements with a gap greater than OPNTOL * characteristic length remain inactive. While sliding is permitted with this option, the magnitude of the sliding can be controlled by the tangential penalty factor. To define frictionless sliding, set the coefficient of friction=0.0 or tangential penalty factor (PENT)=0.0. (Default=0)
GUPDATE	Geometry update flag. (Default=0 when PARAM,LGDISP,-1; Default=2 for PARAM,LGDISP,1; INTEGER) 0 – Contact geometry updates will not be done during the analysis. 1 – Contact geometry updates occur when SLIP > (GUPTOL * Average element length). 2 - Contact geometry updates occur at the start of each step and when SLIP > (GUPTOL * Average element length). 3 - Contact geometry updates occur once a step. 4 - Contact geometry updates occur every iteration.
GUPTOL*	Geometry update tolerance. If the relative sliding distance between the source and target regions exceeds this tolerance, a geometry update will be initiated with large displacement. (Default = 0.25 * characteristic length; REAL>0.0)

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
FRICMOD	Tangential stiffness selection. 0 - The tangential stiffness if fixed throughout the subcase. 1 - The tangential stiffness is adaptively modlfied at every iteration as a function of contact pressure. (Default) The tangential stiffness is then computed as: (FRICi * contact pressure) / SCRIT where FRICi is the coefficient of friction defined on the BCTSET entry, and SCRIT is the critical slip parameter.
SCRIT*	Defines the critical slip when FRICMOD=1, where the critical slip = SCRIT * characteristic length for each pair. (Default=0.005; REAL) A negative SCRIT value is treated as an absolute value and is not scaled by the characteristic length.
DISCAL	Displacement scaling option. 0 – No scaling will be done. 1 – Scaling will be done if required during every iteration. A check will be performed after every displacement increment to see if the incremental displacements would cause penetration between the source and target regions. If the penetrations exceed DISTOL, the entire incremental displacements will be scaled back to limit the penetrations in the model. (Default)
DISTOL	Tolerance for displacement scaling feature. (Default = 1.0* characteristic length)
KSTAB	Stiffness stabilization for contact. 0 – Stiffness stabilization is off. (Default) 1 – The stiffness matrix is stabilized when it is singular due to inactive contact constraints. The stabilization adds a factor (1.0) to the diagonal terms of the stiffness matrix. KSTAB=1 is only supported with the sparse solver, and will disable any open contact stiffness specified through the OPNSTF parameter.

Table 10-5. Primary parameters supported by SOL 401:

Name	Description
AUTOSCAL*	When PENN is not defined explicitly, scales the automatically calculated normal penalty factor PENN either up or down. AUTOSCAL can be used to scale the stiffness of specific contact pairs if convergence issues occur (Real>0.0; Default=1.0). See Remark 2.
TANSCL*	When PENT is not defined explicitly, scales the automatically calculated tangential penalty factor PENT either up or down. TANSCL can be used to scale the stiffness of specific contact pairs if convergence issues occur (Real>0.0; Default=1.0). See Remark 2.
FRICDLY *	Option to delay contact friction to help alleviate convergence problems. 0 – Friction is not delayed. (Default) 1 – Friction is delayed. It is not included in the solution until the second contact iteration.

Table 10-6. Secondary parameters supported by SOLs 101, 103, 111, and 112:

The following parameters are available for special cases.

Name	Description
PENN*	Penalty factor for normal direction. PENN and PENT are automatically calculated by default. See Remark 2. When PENT is defined but PENN is undefined, PENN = 10 * PENT.
PENT*	Penalty factor for transverse direction. PENN and PENT are automatically calculated by default. See Remark 2. When PENN is defined but PENT is undefined, PENT = PENN / 10.
PENTYP*	Changes how contact element stiffness is calculated (Default=1). See Remark 2. 1- PENN and PENT are entered as units of 1/Length. 2 - PENN and PENT are entered as units of Force/(Length x Area).

Table 10-6. Secondary parameters supported by SOLs 101, 103, 111, and 112:

The following parameters are available for special cases.

Name	Description
AUTOSCAL*	Scales the automatically calculated penalty factors PENN and PENT either up or down. AUTOSCAL can be used to scale the stiffness of specific contact pairs if convergence issues occur (0<Real; Default=1.0). See Remark 2.
RESET	Flag to indicate if the contact status for a specific subcase is to start from the final status of the previous subcase 0 - Starts from previous subcase. (Default) 1 - Starts from initial state.
REFINE	Requests that the software refine the mesh on the source region during the solution to be more consistent with the target side mesh. 0 - Refinement does not occur. 2 - Refinement occurs (default).
INTORD	Determines the number of contact evaluation points for a single element edge or face on the source region. The number of contact evaluation points is dependent on the value of INTORD, and on the type of element face. See the table in Remark 4 for specific values. 1 – The reduced number of contact evaluation points is used. 2 – Use an increased number of contact evaluation points (default). 3 – Use a high number of contact evaluation points.
CSTRAT	Prevents all of the contact elements from becoming inactive. See Remark 5. 0 - All contact elements can become inactive (Default). 1 - The software will reduce the likelihood of all of the contact elements becoming inactive.
PREVIEW	Requests the export of a bulk data representation of the element edges and faces where contact elements are created. See Remark 6. 0 - The bulk data export does not occur (Default). 1 - The bulk data export occurs.

Table 10-7. Secondary parameters supported by SOL 401:

The following parameters are available for special cases.

Name	Description
PENN*	Penalty factor for normal direction. PENN and PENT are automatically calculated by default. See Remark 2. When PENT is defined but PENN is undefined, PENN = 10 * PENT.
PENT*	Penalty factor for transverse direction. PENN and PENT are automatically calculated by default. See Remark 2. When PENN is defined but PENT is undefined, PENT = PENN / 1000.
PENTYP*	Changes how contact element stiffness is calculated (Default=1). See Remark 2. 1- PENN and PENT are entered as units of 1/Length. 2 - PENN and PENT are entered as units of Force/(Length x Area).
REFINE	Requests that the software refine the mesh on the source region during the solution to be more consistent with the target side mesh. 0 - Refinement does not occur. 2 - Refinement occurs (default).
INTORD	Determines the number of contact evaluation points for a single element edge or face on the source region. The number of contact evaluation points is dependent on the value of INTORD, and on the type of element face. See the table in Remark 4 for specific values. 1 – The reduced number of contact evaluation points is used. 2 – Use an increased number of contact evaluation points (default). 3 – Use a high number of contact evaluation points.

* Can be defined on local BCTPARM entries. The BCTPARM bulk entries associated to individual BCTSET bulk entries, which are then combined with a BCTADD bulk entry, define local parameters. A local parameter definition overrides a global definition.

See “Contact Control Parameters – BCTPARM” in the *NX Nastran User’s Guide* for more information on the BCTPARM options.

REMARKS:

1. In SOLs 101, 103, 111, and 112, all of the parameters are supported for surface-to-surface contact definitions. For edge-to-edge contact definitions, the parameters CTOL, MAXF, MAXS, NCHG, INIPENE, PENN, PENT, PENTYP, RESET, CSTRAT, PREVIEW, and AUTOSCAL are supported.

2. The penalty factors PENN and PENT are automatically calculated by default. The automatic calculation is turned off if either PENN or PENT are defined.

   When PENTYP=1 (default) is defined, PENN and PENT have units of 1/(Length), and the contact element stiffness is calculated by K = e*E*dA where e represents PENN or PENT, E is the modulus value, and dA is area. A physical interpretation is that it is equivalent to the axial stiffness of a rod with area dA, modulus E, and length 1/e.

   When PENTYP=2 is defined, PENN and PENT become a spring rate per area Force/(Length x Area), and the contact element stiffness is calculated as K=e*dA. The spring rate input is a more explicit way of entering contact stiffness since it is not dependent on the modulus value.

   The penalty factors influence the rate of convergence, and to a lesser extent, the accuracy of the contact solution. The automatic penalty factor calculation works well for most instances, but manual adjustments may be necessary, particularly if a contact problem fails to converge. See “Tips for Setting PENN and PENT” in the *NX Nastran User's Guide* for tips on adjusting penalty factors.

3. In SOLs 101, 103, 111, and 112, if NCHG is a real number and is < 1.0, the software treats it as a fraction of the number of active contact elements in each outer loop of the contact algorithm. The number of active contact elements is evaluated at each outer loop iteration.

   If NCHG is an integer ≥ 1, the value defines the allowable number of contact changes.

   If NCHG = 0, no contact status changes can exist.

   Consider defining a lower NCHG value than the default when a large number of pairs are defined in a “stack up” type of configuration where the cumulative effect of a small contact element status change within some of the pairs will impact the contact element status of the other pairs.

4. A higher number of contact evaluation points can be used to increase the accuracy of a contact solution. Inaccuracies sometimes appear in the form of nonuniform contact pressure and stress results. There may be a penalty associated with using more evaluation points since the time for a contact problem to converge may be longer. The table below shows how the number of contact evaluation points is dependent on the element type, and how it can be adjusted

using the INTORD option. The “Face Type” column applies to shell elements, and to the solid element with the associated face type.

	Number of Contact Evaluation Points		
Face Type	INTORD=1	INTORD=2	INTORD=3
Linear Triangle	1	3	7
Parabolic Triangle	3	7	12
Linear Quad	1	4	9
Parabolic Quad	4	9	16

5. In SOLs 101, 103, 111, and 112, under certain conditions, all of the contact elements could become inactive which may lead to singularities. Setting the parameter CSTRAT=1 will reduce the likelihood of all contact elements becoming inactive.

6. In SOLs 101, 103, 111, and 112, setting the PREVIEW parameter to “1” requests a bulk data representation of the element edges and faces where contact elements are created. The software will write a bulk data file containing dummy shell element entries for face locations, and dummy PLOTEL entries for edge locations. Dummy GRID, property and material entries are also written. You can import the file into a preprocessor to display both source and target contact locations. The preview file has the naming convention

```
<input_file_name>_cnt_preview_<subcaseid>_<contactsetid>.dat
```

7. In SOLs 101, 103, 111, and 112, the following applies to the initial contact condition* when INIPENE= 2 or 3:

   * The initial contact condition is the contact status before the solution iterates on the contact condition. This is the contact status before the solution has applied loads.

   - If shell element contact regions exist and SHLTHK=0 (default) on the BCTPARM entry, the shell element thickness is applied *before* the software evaluates any gaps or penetrations as a result of the INIPENE setting. For example, when INIPENE=2, penetrations are reset to a new initial condition in which there is no interference. This includes the penetrations as a result of the shell thickness. When INIPENE=3, since penetrations and gaps are reset to a new initial condition, the shell thickness is not considered by the contact condition.

   - If shell element contact regions exist, and a shell element offset is defined with the ZOFF field on the element entry, and ZOFFSET=0 (default) on the BCTPARM entry, the shell element offset is added *after* the software removes any gaps or penetrations as a result of the INIPENE setting.

8. If a region offset is defined with the OFFSET parameter on the BCRPARA entry, the region offset is added *after* the software removes any gaps or penetrations as a result of the INIPENE setting.

BCTSET

Contact Set Definition

Defines contact pairs of a 2D or 3D contact set (SOLs 101, 103, 105, 111, 112, 401, 601 and 701).

FORMAT:

1	2	3	4	5	6	7	8	9	10
BCTSET	CSID	SID1	TID1	FRIC1	MIND1	MAXD1			
		SID2	TID2	FRIC2	MIND2	MAXD2			
		-etc-							

SOL 601 & 701 EXAMPLE:

BCTSET	4	3	2	0.0					
		1	8	0.0					

SOL 101 EXAMPLE:

BCTSET	4	3	2	0.0	0.0	0.01			
		1	8	0.0	-0.001	0.005			

FIELDS:

Field	Contents
CSID	Contact set identification number. (Integer > 0)
SIDi	Source region (contactor) identification number for contact pair i. (Integer > 0)
TIDi	Target region identification number for contact pair i. (Integer > 0)

Field	Contents
FRICi	Static coefficient of friction for contact pair i. (Real; Default=0.0)
MINDi	Minimum search distance for contact. (Real; Default=0.0) See Remark 6. Ignored by SOLs 601 and 701.
MAXDi	Maximum search distance for contact. (Real; Default=0.0) See Remark 6. Ignored by SOLs 601 and 701.

REMARKS:

1. CSID is selected by the case control command BCSET. If BCSET is not specified, the model will not include contact.

2. The following table summarizes all supported contact conditions.

Table 10-8. Contact Summary

Type	Description	Solution Support
Edge-to-Edge	Contact between the edges of axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8).	Solution 101 and consecutive solutions 103, 105, 111, and 112 (elements in XZ or XY plane). Solution 401. Solution 601 (elements in XZ plane only).
Surface-to-Surface	Contact between shell element faces (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR) and solid element faces (CHEXA, CPYRAM, CPENTA, CTETRA).	Solution 101 and consecutive solutions 103, 105, 111, and 112. Solution 401 (solid element faces only: CHEXA, CPYRAM, CPENTA, CTETRA). Solutions 601 and 701.

3. The parameters for a contact set are defined by the BCTPARM entry.

4. SID and TID are contact regions defined via BCPROP, BCPROPS, BSURF, or BSURFS entries for 3D contact, and with BEDGE or BLSEG entry for 2D contact.

5. SIDi can be equal to TIDi to define a self-contacting pair.

6. The following rules apply to MINDi and MAXDi:

   - Both default to “0”, but at least one must be explicitly defined as non-zero.

   - MAXDi must be greater than MINDi (MINDi<MAXDi).

   - Negative values are allowed.

7. A converged SOL 101 contact condition can be included in a consecutive normal mode solution (SOL 103). The normal mode solution results, which included the SOL 101 contact conditions, can then be used in an optional dynamic response calculation (SOLs 111 and 112). See the chapter on contact in the *NX Nastran User's Guide* for more information.

8. When defining contact regions and pairs on geometry which is not tangent continuous, creating single contact regions which cross corner transitions can result in nonuniform stress results around the corners. It is recommended to break these areas into multiple regions and pairs. See the chapter on contact in the *NX Nastran User's Guide* for more information.

9. Contact definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the contact definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BCRESULTS case control command.

10. If the contact solution is having problems converging, the fields on the BCTPARM bulk data entry can be modified from their defaults. See the BCTPARM bulk data entry, and the chapter on surface contact in the NX Nastran User’s Guide for more information.

11. Contact conditions can be included in SMP and RDMODES solutions. See “Recursive Domain Normal Modes Analysis (RDMODES)” in the *Parallel Processing Guide*. Contact conditions cannot be included in a GDMODES, FDMODES, or HDMODES DMP solution.

12. Multiple contact sets (BCTSETs) with unique CSID's can be combined by including their CSID's on a BCTADD entry. The BCTADD entry has its own unique CSID which will be used in the BCSET case control command. The reason to have multiple contact sets with their own CSID is when different parameters (BCTPARA) are needed for different contact sets.

13. If multiple eigenvalue subcases (solutions 103 or 105) include STATSUB commands to select linear static subcases, and the linear static subcases include glue and/or linear contact definitions, the BCSET and/or BGSET case control

commands must be specified in the global case. Failure to do this can result in erroneous results.

14. Contact conditions can be included in a model with main bulk data type superelements, but the contact regions must be in the residual. Contact conditions are not supported in a model with partitioned type superelements. Main bulk data and partitioned type superelements are both described in Chapter 2 of the *NX Nastran Superelement User's Guide*.

15. You cannot restart from a solution which included contact conditions. You cannot add contact conditions when restarting.

16. The combination of a bolt preload in which the bolt is meshed with solid elements, the element iterative solver, and contact conditions is not supported.

REMARKS RELATED TO SOL 601 AND 701:

1. The parameters for a SOLs 601,701 contact set are defined by the BCTPARA entry.

2. If multiple BCTSET entries with the same CSID are specified, they will be combined. If duplicate contact pairs are defined, the last one defined will be used.

3. For SOL 601, contact surfaces do not support element faces which are missing midside nodes on variable node CPYRAM and CTETRA elements.

BDYOR

CHBDYi Entry Default Values

Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BDYOR	TYPE	IVIEWF	IVIEWB	RADMINF	RADMIDB		PID	GO	
	CE	E1	E2	E3					

EXAMPLE:

BDYOR	AREA4	2	2	3	3		10		

FIELDS:

Field	Contents
TYPE	Default surface type. See Remark 2 (Character).
IVIEWF	Default identification number of the front VIEW entry. (Integer > 0 or blank)
IVIEWB	Default identification number of the back VIEW entry. (Integer > 0 or blank)
RADMIDF	Default identification number of a RADM entry for the front face. (Integer ≥ 0 or blank)
RADMIDB	Default identification number of a RADM entry for the back face. (Integer ≥ 0 or blank)
PID	Default PHBDY property entry identification number. (Integer > 0 or blank)
GO	Default orientation grid point. (Integer ≥ 0; Default=0)

Field	Contents
CE	Default coordinate system for defining the orientation vector. (Integer ≥ 0 or blank)
E1, E2, E3	Default components of the orientation vector in coordinate system CE. The origin of this vector is grid point G1 on a CHBDYP entry. (Real or blank)

REMARKS:

1. Only one BDYOR entry may be specified in the Bulk Data Section.
2. TYPE specifies the type of CHBDYi element surface; allowable values are: POINT, LINE, REV, AREA3, AREA4, ELCYL, FTUBE, AREA6, AREA8, and TUBE.
3. IVIEWF and IVIEWB are specified for view factor calculations only. (See the VIEW entry).
4. GO is only used from BDYOR if neither GO nor the orientation vector is defined on the CHBDYP entry and GO is > 0.
5. E1, E2, E3 is not used if GO is defined on either the BDYOR entry or the CHBDYP entry.

REMARKS RELATED TO SOL 601:

1. Only TYPE, RADMINF and RADMINB are supported.

BEAMOR

CBEAM Entry Default Values

Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			X1	X2	X3		

EXAMPLE:

BEAMOR		39			0.6	2.9	-5.87		

ALTERNATE FORMAT AND EXAMPLE:

BEAMOR		PID			G0				

BEAMOR		39			86				

FIELDS:

Field	Contents
PID	Property identification number of the PBEAM or PBCOMP entry. (Integer > 0 or blank)
X1, X2, X3	Components of the orientation vector, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector , using grid point G0. The direction of the vector is from GA to G0. is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBEAM entry)

REMARKS:

1. The contents of fields on this entry will be assumed for any CBEAM entry with corresponding fields that are blank.
2. Only one BEAMOR entry is allowed.
3. For an explanation of beam element geometry, see the CBEAM entry description.
4. If X1 or G0 is an integer, G0 is used. If X1 or G0 is blank or real, then X1, X2, X3 is used.

BEDGE

Defines a Glue or Contact Edge Region

Defines a Glue or Contact Edge Region

FORMAT:

1	2	3	4	5	6	7	8	9	10
BEDGE	ID	EID1	GID1,1	GID1,2		EID2	GID2,1	GID2,2	
		EID3	GID3,1	GID3,2		-etc-			

EXAMPLE:

BEDGE	14	101	15	190		5	42	65	
		23	57	82		16	132	4	
		94	255	26		18	39	97	

FIELDS:

Field	Contents
ID	Edge region identification number. (Integer > 0) See Remark 1.
EIDi	Element ID. (Integer > 0)
GIDi,1	First grid point identification number for EIDi. (Integer > 0) See Remark 2.
GIDi,2	Second grid point identification number for EIDi. (Integer > 0) See Remark 2.

REMARKS:

1. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.

2. With edge-to-surface or edge-to-edge gluing:

   - With edge-to-surface gluing, a shell edge region defined with the BEDGE entry is glued to a face region defined with a BSURF, BSURFS, BCPROP, or BCPROPS entry.

   - With edge-to-edge gluing, an edge region defined with the BEDGE entry is glued to another edge region defined by a BEDGE entry or by a BLSEG entry.

   - All glue pairs are defined with the BGSET entry.

   - The grid point IDs on the BEDGE entry used to define a glue edge region must be included in the connectivity of the shell elements CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, CTRIA6, the axisymmetric elements CTRAX3, CQUADX4, CTRAX6, CQUADX8, the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, or the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8. Axisymmetric, plane stress and plane strain elements can all be paired. Edges on 3D solid elements are not supported.

3. With edge-to-edge contact:

   - Edge-to-edge contact can be defined between the edges of axisymmetric elements, plane stress elements, and plane strain elements. Shell elements are not supported. The BEDGE entry is paired to another edge region defined by a BEDGE entry or by a BLSEG entry. The edge-to-edge contact pairs are defined with the BCTSET entry.

   - The grid point IDs on the BEDGE entry used to define a contact edge region must be included in the connectivity of the axisymmetric elements CTRAX3, CQUADX4, CTRAX6, CQUADX8, of the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, or the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8. Note that shell element edges and 3D solid element edges are not supported.

4. For edge-to-edge glue (GLUETYPE=1) with heat transfer, plane stress, plane strain, axisymmetric solids, and shells are supported for ALL permutations and combinations of these elements. Note that shells in plane stress or plane strain mode will NOT be supported. Edges on solid elements are also NOT supported.

5. For parabolic elements, GIDi,1 and GIDi,2 would represent the vertex/corner grids.

6. The generalized plane strain element used in SOL 401 is not supported in a glue or contact region.

REMARKS RELATED TO SOL 601:

1. The BEDGE or BLSEG entries define a flexible or rigid 2D contact region on axisymmetric elements CQUADX4, CQUADX8, CTRAX3 and CTRAX6, plane stress elements CPLSTS3, CPLSTS4, CPLSTS6 and CPLSTS8, and plane strain elements CPLSTN3, CPLSTN4, CPLSTN6 and CPLSTN8. The BLSEG entry must be used to define a rigid target region when the grid points are not attached to any elements.

2. The element edges specified in BEDGE do not need to be in a sequence. However, when BEDGE defines a contact region, all element edges must form a contiguous 2D contact region.

3. Axisymmetric, plane stress and plane strain elements must be in the XZ plane.

4. Contact region properties are defined by BCRPARA entry and contact set properties are defined by BCTPARA entry in a similar way as for 3-D contact. In addition, global contact settings may be specified in the NXSTRAT entry.

BFLUID

Fluid Boundary Definition (SOL 601,106 only)

Defines a fluid boundary by referencing BSURFS, BCPROPS or BEDGE entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BFLUID	BID	TYPE	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc-						

EXAMPLE:

BFLUID	23	FLUID	52	21	28	4	23	19	
	5	30	32	33					

FIELDS:

Field	Contents
BID	Identification number of the fluid boundary. (Integer > 0)
TYPE	Type of fluid boundary. See remarks 2 to 6. (Character: “FLUID”, “STRUC”, “FREE”, “RIGID”)
IDn	Boundary identification numbers defined in a BSURFS, BCPROPS or BEDGE entry. (Integer > 0)

REMARKS:

1. BID must be unique with respect to all other BFLUID entries.
2. TYPE=”FLUID” defines a fluid to fluid interface, TYPE=”STRUC” defines a fluid to structure interface, TYPE=”FREE” defines a free surface interface, and TYPE=”RIGID” defines a rigid wall.

3. Fluid-structure and fluid-fluid interfaces must have separate but compatible meshes.

4. For TYPE=”FLUID”, both fluid boundary regions must be included in the definition.

5. For TYPE=”STRUC”, only the fluid boundary region need to be included in the definition. Note that the program can automatically detect fluid-structure boundaries. Hence, the specification of fluid-structure boundaries is optional.

6. A fluid boundary without any interface are treated as a rigid wall. Hence, the specification of a rigid wall boundary is optional.

BFRIC

Slideline Contact Friction

Defines frictional properties between two bodies in contact.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BFRIC	FID		FSTIF	MU1					

EXAMPLE:

BFRIC	33			0.3					

FIELDS:

Field	Contents
FID	Friction identification number. See Remark 1 (Integer > 0).
FSTIF	Frictional stiffness in stick. See Remarks 2 and 3 (Real > 0.0; Default=automatically selected by the program.)
MU1	Coefficient of static friction. (Real > 0.0)

REMARKS:

1. This identification number must be unique with respect to all other friction identification numbers. This is used in the FRICID field of the BCONP Bulk Data entry.

2. The value of frictional stiffness requires care. A method of choosing its value is to divide the expected frictional strength (MU1 * expected normal force) by a reasonable value of the relative displacement that may be allowed before slip occurs. The relative value of displacement before slip occurs must be small

compared to expected relative displacements during slip. A large stiffness value may cause poor convergence, while too small a value may cause poor accuracy.

Frictional stiffness specified by the user is selected as the initial value. If convergence difficulties are encountered during the analysis, the frictional stiffness may be reduced automatically to improve convergence.

3. The stiffness matrix for frictional slip is unsymmetrical. However, the program does not use the true unsymmetrical matrix, but uses the symmetric terms instead. This is to avoid using the unsymmetrical solver to reduce CPU time.

BGADD

Surface-to-Surface Glue Set Combination

Defines a surface-to-surface glue set as a union of glue sets defined on BGSET entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BGADD	GSID	SI	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc-						

EXAMPLE:

BGADD	12	1	3	6	5	9			

FIELDS:

Field	Contents
GSID	Glue set identification number. (Integer > 0)
Si	Identification numbers of glue sets defined via BGSET entries. (Integer > 0)

REMARKS:

1. Multiple glue sets (BGSETs) with unique GSIDs can be combined by including their GSIDs on a BGADD entry. The BGADD entry has its own unique GSID which is used on the BGSET case control command. Multiple glue sets with their own GSID are necessary when different parameters (BGPARM) are needed for different glue sets.

2. To include several glue sets defined via BGSET entries in a model, BGADD must be used to combine the glue sets. GSID on BGADD is then selected with the BGSET case control command.

3. Si must be unique and may not be the identification of this or any other BGADD entry.

4. Glue definitions are supported in all solution sequences except for SOLs 144–146 and 701. The BGADD entry is not supported by SOL 601. In a SOL 153/159 heat transfer analysis, glue definitions are treated as a constant near infinite conductivity connection.

BGPARM

Glue Parameters

Control parameters for the glue algorithm.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BGPARM	GSID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	-etc.-				

EXAMPLE:

BGPARM	4	INTORD	2	PENN	10.0	PENT	10.0		
	REFINE	0							

FIELDS:

Field	Contents
GSID	Glue set ID. Parameters defined in this command apply to glue set GSID defined by a BGSET entry. (Integer > 0)
PARAMi	Name of the BGPARM parameter. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. See below for the parameter listing. (Real or Integer)

Table 10-9. BGPARM Parameters:

Name	Description
GLUETYPE*	Selects the glue formulation for surface-to-surface glue. (Default=2) See Remark 2. Edge-to-surface and edge-to-edge glue pairs always use GLUETYPE=2. 1 - Normal and tangential springs will be used to define the connections. See Remark 7. 2 - A “weld like” connection will be used to define the connections.
PENTYP*	Changes how glue element stiffness and “conductance” are calculated. (Default=1) See Remark 2.
PENN*	Penalty factor for normal direction when GLUETYPE=1. (Default=100) See Remarks 1 and 2.
PENT*	Penalty factor for transverse direction when GLUETYPE=1. (Default=100) See Remarks 1 and 2.
PENGLUE*	Penalty factor when GLUETYPE=2. (Default=1) See Remarks 1 and 2.
INTORD	Determines the number of glue points for a single element face on the source region. Edge-to-surface glue pairs always use INTORD=3. 1 - Low order 2 - Medium order (default) 3 - High order
REFINE	Requests that the software refine the mesh on the source region during the solution to be more consistent with the target side mesh. 0 - Refinement does not occur. 2 - Refinement occurs. (default).
PREVIEW	Requests the export of a bulk data representation of the element edges and faces where glue elements are created. See Remark 6. 0 - The bulk data export does not occur (default). 1 - The bulk data export occurs.

Table 10-9. BGPARM Parameters:

Name	Description
ESOPT	Changes the edge-to-surface glue stiffness. Shell theory does not account for changes in shell thickness or normal strains perpendicular to the plane of the shell element. The ESOPT parameter gives you a choice of how to handle the linking of the zero normal strains which exist in the shell element associated with the edge, to the surface being glued. 0 - Strains in the plane of the surface being glued in the direction perpendicular to the edge are not constrained by the glue stiffness (default). 1- Strains in the plane of the surface being glued in the direction perpendicular to the edge are constrained by the glue stiffness.
SLIDE	Requests the sliding glue in SOL 401. Sliding glue includes normal stiffness but no tangential stiffness. Gaps between the glued surfaces are preserved as sliding occurs. 0 - Sliding glue is off (default). 1 - Sliding glue is on.

* Can be defined on local BGPARM entries. The BGPARM bulk entries associated to individual BGSET bulk entries, which are then combined with a BGADD bulk entry, define local parameters. A local parameter definition overrides a global definition.

REMARKS:

1. The following table summarizes the parameters and solution support for the different glue types.

Glue Type	Parameter and Solution Support
Surface-to-Surface Glue	All parameters except ESOPT are supported. Surface-to-surface glue is supported in structural solutions, in heat transfer solutions 153 and 159, and for gluing fluid surfaces to other fluid surfaces in a coupled acoustics analysis. In a coupled acoustics analysis, GLUETYPE, PENTYP, and PENT are all ignored, and PENN is described in Remark 3.

Edge-to-Surface Glue	Only the parameters PENTYPE, PENGLUE, PREVIEW, and ESOPT are supported. GLUETYPE=2 formulation is always used. Edge-to-surface glue is supported in structural solutions except solution 401, and in heat transfer solutions 153 and 159.
Edge-to-Edge Glue	Only the parameters PENTYPE, PENGLUE, and PREVIEW are supported. GLUETYPE=2 formulation is always used. Edge-to-edge glue is supported in structural solutions, and in heat transfer solutions 153 and 159.

2. GLUETYPE has unique penalty factor inputs in structural solutions and in heat transfer solutions 153 and 159. These inputs and units are described below.

GLUETYPE=2 (default for structural solutions)	The glue penalty stiffness is defined by PENGLUE. A physical interpretation of this glue connection is a beam like element. (Solutions 153 and 159 - Heat transfer analysis always uses GLUETYPE=1)	
Structural solutions:	PENTYP=1 (default)	PENGLUE is a unitless scale factor of the beam like element stiffness. The averaged modulus of the elements associated with the source side region are used when computing this stiffness.
	PENTYP=2	PENGLUE is the beam like element modulus (E) with units of Force/Area.
Solutions 153 and 159:	Always uses GLUETYPE=1.	

GLUETYPE=1	The glue penalty stiffness is defined by PENN and PENT.	
	PENTYP=1	Structural solutions: PENN and PENT have units of 1/(length), and the glue element stiffness is calculated by K = e*E*dA where e represents PENN or PENT, E is an average modulus (averaged over elements associated with the source side region), and dA is area. A physical interpretation is that it is equivalent to the stiffness of a rod with area dA, modulus E, and length 1/e.
		Solutions 153 and 159: PENT is ignored. PENN has the units of 1/(length), and “conductance” at the glue connection is calculated as $C = e*k_{avg}*dA*100$ where e represents PENN, k_{avg} is an average of the thermal conductivity (k) values defined on the MAT4 entries (averaged over elements

		associated with the source side region), and dA is area. A physical interpretation is that it is equivalent to the axial "conductance" of a rod with area dA, conductivity k_{avg}, and length 1/e.
	PENTYP=2	Structural solutions: PENN and PENT become a spring rate per area Force/(Length x Area), and the glue element stiffness is calculated as K=e*dA. The spring rate input is a more explicit way of entering glue stiffness since it is not dependent on the average modulus.
		Solutions 153 and 159: PENT is ignored. PENN has the units of (thermal conductivity*length)/area, and the “conductance” at the glue connection is calculated as C = e*dA. Another term for e is heat flux.

3. When gluing fluid surfaces to other fluid surfaces for a coupled acoustics analysis, GLUETYPE, PENTYPE, PENT are all ignored, and PENN is used to calculate the acoustics penalty matrix K:

   $$K = e * \frac{1}{\rho} * dA$$

   where e is PENN, ρ is the average density of all fluid elements in the model, and dA is the surface area. The K matrix for an acoustic element is defined by

   $$K = \frac{1}{\rho} \int_V \nabla N^T \nabla N \, dV$$

   For a small fluid column (tube of length L and cross-sectional area dA), the K matrix can be written as

   $$K = \frac{1}{L} * \frac{1}{\rho} * dA$$

   Therefore, the penalty factor PENN (e) can be interpreted as 1/L.

   Edge-to-surface glue pairs can not be used as acoustics glue connections.

4. BGPARM is not supported in SOL 601, although glue definitions (BGSET) are supported.

5. The BGPARM bulk entry is not required. When it is not present, the default values are used. At least one parameter should be defined when a BGPARM entry exists.

6. Setting the PREVIEW parameter to “1” requests a bulk data representation of the element edges and faces where glue elements are created. The software will write a bulk data file containing dummy shell element entries for face locations, and dummy PLOTEL entries for edge locations. Dummy GRID, property and material

entries are also written. You can import the file into a preprocessor to display both source and target glue locations. The preview file has the naming convention

```
<input_file_name>_glue_preview_<subcaseid>_<gluesetid>.dat
```

7. SOL 401 only supports GLUETYPE = 2. If you request GLUETYPE=1, the software will continue the solution using GLUETYPE=2.

BGSET

Glue Contact Set Definition

Defines glued contact pairs.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BGSET	GSID	SID1	TID1	SDIST1		EXT1			
		SID2	TID2	SDIST2		EXT2			
		-etc-							

EXAMPLE:

BGSET	4	1	2						
		4	3						

FIELDS:

Field	Contents
GSID	Glue set identification number. (Integer > 0)
SIDi	Source region identification number for glue pair i. (Integer > 0)
TIDi	Target region identification number for glue pair i. (Integer > 0)
SDISTi	Search distance for glue regions (Real); (Default=10.0)
EXTi	Extension factor for target region (SOL 601 only). The extension value is equal to the extension factor multiplied by element edge length. (0.0 < Real < 0.25) (Default = 0.01)

REMARKS:

1. SDIST is used to glue the closest elements on the source and target regions. The software searches from the source region.

   From edge regions and solid face regions, the software searches using SDIST in the outward normal direction. In addition, the software searches a small distance in the inward normal direction in order to glue faces which may interfere due to meshing irregularities.

   From shell element face regions, the software searches using SDIST in both the top and bottom normal directions.

2. The following table summarizes all supported glue conditions.

Table 10-10. Glue Summary

Type	Description	Solution Support
Edge-to-Edge	Glue between the edges of shell, axisymmetric, plane stress, and plane strain elements. Elements being glued must be in the same plane.	All solutions except 144 -146 and 701. SOLs 401 and 601 do not support gluing of shell edges. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.
Edge-to-Surface	Glue between shell element edges and shell or solid element faces.	All solutions except 144 -146, 401, 601, and 701. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They cannot be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.

Table 10-10. Glue Summary		
Surface-to-Surface	Glue between shell or solid element faces and shell or solid element faces.	All solutions except for SOL 144–146 and 701. SOLs 401 and 601 do not support gluing of shell faces. In solutions 153 and 159 heat transfer analysis, they are treated as constant conductivity connections. They can be used to represent fluid-to-fluid glue connections in a coupled acoustic analysis.

3. Shell offsets are not accounted for in any glue definitions. The gluing occurs at the grid locations and not at the offset location.

4. When defining glue regions and pairs on geometry which is not tangent continuous, creating single glue regions which cross corner transitions can result in non-uniform stress results around the corners. It is recommended to break these regions into multiple regions and pairs. See the chapter on constraints in the *NX Nastran User's Guide* for more information on glue definitions.

5. Glue definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the glue definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BGRESULTS case control command.

6. SID and TID are glue regions defined via BCPROP, BCPROPS, BSURF, or BSURFS entries for glue surfaces and via BLSEG entry for glue edges.

7. With edge-to-surface gluing, an edge region defined with the BLSEG entry is glued to a face region defined with a BSURF, BSURFS, BCPROP, or BCPROPS entry. The edge or the face region can be defined as either the source (SID) or the target (TID).

8. If multiple eigenvalue subcases (solutions 103 or 105) include STATSUB commands to select linear static subcases, and the linear static subcases include glue and/or linear contact definitions, the BCSET and/or BGSET case control commands must be specified in the global case. Failure to do this can result in erroneous results.

9. Glue conditions can be included in a model with main bulk data type superelements. Each glue pair must be contained in a single superelement or in

the residual. That is, the glue regions cannot cross superelement boundaries. Glue conditions are not supported in a model with partitioned type superelements. Main bulk data and partitioned type superelements are both described in Chapter 2 of the *NX Nastran Superelement User's Guide*.

10. When gluing solid element faces, a glue condition is not created if the dot product of the element face normals is greater or equal to 0. Including SYSTEM(516)=1 will override this and allow the glue condition to be created if the search distance is satisfied.

11. You cannot add glue conditions when restarting. You can restart from a solution which included glue conditions, but only to request addition output.

12. Glue definitions model linear glue behavior in solution 106. The glue condition does not update in regard to geometry updates, nonlinear material changes, or orientation changes. A glue definition should only be defined in portions of the structure that exhibit linear behavior.

REMARKS RELATED TO SOL 601:

1. SDISTi is not used by SOL 601. SOL 601 uses a search distance that is equal to the largest element edge in the source and target region.

2. BGSET can only be used to glue 2D solid elements or 3D solid elements. For 2D solid elements, source and target regions must be defined by the BEDGE entry. For 3D solid elements, source and target regions must be defined by BSURFS or BCPROPS entry.

   To glue shell element faces, contact can be defined with the TIED=1 option on BCTPARA. Note that TIED=1 is only valid for small rotations. See BCTSET and BCTPARA.

3. If any face of a variable noded solid element is selected as part of a glue surface, that face cannot have any missing mid-side nodes.

BLSEG

Defines Glue or Contact Edge Region, or Curve for Slideline Contact

Defines a glue or contact edge region or a curve for slideline contact via grid numbers.

FORMAT 1: (FORMATS 1 AND 2 CANNOT BE COMBINED ON THE SAME LINE)

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	

FORMAT 2:

BLSEG	ID	G1	“THRU”	G2	“BY”	INC			

CONTINUATION FORMAT 1: (CONTINUATION FORMATS 1 AND 2 CANNOT BE COMBINED ON A SINGLE CONTINUATION LINE)

	G8	G9	G10	G11	–etc.–				

CONTINUATION FORMAT 2:

	G8	“THRU”	G9	“BY”	INC				

EXAMPLE:

BLSEG	14	101	THRU	190	BY	5			
	46	23	57	82	9	16			
	201	THRU	255						
	93	94	95	97					

FIELDS:

Field	Contents
ID	Edge region or curve identification number. See Remark 1 (Integer > 0).
Gi	Grid point identification numbers in a continuous topological order. See Remark 2 (Integer > 0).
INC	Grid point identification number increment. See Remark 3 for default (Integer or blank).

REMARKS:

1. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.

2. An edge region or curve consists of one or more line segments defined between consecutive grid points. The grid points defining the edge region or curve must be entered in a continuous topological order. If an edge region or curve forms a closed loop, for example, the grids around the perimeter of a cylinder edge, the last grid point identification number should be the same as the first grid point number.

3. When selecting grid points in a range using “THRU”, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing.

4. With edge-to-surface gluing:

   - With edge-to-surface gluing, an edge region defined with the BLSEG entry is glued to a face region defined with a BSURF, BSURFS, BCPROP, or BCPROPS entry. The edge-to-surface glue pairs are defined with the BGSET entry.

   - The grid point IDs on the BLSEG entry used to define a glue edge region can only be part of the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR and CTRIA6 element connectivity.

   - The BWIDTH bulk entry is ignored.

5. With slideline contact:

- A corresponding BWIDTH Bulk Data entry may be required to define the width/thickness of each line segment. If the corresponding BWIDTH is not present, the width/thickness for each line segment is assumed to be unity.

- Each line segment has a width in a 3-D slideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width/thickness of each line segment is defined via the BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWIDTH. That is, there must be a one to one correspondence between BLSEG and BWIDTH. BWIDTH Bulk Data entry may be omitted only if the width/thickness of each segment is unity.

- The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the 'CID' field of the BCONP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.

REMARKS RELATED TO SOL 601 EDGE CONTACT:

1. BLSEG defines a flexible or rigid 2D contact region on axisymmetric elements CQUADX4, CQUADX8, CTRAX3 and CTRAX6, plane stress elements CPLSTS3, CPLSTS4, CPLSTS6 and CPLSTS8, plane strain elements CPLSTN3, CPLSTN4, CPLSTN6 and CPLSTN8, or a rigid 2D contact target region when the grid points are not attached to any elements.

2. The grid points in a BLSEG entry must either be all attached to elements or all not attached to elements.

3. For a rigid target region, it is important to note that the top surface is on the left side of the line from Gi to Gi+1. By default, contact is expected to occur from the top surface. SURF='BOT' in BCRPARA entry may be used to change the contact side.

4. Grid points in BLSEG entry must lie in the basic XZ plane.

5. The BWIDTH bulk entry is ignored.

6. Contact set pairs are defined by BCTSET entry instead of BCONP entry.

7. Contact region properties are defined by BCRPARA entry and contact set properties are defined by BCTPARA entry in a similar way as for 3-D contact. In addition, global contact settings may be specified in the NXSTRAT entry.

BNDFIX

Fixed Boundary Degrees of Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BNDFIX	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

BNDFIX	2	135	14	6					

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the integers 1 through 6 for grid points. No embedded blanks.)

REMARKS:

1. BSET and BNDFIX entries are equivalent.
2. If there are no BNDFREEi or BNDFIXi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFREEi or BNDFIXi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFIX1

Fixed Boundary Degrees of Freedom, Alternate Form of BNDFIX Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BNDFIX1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

EXAMPLE:

BNDFIX1	2	135	14	6	23	24	25	26	
	122	127							

ALTERNATE FORMAT AND EXAMPLE:

BNDFIX1	C	ID1	“THRU”	ID2					
BNDFIX1	3	6	THRU	32					

FIELDS:

Field	Contents
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1< ID2)

REMARKS:

1. BSET1 and BNDFIX1 entries are equivalent.

2. If there are no BNDFREEi or BNDFIXi entries present, all a-set points are considered fixed during component mode analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFREEi or BNDFIXi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFREE

Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BNDFREE	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

BNDFREE	124	1	5	23	6	16			

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. CSET and BNDFREE entries are equivalent.
2. If there are no BNDFREEi or BNDFIXi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFREEi or BNDFIXi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFREE1

Free Boundary Degrees-of-Freedom, Alternate Form of BNDFREE Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BNDFREE1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

EXAMPLE:

BNDFREE1	124	1	5	7	6	9	12	122	
	127								

ALTERNATE FORMATS AND EXAMPLE:

BNDFREE1	C	ID1	“THRU”	ID2					
BNDFREE1	3	6	THRU	32					

BNDFREE1		“ALL”							
BNDFREE1		ALL							

FIELDS:

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)

REMARKS:

1. CSET1 and BNDFREE1 entries are equivalent.

2. If there are no BNDFREEi or BNDFIXi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFREEi or BNDFIXi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.

4. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDGRID

Boundary Grid Points

Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).

FORMAT:

1	2	3	4	5	6	7	8	9	10
BNDGRID	C	GP1	GP2	GP3	GP4	GP5	GP6	GP7	
	GP8	-etc.-							

EXAMPLE:

BNDGRID	123	41	42	43	44	45	46	47	
	49								

ALTERNATE FORMAT AND EXAMPLE:

BNDGRID	C	GP1	“THRU”	GP2					
BNDGRID	123	41	THRU	49					

FIELDS:

Field	Contents
C	Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1 .
GPi	Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)

REMARKS:

1. C specifies the components for the listed grid points for which boundary motion is prescribed.

2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.

3. Both fixed and free shape boundary grid point identification numbers are listed on this entry.

4. The degrees-of-freedom specified on BNDGRID entries must be sufficient to statically constrain the model.

5. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

BOLT

Bolt definition

Selects the elements to be included in the bolt preload calculation.

FORMAT FOR ETYPE = 1:

1	2	3	4	5	6	7	8	9	10
BOLT	BID	ETYPE	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	“THRU”	EID8	“BY”	INC				
	-etc-								

FORMAT FOR ETYPE = 2:

1	2	3	4	5	6	7	8	9	10
BOLT	BID	ETYPE	CSID	IDIR	G1	G2	G3	G4	
	G5	“THRU”	G6	“BY”	INC				
	-etc-								

FORMAT FOR ETYPE = 3:

1	2	3	4	5	6	7	8	9	10
BOLT	BID	ETYPE	CSID	IDIR	GP				
	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8	
	EID9	“THRU”	EID10	“BY”	INC				
	-etc-								

EXAMPLES:

ETYPE = 1 for SOLs 101, 103, 105, 107 through 112

BOLT	4	1	11						

ETYPE = 1 for SOL 601

BOLT	4	1	11	8	2	1	20	14	
	15	16	28	30	33				

ETYPE = 2

BOLT	8	2	4	3	12	23	55	128	
	133	THRU	165						

ETYPE = 3

BOLT	9	3	1	3	148				
	56	24	43	21	73	52	62	41	
	106	THRU	202						

FIELDS:

Field	Contents
BID	Bolt identification number. (Integer > 0)
ETYPE	Element type. (Integer; No default)
	= 1 to model bolts with CBAR and CBEAM elements in SOLs 101, 103, 105, 107 through 112, and 601.
	= 2 to model bolts with CHEXA, CPENTA and CTETRA elements in SOLs 101, 103, 105, 107 through 112.
	= 3 to model bolts with CHEXA, CPENTA, CTETRA, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 elements in SOL 401, or CHEXA, CPENTA, CPYRAM, and CTETRA elements in SOL 601.
“BY”	Specifies an increment when using THRU option. (Character)

Field	Contents
INC	Increment used with THRU option. (Integer; Default = 1) INC > 0 can be defined, for example ...106,THRU,126,BY,INC,2 INC < 0 can be defined, for example ...126,THRU,106,BY,INC,-2

FIELDS FOR ETYPE = 1:

Field	Contents
EIDi	Selects element identification numbers to include in the bolt preload calculation. See Remark 2 and SOL 601 Remark 1. (Integer > 0, or using “THRU”; EID7 < EID8 for THRU option; No default)

FIELDS FOR ETYPE = 2:

Field	Contents
CSID	Identification number of the coordinate system used to define the bolt axis. For the basic coordinate system, CSID = 0. (Integer ≥ 0; Default = 0)
IDIR	Direction of bolt axis relative to CSID. (Integer; No default) = 1 for the X direction = 2 for the Y direction = 3 for the Z direction
Gi	Identification numbers of grid points that form a cross section through the bolt. See Remarks 3 and 4. (Integer ≥ 0; No default)

FIELDS FOR ETYPE = 3:

Field	Contents
EIDi	Selects element identification numbers to include in the bolt preload calculation. All elements representing the bolt must be included in EIDi, and have the same PID. (Integer > 0, or using “THRU”; EID7 < EID8 for THRU option; No default)
CSID	Identification number of the coordinate system used to define the bolt axis. For the basic coordinate system, CSID = 0. (Integer ≥ 0; Default = 0)
IDIR	Direction of bolt axis relative to CSID. (Integer; No default for SOL 401; Default = 0 for SOL 601) = 0 for automatic determination of the bolt axis in SOL 601. IDIR = 0 is not applicable to SOL 401. See SOL 601 Remark 2. = 1 for the X direction = 2 for the Y direction = 3 for the Z direction
GP	For SOL 401, the identification number of the grid point where the bolt cross sectional area is calculated. See Remarks 3 and 5. (Integer > 0; No default) For SOL 601, the identification number of the grid point where the bolt is split. See Remark 3 and SOL 601 Remarks 3 and 4. (Integer ≥ 0 or blank)

REMARKS:

1. Each BOLT entry defines a single physical bolt which can be composed of multiple elements.
2. If multiple CBAR and CBEAM elements are used to model a bolt in SOL 101, 103, 105, 107 through 112, only one of the elements must be listed on the BOLT entry. Enter the element ID in the EID1 field.
3. Any grid point listed in the Gi or GP fields must be included in the connectivity of elements that are used to model the bolt.

4. Gi must select enough GRID entries to define a cross section through the bolt. The selected Gi can only be included on a single BOLT entry. The grids can be listed in any order on the BOLT entry. For parabolic elements, mid-nodes must also be listed. Gi on the BOLT entry. Any Gi listed on a BOLT entry cannot be used in the connectivity of a solid composite element or be included in an SPC.

   The software splits the bolt mesh by duplicating each Gi on the BOLT entry. The identification numbers for the duplicated grid points start at the highest user-defined grid ID in the model plus one and continue sequentially higher.

   A pressure load cannot be applied to any face of an element if the connectivity of the element includes a Gi listed on a BOLT entry.

5. As a best practice, select GP such that it is near the middle of the cross section of the bolt.

6. In SOL 105, both the bolt preload and service load are scaled to determine the buckling load.

7. In SOL 401, composite solid elements (PCOMPS property card) are not supported for preloaded bolts.

REMARKS RELATED TO SOL 601:

1. All CBAR and CBEAM elements used to model the bolt must be included in EIDi list and they all must have the same PID.

2. When IDIR = 0 or blank (default), the direction of the bolt axis is automatically determined by the software to coincide with minimum principal moment of inertia of the bolt. CSID is ignored when IDIR = 0 or blank.

3. The software splits the bolt mesh at the grid point entered in the GP field.

4. GP = 0 or blank is allowed only if IDIR = 0 or blank. In this case, the location of the bolt plane is automatically determined by the program.

BOLTFOR

Preload Force on Set of Bolts

Defines preload force for a set of bolts.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BOLTFOR	SID	LOAD	B1	B2	B3	B4	B5	B6	
	B7	THRU	B8						
	B9	B10	-etc-						

EXAMPLE:

BOLTFOR	4	1000.0	12	THRU	21				
	1	4	6	9	10	26	32	34	
	37	43	51						

FIELDS:

Field	Contents
SID	Set identification number (SID) of BOLTFOR bulk entry. (Integer>0)
LOAD	Magnitude of the preload force. (Real)
Bi	Bolt identification numbers (BID) defined by BOLT bulk entries. (Integer>0 or use “THRU” option. For “THRU” option, B7<B8)

REMARKS:

1. The following rules apply when BOLTFOR, BOLTFRC, BOLTLD, or BOLTSEQ bulk entries are defined using the same SID.

- Multiple BOLTFOR and BOLTFRC entries with the same SID can be referenced by the BOLTLD case control command directly, and by the BOLTSEQ and BOLTLD bulk entries. The software will combine and apply them together.

- If a BOLTSEQ entry is referenced by the BOLTLD case control command, BOLTFOR, BOLTFRC, and BOLTLD entries with the same SID as the BOLTSEQ are only used if they are referenced in a sequence defined on the BOLTSEQ entry.

- If a BOLTLD bulk entry is referenced by the BOLTLD case control command or by the BOLTSEQ bulk entry, BOLTFOR and BOLTFRC entries with the same SID as the BOLTLD entry are used if they are referenced on the BOLTLD entry.

- Only one BOLTSEQ entry can be referenced. The software will fatal if multiple BOLTSEQ entries with the same SID are referenced.

- Only one BOLTLD entry can be referenced. The software will fatal if multiple BOLTLD entries with the same SID are referenced.

2. A BID can only be referenced once in a subcase. This includes BID refererenced on a single BOLTFOR or BOLTFRC entry, and BID on multiple BOLTFOR or BOLTFRC entries.

BOLTFRC

Defines bolt preload force, displacement, or strain in SOL 401.

Defines bolt preload force, displacement, or strain in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BOLTFRC	SID	TYPE	D	LEN					
	B1	B2	B3	B4	B5	B6	B7	B8	
	B9	THRU	B10						
	B11	B12	-etc-						

EXAMPLE:

BOLTFRC	4	STRAIN	.0025						
	1	4	6	9	10	26	32	34	
	37	43	51						

FIELDS:

Field	Contents
SID	Bolt preload set identification number (Integer; No default)
TYPE	TYPE=DISP, STRAIN, or LOAD (Character; No default)
D	Preload value. (Real; No default)
	When TYPE=DISP, D is a preload displacement.
	When TYPE=STRAIN, D is a preload strain.
	When TYPE=LOAD, D is a preload force.

Field	Contents
LEN	Optional bolt length when TYPE=DISP. (Real; Software automatically computes the length if LEN is undefined.)
Bi	Bolt identification numbers (BID) defined by BOLT bulk entries. (Integer>0 or use “THRU” option. For “THRU” option, B7<B8)

REMARKS:

1. BOLTFRC is only supported in SOL 401.

2. A bolt preload force equal to 0.0 is supported. The zero preload is useful, for example, to predict the residual axial bolt strain after applying a bolt preload, applying service loads, then finally unloading the bolt to the zero axial force condition.

3. The following rules apply when BOLTFOR, BOLTFRC, BOLTLD, or BOLTSEQ bulk entries are defined using the same SID in SOL 401.

   - Multiple BOLTFOR and BOLTFRC entries with the same SID can be referenced by the BOLTLD case control command directly, and by the BOLTSEQ and BOLTLD bulk entries. The software will combine and apply them together.

   - If a BOLTSEQ entry is referenced by the BOLTLD case control command, BOLTFOR, BOLTFRC, and BOLTLD entries with the same SID as the BOLTSEQ are only used if they are referenced in a sequence defined on the BOLTSEQ entry.

   - If a BOLTLD bulk entry is referenced by the BOLTLD case control command or by the BOLTSEQ bulk entry, BOLTFOR and BOLTFRC entries with the same SID as the BOLTLD entry are used if they are referenced on the BOLTLD entry.

   - Only one BOLTSEQ entry can be referenced. The software will fatal if multiple BOLTSEQ entries with the same SID are referenced.

   - Only one BOLTLD entry can be referenced. The software will fatal if multiple BOLTLD entries with the same SID are referenced.

4. A BID can only be referenced once in a subcase. This includes BID refererenced on a single BOLTFOR or BOLTFRC entry, and BID on multiple BOLTFOR or BOLTFRC entries.

5. You can define a bolt preload as an axial load, displacement, or strain. For any of these definitions, the result is an axial preload strain in which the software will store, and reapply in consecutive preload steps in a sequence, and in consecutive sequentially dependent subcases. The software continues to reapply the axial preload strain until you either define a new bolt preload step or subcase which redefines the axial preload strain, or you define a nonsequentially dependent subcase.

   Note that in consecutive steps and subcases, the total axial strain on a preloaded bolt includes the axial preload strain, along with strain resulting from any other loads you apply, including preloads on neighboring bolts.

6. When a bolt preload is defined as a load, the software computed axial preload strain includes the effects of any other loads you included in the preload step (thermal loads, contact forces, preloaded bolts from a previous preload step, and conditions from a previous subcase). For example, if you compare the results from a solution with only bolt preloads defined with the results from the same model with the addition of thermal loads and contact, the bolt force will be the same, but the final bolt strain needed to achieve the bolt force will be different.

BOLTLD (bulk entry)

Bolt Preload Combining and Scaling

Combines and optionally scales bolt preload sets defined with the BOLTFOR and BOLTFRC bulk entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BOLTLD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc-						

EXAMPLE:

BOLTLD	101	0.5	1.0	3	6.2	4			

FIELDS:

Field	Contents
SID	Set identification number (SID) of BOLTLD bulk entry. (Integer>0)
S	Overall scale factor. (Real)
Si	Scale factor for individual Li. (Real)
Li	SID of BOLTFOR or BOLTFRC bulk entries. (Integer>0)

REMARKS:

1. Bolt preload is supported in SOLs 101, 103, 105, 107 through 112, 401, and 601.

2. The applied preload value P_i for each BOLT bulk entry referenced by BOLTFOR bulk entry Li is given by:

   $P_i = S * Si * P_{Li}$

   where P_{Li} is the preload value defined for BOLTFOR bulk entry Li, Si is the individual scale factor corresponding to Li, and S is the overall scale factor.

3. A BID can only be referenced once in a subcase. This includes BID refererenced on a single BOLTFOR or BOLTFRC entry, and BID on multiple BOLTFOR or BOLTFRC entries.

4. The following rules apply when BOLTFOR, BOLTFRC, BOLTLD, or BOLTSEQ bulk entries are defined using the same SID.

   - Multiple BOLTFOR and BOLTFRC entries with the same SID can be referenced by the BOLTLD case control command directly, and by the BOLTSEQ and BOLTLD bulk entries. The software will combine and apply them together.

   - If a BOLTSEQ entry is referenced by the BOLTLD case control command, BOLTFOR, BOLTFRC, and BOLTLD entries with the same SID as the BOLTSEQ are only used if they are referenced in a sequence defined on the BOLTSEQ entry.

   - If a BOLTLD bulk entry is referenced by the BOLTLD case control command or by the BOLTSEQ bulk entry, BOLTFOR and BOLTFRC entries with the same SID as the BOLTLD entry are used if they are referenced on the BOLTLD entry.

   - Only one BOLTSEQ entry can be referenced. The software will fatal if multiple BOLTSEQ entries with the same SID are referenced.

   - Only one BOLTLD entry can be referenced. The software will fatal if multiple BOLTLD entries with the same SID are referenced.

5. Li must be unique. However, a single Li on a BOLTLD bulk entry can reference multiple BOLTFOR and BOLTFRC bulk entries. When this occurs, all the preload forces defined by these entries sharing the same SID are scaled by the corresponding individual scale factor Si and the overall scale factor S.

6. The bolt identification numbers (BID) in all BOLTFOR and BOLTFRC bulk entries selected by a single BOLTLD bulk entry must be unique.

7. The SID of each BOLTLD bulk entry must be unique.

8. Inertia relief (PARAM,INREL) is not supported with bolt preload.

9. The combination of a bolt preload in which the bolt is meshed with solid elements, the element iterative solver, and contact conditions is not supported.

10. Enforced displacements defined with non-zero SPC or SPCD bulk entries can be included with a bolt preload. When an SPCD is defined with a bolt preload, system cell 581 must be set to 1 to override an error trap that exists in the software. The software behaviour changes during the bolt preload solution steps for various input scenarios (with and without contact defined; enforced displacements defined with SPCD entries or with SPC entries). See “Enforced Displacements and Bolt Preloads” in the *User’s Guide* for a description of the software behaviour.

BOLTSEQ

Specifies a bolt preload sequence in SOL 401.

Specifies a bolt preload sequence.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BOLTSEQ	SID								
	S_NOi	B_IDi	NINCi						

EXAMPLES:

BOLTSEQ	100								
	1	11	3						
	2	22	2						

FIELDS:

Field	Contents
SID	Bolt preload set identification number (Integer; No default)
S_NOi	Sequence order number for the BOLTLD, BOLTFOR, and BOLTFRC ID's to be applied. (Integer; No default)
B_IDi	SID of BOLTLD, BOLTFOR, or BOLTFRC bulk entries defining a bolt preload. (Integer; No default)
NINCi	Number of increments in which to ramp up the bolt preload defined in BOLTLD, BOLTFOR, or BOLTFRC entries. (Integer; Default = 1)

REMARKS:

1. BOLTSEQ is only supported in SOL 401.
2. A bolt preload force equal to 0.0 is supported. The zero preload is useful, for example, to predict the residual axial bolt strain after applying a bolt preload, applying service loads, then finally unloading the bolt to the zero axial force condition.
3. The S_NO values must be sequential.
4. If NINCi is defined on the BOLTSEQ entry, that value overrides the value defined on the TSTEP1 increment.
5. The parameter MSGLVLB on the NLCNTL bulk entry controls the printed bolt preload output to the .f06 file.
6. The following rules apply when BOLTFOR, BOLTFRC, BOLTLD, or BOLTSEQ bulk entries are defined using the same SID in SOL 401.
   - Multiple BOLTFOR and BOLTFRC entries with the same SID can be referenced by the BOLTLD case control command directly, and by the BOLTSEQ and BOLTLD bulk entries. The software will combine and apply them together.
   - If a BOLTSEQ entry is referenced by the BOLTLD case control command, BOLTFOR, BOLTFRC, and BOLTLD entries with the same SID as the BOLTSEQ are only used if they are referenced in a sequence defined on the BOLTSEQ entry.
   - If a BOLTLD bulk entry is referenced by the BOLTLD case control command or by the BOLTSEQ bulk entry, BOLTFOR and BOLTFRC entries with the same SID as the BOLTLD entry are used if they are referenced on the BOLTLD entry.
   - Only one BOLTSEQ entry can be referenced. The software will fatal if multiple BOLTSEQ entries with the same SID are referenced.
   - Only one BOLTLD entry can be referenced. The software will fatal if multiple BOLTLD entries with the same SID are referenced.
7. You can define a bolt preload as an axial force, displacement, or strain on the BOLTFRC bulk entry. For any of these definitions, the result is an axial preload strain in which the software will store, and reapply in consecutive preload steps in a sequence, and in consecutive sequentially dependent subcases. The software continues to reapply the axial preload strain until you either define a new bolt preload step or subcase which redefines the axial preload strain, or you define a nonsequentially dependent subcase.

Note that in consecutive steps and subcases, the total axial strain on a preloaded bolt includes the axial preload strain, along with strain resulting from any other loads you apply, including preloads on neighboring bolts.

8. When a bolt preload is defined as a force, the software computed axial preload strain includes the effects of any other loads you included in the preload step (thermal loads, contact forces, preloaded bolts from a previous preload step, and conditions from a previous subcase). For example, if you compare the results from a solution with only bolt preloads defined with the results from the same model with the addition of thermal loads and contact, the bolt force will be the same, but the final bolt strain needed to achieve the bolt force will be different.

BOUTPUT

Output for Slideline Contact

Defines slave nodes at which output is requested.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BOUTPUT	ID	G1	G2	G3	G4	G5	G6	G7	

ALTERNATE FORMAT:

BOUTPUT	ID	G1	“THRU”	G2	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

CONTINUATION ENTRY FORMAT 1:

	G8	G9	G10	G11	-etc.-				

CONTINUATION ENTRY FORMAT 2:

	G8	“THRU”	G9	“BY”	INC				

EXAMPLE:

BOUTPUT	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

BOUTPUT

FORMAT AND EXAMPLE USING “ALL” (NO CONTINUATION ENTRY IS ALLOWED):

BOUTPUT	ID	ALL							
BOUTPUT	15	ALL							

FIELDS:

Field	Contents
ID	Contact region identification number of a BCONP entry for which output is desired. (Integer > 0)
Gi	Slave node numbers for which output is desired. (Integer > 0)
INC	Grid point identification number increment. See Remark (Integer or blank).

REMARKS:

For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).

BSET

Fixed Boundary Degrees of Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

BSET	2	135	14	6					

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the integers 1 through 6 for grid points. No embedded blanks.)

REMARKS:

1. BSET and BNDFIX entries are equivalent.
2. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees-of-freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSET1

Fixed Boundary Degrees of Freedom, Alternate Form of BSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

EXAMPLE:

BSET1	2	135	14	6	23	24	25	26	
	122	127							

ALTERNATE FORMAT AND EXAMPLE:

BSET1	C	ID1	“THRU”	ID2					
BSET1	3	6	THRU	32					

FIELDS:

Field	Contents
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1< ID2)

REMARKS:

1. BSET1 and BNDFIX1 entries are equivalent.

2. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees-of-freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSURF

Contact or Glue Region Definition by Shell Elements

Defines a contact or glue region by shell element IDs.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BSURF	ID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	
	EID8	EID9	EID10	-etc-					

ALTERNATE FORMAT:

BSURF	ID	EID1	THRU	EID2	BY	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

CONTINUATION ENTRY FORMAT 1:

	EID8	EID9	EID10	EID11	-etc.-				

CONTINUATION ENTRY FORMAT 2:

	EID8	“THRU”	EID9	“BY”	INC				

EXAMPLE:

BSURF	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						

BSURF

	67	68	70	85	92				

FIELDS:

Field	Contents
ID	Identification number of a contact or glue region. See Remarks 2 and 3. (Integer > 0)
EIDi	Element identification numbers of shell elements (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries). (Integer > 0)
INC	Identification number increment. See Remark 4 (Integer or blank).

REMARKS:

1. The continuation field is optional.
2. BSURF is a collection of one or more shell elements (CTRIA3, CTRIA6, CTRIAR, CQUAD4,CQUAD8, CQUADR). BSURF defines a contact or glue region which may act as a source or target.
3. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.
4. For automatic generation of element ids, the default increment value is 1 if element numbers are increasing or -1 if element numbers are decreasing (i.e., the user need not specify BY and the increment value).
5. Understanding shell element normals, and making sure those in the same regions are consistent is very important to ensuring that contact elements will be created as expected. See the chapter on contact in the NX Nastran User's Guide for more information.

REMARKS RELATED TO SOLS 601 AND 701:

1. ID must be less than 100000. Otherwise, the program will issue an error.

BSURFS

Contact or Glue Region Definition by Solid Elements

Defines a contact or glue region by the faces of the CHEXA, CPENTA, CPYRAM, or CTETRA elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BSURFS	ID				EID1	G1	G2	G3	
	EID2	G1	G2	G3	EID3	G1	G2	G3	
	-etc-								

EXAMPLE:

BSURFS	7				10	15	16	20	
	15	16	17	21	20	17	18	22	
	11	19	20	24	16	20	21	25	
	21	21	22	26					

FIELDS:

Field	Contents
ID	Identification number of a contact or glue region. See Remarks 2 and 3. (Integer > 0)
EIDi	Element identification numbers of solid elements (CHEXA, CPENTA, CPYRAM, or CTETRA entries). (Integer > 0)
G1 - G3	Identification numbers of 3 corner grid points on the face (triangular or quadrilateral) of the solid element. (Integer > 0)

REMARKS:

1. The continuation field is optional.
2. BSURFS is a collection of one or more element faces on solid elements. BSURFS defines a contact or glue region which may act as a source or target.
3. The ID must be unique with respect to all other BEDGE, BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.
4. The collapsed CHEXA element used in a SOL 401 crack simulation is not supported in a glue or contact region.

REMARKS RELATED TO SOLS 601 AND 701:

1. ID must be less than 100000. Otherwise, the program will issue an error.

BWIDTH

Boundary Line Segment Width or Thickness

Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
BWIDTH	ID	W1	W2	W3	W4	W5	W6	W7	

ALTERNATE FORMAT:

BWIDTH	ID	W1	“THRU”	W2	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

CONTINUATION ENTRY FORMAT 1:

	W8	W9	W10	W11	-etc.-				

CONTINUATION ENTRY FORMAT 2:

	W8	“THRU”	W9	“BY”	INC				

EXAMPLE:

BWIDTH	15	2.0	THRU	5.0	BY	1.0			
	2.0	2.0	2.0	2.0					
	35.	THRU	44.						

BWIDTH

	1.5	3.4	7.6	0.4	0.7				

FIELDS:

Field	Contents
ID	BLSEG entry identification number. (Integer > 0)
Wi	Width values for the corresponding line segments defined in the BLSEG entry. See Remark 1 (Real > 0.0).
INC	Width value increment. See Remark 2 (Real or blank).

REMARKS:

1. BWIDTH may be omitted if the width of each segment defined in the BLSEG entry is unity. The number of widths to be specified is equal to the number of segments defined in the corresponding BLSEG entry.

2. The default value for INC is 1.0 if the width is increasing or -1.0 if the width is decreasing. That is, the user need not specify BY and the increment value. If the number of widths specified is less than the number of segments defined in the corresponding BLSEG entry, the width for the remaining segments is assumed to be equal to the last width specified.

3. If there is only one grid point in the corresponding BLSEG entry, there is no contributory area associated with the grid point. To compute correct contact stresses, an area may be associated with a single grid point by specifying the area in field W1.

Chapter 11: Bulk Data Entries CA—CM

CAABSF

Frequency-Dependent Acoustic Absorber Element

Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAABSF	EID	PID	G1	G2	G3	G4			

EXAMPLE:

CAABSF	44	38	1	10	20				

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number that matches a PAABSF entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of fluid connection points. (Integer ≥ 0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. If only G1 is specified then a point impedance is assumed. If G1 and G2 are specified then a line impedance is assumed. If G1, G2, and G3 are specified, then an impedance is associated with the area of the triangular face. If G1

through G4 are specified, then an impedance is associated with the quadrilateral face. See Figure 11-1.

3. The CAABSF element must connect entirely to fluid points on the fluid-structure boundary.

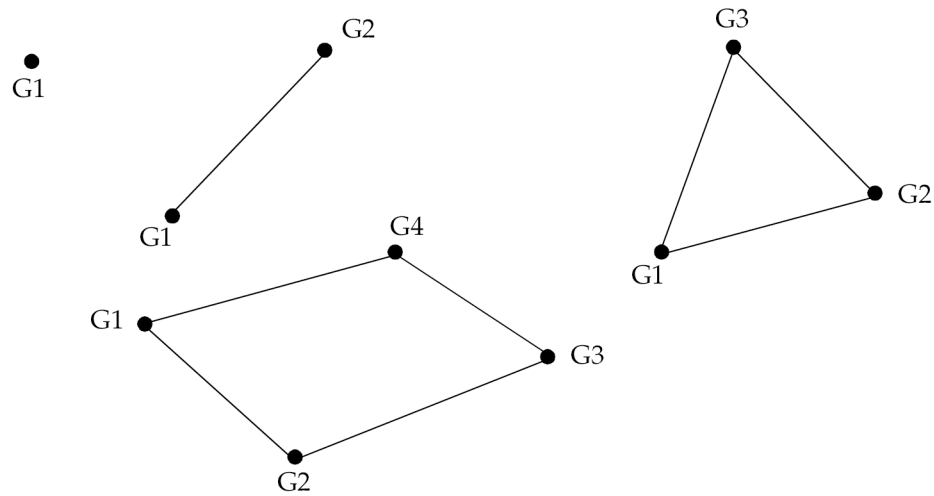


Figure 11-1. Four Types of CAABSF Elements

CAERO1

Aerodynamic Panel Element Connection

Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for the Doublet-Lattice theory for subsonic aerodynamics and, if licensed, the ZONA51 theory for supersonic aerodynamics.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAERO1	EID	PID	CP	NSPAN	NCHORD	LSPAN	LCHORD	IGID	
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

EXAMPLE:

CAERO1	1000	1		3			2	1	
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PAERO1 entry; used to specify associated bodies. Required even if there are no associated bodies. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
NSPAN	Number of spanwise boxes; if a positive value is given, NSPAN equal divisions are assumed. If zero or blank, a list of division points is given by LSPAN, field 7. (Integer ≥ 0)

Field	Contents
NCHORD	Number of chordwise boxes; if a positive value is given NCHORD, equal divisions are assumed; if zero or blank, a list of division points is given at LCHORD, field 8. (Integer ≥ 0)
LSPAN	ID of an AEFACT data entry containing a list of division points for spanwise boxes. Used only if the NSPAN field 5 is zero or blank. (Integer > 0)
LCHORD	ID of an AEFACT data entry containing a list of division points for chordwise boxes. Used only if the NCHORD, field 6, is zero or blank. (Integer > 0)
IGID	Interference group identification; aerodynamic elements with different IGIDs are uncoupled. (Integer > 0) See Remark 8 for restriction.
X1, Y1, Z1	Location of points 1 and 4 in coordinate system CP. (Real)
X4, Y4, Z4	
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0.0, but not both zero.)

REMARKS:

1. The boxes and corner point nodes are numbered sequentially, beginning with EID. The user should be careful to ensure that all box and corner point node numbers are unique. There can be overlapping ID's between the structural and aerodynamic model, but post processors may not be able to display any results. Also, non-unique corner ID's are allowed, but most post processor do not support this capability.

2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).

3. A triangular element is formed if X12 or X43=0.0.

4. The element coordinate system is right-handed as shown in Figure 11-2.

5. The continuation is required.

6. It is recommended that NCHORD or LCHORD be chosen such that the typical box chord length Δx satisfies the condition $\Delta x < 0.08 V/f$. Recent studies indicate that .02 V/f is needed to get converged stability derivatives. Where V is the minimum velocity and f, in hertz, is the maximum frequency to be analyzed (see the *NX Nastran Aeroelastic Analysis User's Guide).*

7. This entry can be used for two different aerodynamic theories. The Doublet-Lattice theory is used for subsonic and, if licensed, the ZONA51 theory for supersonic. The proper theory is selected based on the specification of Mach number on the MKAEROi or TRIM entry.

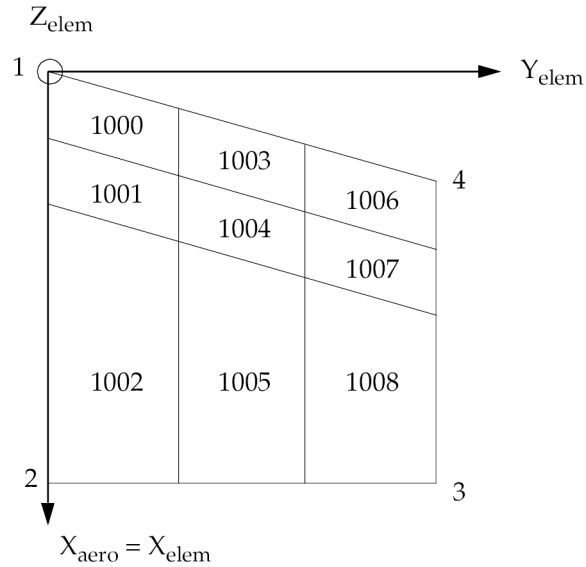


Figure 11-2. Element Coordinate System for Aerodynamic Panel

8. CAERO1 and CAERO2 panels should be defined in an IGID increasing order. For example, if you have a CAERO1 or CAER02 card with EID1 and ... IGID1, and another with EID2 and ... IGID2, IGID2 must be greater than IGID1, independent of whether EID1 < or > EID2. Failure to follow this restriction will create an erroneous pressure distribution resulting in incorrect flutter results.

CAERO2

Aerodynamic Body Connection

Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAERO2	EID	PID	CP	NSB	NINT	LSB	LINT	IGID	
	X1	Y1	Z1	X12					

EXAMPLE:

CAERO2	1500	2	100		4	99		1	
	-1.0	100.	-30.	175.					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PAERO2 entry. (Integer > 0)
CP	Coordinate system for locating point 1. (Integer ≥ 0; Default = 0)
NSB	Number of slender body elements. If NSB > 0, then NSB equal divisions are assumed; if zero or blank, specify a list of divisions in LSB. (Integer ≥ 0)
NINT	Number of interference elements. If NINT > 0, then NINT equal divisions are assumed; if zero or blank, specify a list of divisions in LINT. (Integer ≥ 0)

Field	Contents
LSB	ID of an AEFACT Bulk Data entry for slender body division points; used only if NSB is zero or blank. (Integer ≥ 0)
LINT	ID of an AEFACT data entry containing a list of division points for interference elements; used only if NINT is zero or blank. (Integer ≥ 0)
IGID	Interference group identification. Aerodynamic elements with different IGIDs are uncoupled. (Integer ≥ 0) See remark 8 for restriction.
X1, Y1, Z1	Location of point 1 in coordinate system CP. (Real)
X12	Length of body in the x-direction of the aerodynamic coordinate system. (Real ≥ 0)

REMARKS:

1. Point 1 is the leading point of the body.

2. All CAERO1 (panels) and CAERO2 (bodies) in the same group (IGID) will have aerodynamic interaction.

3. At least one interference element is required for the aerodynamic body specified by this entry.

4. The beams and connection points are numbered sequentially beginning with EID. The user should be careful to ensure that all aero elements and connection point IDs are unique. Overlapping IDs between structure and aerodynamic models are allowed, but will prevent results visualization in most post processors.

   Old rules regarding numbering among Z, ZY, Y bodies and CAERO1 no longer apply: arbitrary ordering is allowed.

5. At least two slender body elements are required for each aerodynamic body.

6. Interference elements are only intended for use with panels.

7. Determining the size of the j-set (i.e., the number of aerodynamic elements) is essential to input D1JE and D2JE matrices. Use the following expressions for locating the proper row in the two matrices:

$$
\begin{aligned}
J = \text{Number of boxes} &= + \text{ Number of I-elements, z} \\
&= + \text{ 2*(Number of I-elements, zy)} \\
&= + \text{ Number of I-elements, y} \\
&= + \text{ Number of S-elements, z} \\
&= + \text{ 2*(Number of S-elements, zy)} \\
&= + \text{ Number of S-elements, y}
\end{aligned}
$$

where I-elements denote interference and S-elements denote slender body.

8. CAERO1 and CAERO2 panels should be defined in an IGID increasing order. For example, if you have a CAERO1 or CAER02 card with EID1 and ... IGID1, and another with EID2 and ... IGID2, IGID2 must be greater than IGID1, independent of whether EID1 < or > EID2. Failure to follow this restriction will create an erroneous pressure distribution resulting in incorrect flutter results.

CAERO3

Aerodynamic Panel Element Configuration

Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAERO3	EID	PID	CP	LISTW	LISTC1	LISTC2			
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

EXAMPLE:

CAERO3	2000	2001	0	22	33				
	1.0	0.0	0.0	100.	17.	130.	0.	100.	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PAERO3 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
LISTW	Identification number of an AEFACT entry that lists (x,y) pairs for structural interpolation of the wing. (Integer > 0)
LISTC1, LISTC2	Identification number of AEFACT entries that list (x,y) pairs for control surfaces, if they exist. (Integer ≥ 0)

Field	Contents
X1, Y1, Z1, X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in the aerodynamic coordinate system. (Real ≥ 0, X12 ≠ 0)

REMARKS:

1. EID must be unique with respect to all other element identification numbers.

2. The (x,y) pairs on LISTW, LISTC1 and LISTC2 AEFACT entries are in the aero element coordinate system (see Figure 11-3). The (x,y) pairs define a set of aerodynamic grid points that are independent of the Mach number and are selected by the user to be representative of the platform and motions of interest. The (x,y) pairs must be sufficient in number and distribution such that: the surface spline provides an accurate interpolation between them and the Mach Box centers that are variously located on the planform as a function of Mach number. A complete description of the Mach Box Method is given in the *NX Nastran Aeroelastic Analysis User's Guide*.

3. The (x,y) pairs are numbered sequentially, beginning with EID for LISTW, then LISTC1, and finally for LISTC2. On SPLINEi entries, the box numbers (BOX1 and BOX2 on SPLINE1, ID1 and ID2 on SPLINE2, and UKID on SPLINE3) refer to the (x,y) pair sequence number appropriate for the surface (primary, or first or second control) being splined.

4. If cranks and/or control surfaces exist, their locations are given on the PAERO3 entry.

5. The numbering system and coordinate system are shown below:

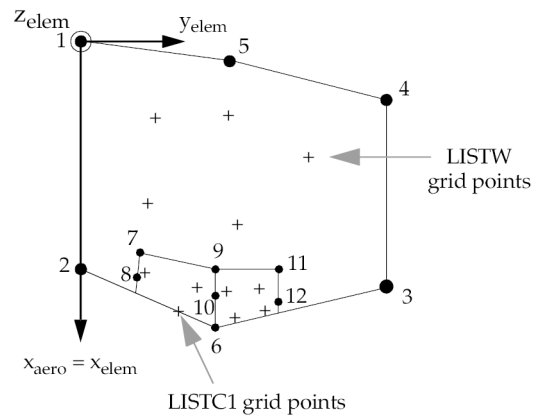


Figure 11-3. CAERO3 Element Configuration

Planform Corners		Control	
1	Leading edge, inboard	7	Hinge line, inboard
2	Trailing edge, inboard	8	On inboard edge (usually at trailing edge)
3	Trailing edge, outboard	9	Hinge line, outboard
4	Leading edge, outboard	10	On outboard edge (usually at trailing edge)
Cranks		**Control (if two)**	
5	Leading edge	9	Hinge line, inboard
6	Trailing edge	10	On inboard edge (usually at trailing edge)
		11	Hinge line, outboard
		12	On outboard edge (usually at trailing edge)

CAERO4

Aerodynamic Macro-Strip Element Connection

Defines an aerodynamic macro element for Strip theory.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAERO4	EID	PID	CP	NSPAN	LSPAN				
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

EXAMPLE:

CAERO4	6000	6001	100		315				
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PAERO4 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
NSPAN	Number of strips; if a positive value is given, NSPAN equal strips are assumed. If zero or blank, LSPAN must be specified. (Integer ≥ 0)
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)
X1, Y1, Z1, X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)

Field	Contents
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0.0, and not both zero.)

REMARKS:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and greater than structural grid, scalar, and extra point IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN = 3, the division points are 0.0, 0.333, 0.667, and 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.

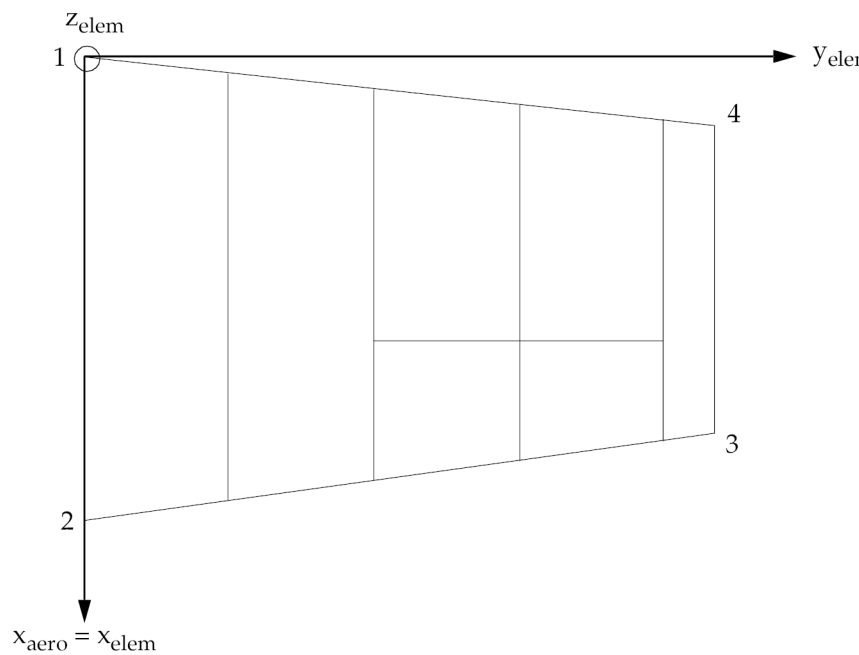


Figure 11-4. CAERO4 Element Connection

CAERO5

Aerodynamic Panel Element Configuration

Defines an aerodynamic macro element for Piston theory.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CAERO5	EID	PID	CP	NSPAN	LSPAN	NTHRY	NTHICK		
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

EXAMPLE:

CAERO5	6000	6001	100		315	0	0		
	0.0	0.0	0.0	1.0	0.2	1.0	0.	0.8	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PAERO5 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
NSPAN	Number of strips. If a positive value is given, equal strips are assumed. If zero or blank, then LSPAN must be specified. (Integer)
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)

Field	Contents
NTHRY	Parameter to select Piston or van Dyke's theory. (Integer = 0, 1, or 2; Default = 0) Blank - Piston theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ or 1 1 = van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with no sweep correction (secΛ = 1.0). 2 = van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with a sweep correction based on the actual Λ.
NTHICK	Parameter to select thickness integrals input. (Integer ≥ 0; Default = 0) 0 or Blank - Thickness integrals are computed internally. >0 - Thickness integrals are input directly and are the ID number of an AEFACT entry that lists the *I* and/or *J* integrals.
X1, Y1, Z1, X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0; X12 and X43 cannot both be zero.)

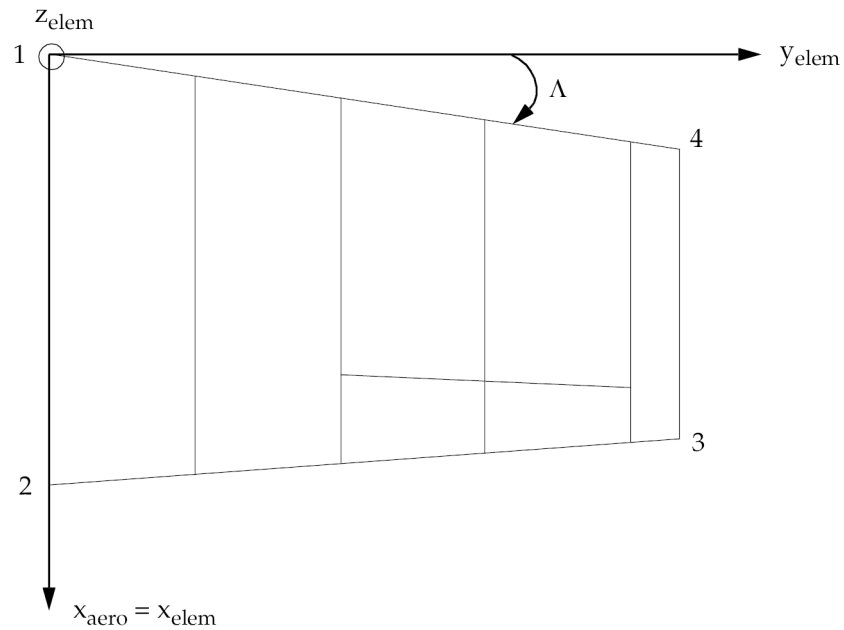


Figure 11-5. CAERO5 Element Configuration

REMARKS:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and different from structural grid IDs.

2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).

3. A triangular element is formed if X12 or X43 = 0.0.

4. $\bar{C}_1 = m/(m^2 - \sec^2 \Lambda)^{1/2}$

   $\bar{C}_2 = [m^4 (\gamma + 1) - 4 \sec^2 \Lambda (m^2 - \sec^2 \Lambda)] / [4(m^2 - \sec^2 \Lambda)^2]$

   where:

   m = Mach number

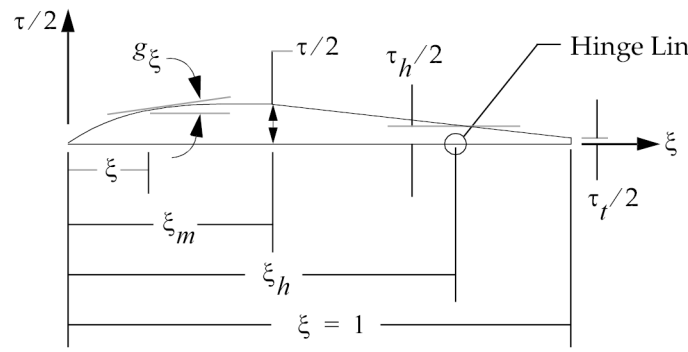
γ	=	Specific heat ratio
Λ	=	Leading edge sweep angle

When sec Λ = 0.0, Piston theory coefficients are obtained (NTHRY = 1).

When sec Λ = 1.0, van Dyke coefficients are obtained (NTHRY = blank or 0).

When sec Λ ≠ 0.0 or ≠ 1.0, sweep corrections are included (NTHRY = 2).

5. *I* and *J* thickness integral definitions:



$$g_\xi \equiv \frac{dg}{d\xi} = \text{slope of airfoil semithickness}$$

$$I_1 = \int_0^1 g_\xi \, d\xi \qquad J_1 = \int_{\xi_h}^1 g_\xi \, d\xi$$

$$I_2 = \int_0^1 \xi g_\xi \, d\xi \qquad J_2 = \int_{\xi_h}^1 \xi g_\xi \, d\xi$$

$$I_3 = \int_0^1 \xi^2 g_\xi \, d\xi \qquad J_3 = \int_{\xi_h}^1 \xi^2 g_\xi \, d\xi$$

$$I_4 = \int_0^1 g_\xi^{\,2} \, d\xi \qquad J_4 = \int_{\xi_h}^1 g^2{}_\xi \, d\xi$$

$$I_5 = \int_0^1 \xi g_\xi^{\,2} \, d\xi \qquad I_5 = \int_{\xi_h}^1 \xi g^2{}_\xi \, d\xi$$

$$I_6 = \int_0^1 \xi^2 g_\xi^{\,2} \, d\xi \qquad J_6 = \int_{\xi_h}^1 \xi^2 g_\xi^{\,2} \, d\xi$$

Figure 11-6. CAERO5 I and J Thickness Integral Definitions

CBAR

Simple Beam Element Connection

Defines a simple beam element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

EXAMPLE:

CBAR	2	39	7	3	0.6	18.	26.		
		513							

ALTERNATE FORMAT AND EXAMPLE:

CBAR	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

CBAR	2	39	7	6	105				
		513							

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)

Field	Contents
PID	Property identification number of a PBAR or PBARL entry. (Integer > 0 or blank; Default = EID unless BAROR has a nonzero entry in field 3.)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see Figure 11-7). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.)
W1A, W2A, W3A, W1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$, respectively (see Figure 11-7) in displacement coordinate systems at points GA and GB, respectively. See Remarks 7 and 8. (Real; Default = 0.0)

Note

See the BEAMOR entry for default options for field 3 and fields 6 through 8.

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The CBAR is referred to as a simple beam since it does not have the full capabilities of the CBEAM, for example, varying cross-section and warping. See the 1D Elements Chapter in the *NX Nastran Element Library Reference* for details on both CBAR and CBEAM elements.

3. Figure 11-7 and Figure 11-8 define bar element geometry with and without offsets.

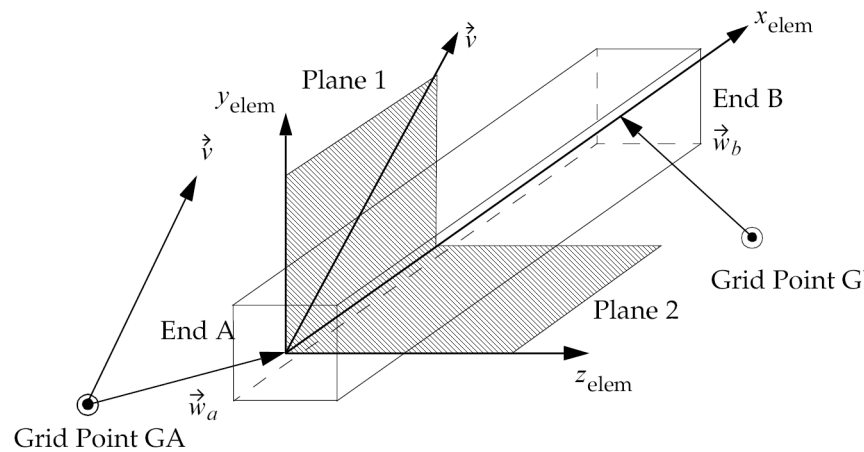


Figure 11-7. CBAR Element Geometry with Offsets

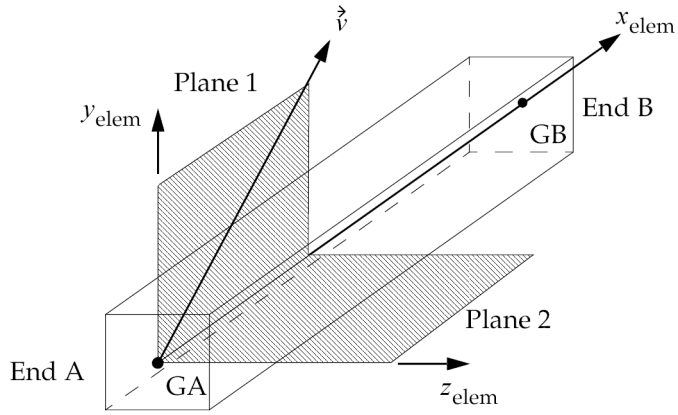


Figure 11-8. CBAR Element Geometry without Offsets

4. Figure 8-10 and Figure 8-11 define the elemental force and moment sign convention.

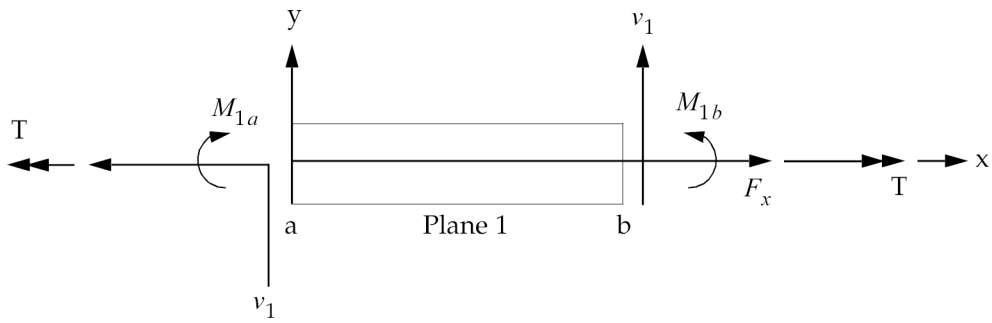


Figure 11-9. CBAR Element Internal Forces and Moments (x-y Plane)

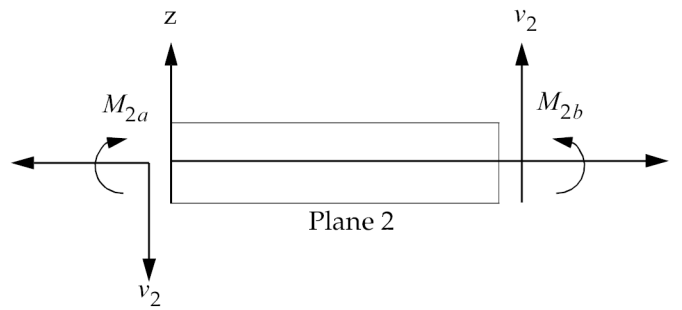


Figure 11-10. CBAR Element Internal Forces and Moments (x-z Plane)

5. The continuation may be omitted if there are no pin flags or offsets.

6. For the case where field 9 is blank and not provided by the BAROR entry, if an integer is specified in field 6, then G0 is used; if field 6 is blank or real, then X1, X2, X3 is used.

7. Element offsets are defined in a Cartesian system located at the connecting grid point. The components of the offsets are always defined in units of translation, even if the displacement coordinate system is cylindrical or spherical. For example, in Figure 11-11, the grid point displacement coordinate system is cylindrical, and the offset vector is defined using Cartesian coordinates u1, u2, and u3 in units of translation.

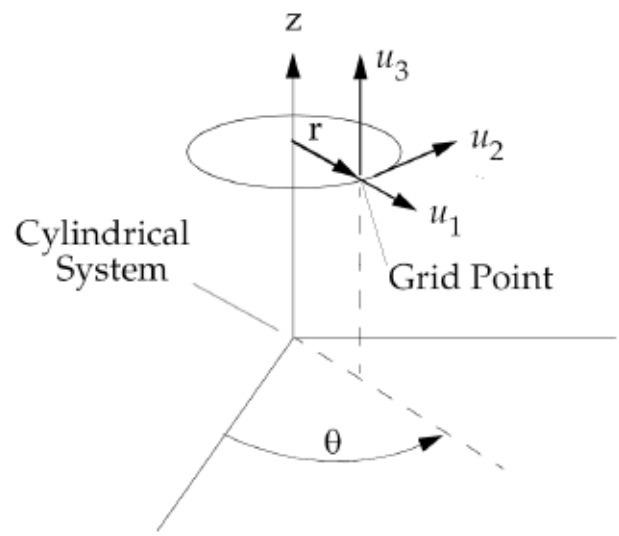


Figure 11-11. OFFSET Coordinates

8. Offset vectors are not affected by thermal loads.

REMARKS RELATED TO SOLS 601 AND 701:

1. W1A, W2A, W3A, W1B, W2B, and W3B are ignored.
2. CBAR defines the same element as CBEAM for SOL 601 and 701.
3. Result output for this element is requested using Case Control command STRESS. However, element forces instead of stresses are output for CBAR (and CBEAM) elements.

CBARAO

Auxiliary Output Points Along Bar Element Axis (CBAR Entry)

Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBARAO	EID	SCALE	X1	X2	X3	X4	X5	X6	

EXAMPLE:

CBARAO	1065	FR	0.2	0.4	0.6	0.8			

ALTERNATE FORMAT AND EXAMPLE:

CBARAO	EID	SCALE	NPTS	X1	DELTAX				
CBARAO	1065	FR	4	0.2	0.2				

FIELDS:

Field	Contents
EID	Element identification of a CBAR entry. (Integer > 0)
SCALE	Defines scale of Xi values. (Character = “LE” or “FR”)
Xi	Series of locations along element axis for stress and force data recovery. (Real > 0)
DELTAX	Incremental distance along element axis. (Real)

Field	Contents
NPTS	Number of stress recovery points, not including the end points. (Integer > 0)

REMARKS:

1. This entry defines intermediate locations on the axis of selected CBAR elements for additional data recovery. The values of Xi are actual distances along the length if SCALE = “LE”. If SCALE = “FR”, the values of Xi are ratios of actual distances to the bar length. A PLOAD1 Bulk Data entry for the CBAR element in question must be present to obtain intermediate data recovery.

2. When the alternate format is used, a series of locations Xi = X[i-1]+DELTAX, i = 1, 2, ..., NPTS is generated.

3. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress values at the end points of the beam will always be output. This output format will be used for all beam and bar elements.

4. Intermediate loads on the element defined by the PLOAD1 entry will be accounted for in the calculation of element stresses and forces. If no PLOAD1 entry is defined for the element, the shear forces are constant, the moments are linear, and it is not necessary that the user define additional points.

5. For each bar element, either the basic format or the alternate format, but not both, may be used. A maximum of six internal points can be specified with the basic form. The end points must not be listed because data will be generated for them, as explained in Remark 3 If more than six unequally spaced internal points are desired, it is advisable to subdivide the bar into two or more elements.

CBEAM

Beam Element Connection

Defines a beam element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

EXAMPLE:

CBEAM	2	39	7	13	8.2	6.1	-5.6		
		513		3.0					
	8	5							

ALTERNATE FORMAT AND EXAMPLE:

CBEAM	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

CBEAM	2	39	7	13	105				
		513							

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of PBEAM, PBCOMP or PBEAML entry. (Integer > 0; Default=EID)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to end A. (Integer > 0; G0 ≠ GA or GB)
PA, PB	Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees-of-freedom of the beam. The degrees-of-freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see Figure 11-12). The beam must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique integers 1 through 6 with no embedded blanks.)
W1A, W2A, W3A, W1B, W2B, W3B	Components of offset vectors, measured in the displacement coordinate systems at grid points A and B, from the grid points to the end points of the axis of shear center. See Remarks 7 and 8. (Real; Default = 0.0)
SA, SB	Scalar or grid point identification numbers for the ends A and B, respectively. The degrees-of-freedom at these points are the warping variables $d\theta/dx$. (Integers > 0 or blank)

Note

See the BEAMOR entry for default options for field 3 and fields 6 through 8.

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. For an additional explanation of the beam element, see “CBEAM Element” in the *NX Nastran Element Library*. Figures 11-12 and 11-13 show CBEAM element geometry, forces and moments.

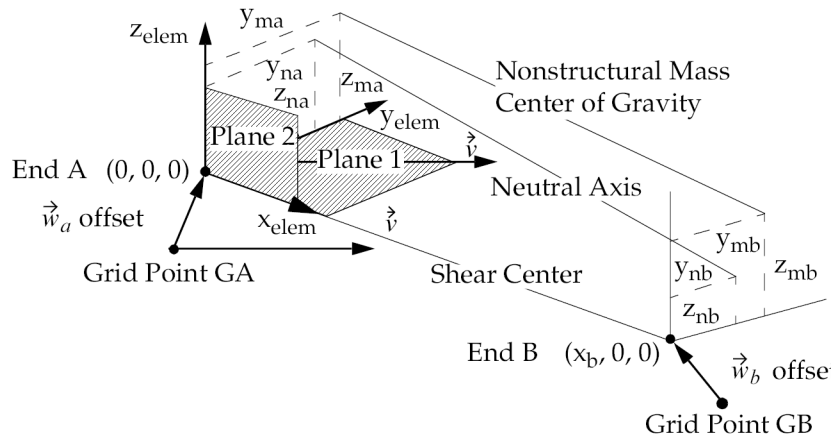


Figure 11-12. CBEAM Element Geometry System

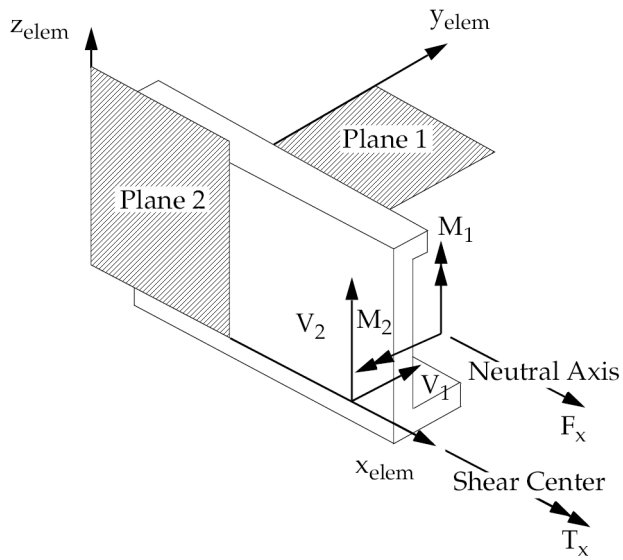


Figure 11-13. CBEAM Internal Element Forces and Moments

3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.

4. G0 cannot be located at GA or GB.

5. The rules for the continuations entries are:

   - Both continuations may be omitted if there are no pin flags, offsets, or warping variables.

   - If the second continuation is used, then the first continuation must be included, even if all fields are blank.

   - If the second continuation is omitted, torsional stiffness due to warping of the cross section will not be considered.

6. If warping is allowed (SA and SB > 0), then SA and SB must be defined with SPOINT or GRID entries. If GRID entries are used, the warping degree-of-freedom is attached to the first (T1) component.

7. Offset vectors are subject to the following limitations.

   - Offset vectors are not affected by thermal loads.

- The specification of offset vectors is not recommended with nonlinear solutions, SOLs 106, 129, 153, and 159. When geometry nonlinear conditions exist (PARAM,LGDISP,1), the offset vectors remain parallel to their original orientation when computing the differential stiffness. Also, when material nonlinear conditions are defined with the MATS1 entry, the offset vectors may produce incorrect results. As a result, the specification of offset vectors is not permitted in these solutions. However, setting SYSTEM(463) to 1 will disable the FATAL error message and the analysis will be allowed to continue (with offset vectors remaining parallel to their original orientation). The use of this system cell may generate incorrect results. Loading conditions that generate follower forces should not be used when SYSTEM(463) = 1. Use PARAM,FOLLOWK,NO if follower boundary conditions are specified.

8. Element offsets are defined in a Cartesian system located at the connecting grid point. The components of the offsets are always defined in units of translation, even if the displacement coordinate system is cylindrical or spherical. For example, in Figure 11-14, the grid point displacement coordinate system is cylindrical, and the offset vector is defined using Cartesian coordinates u1, u2, and u3 in units of translation.

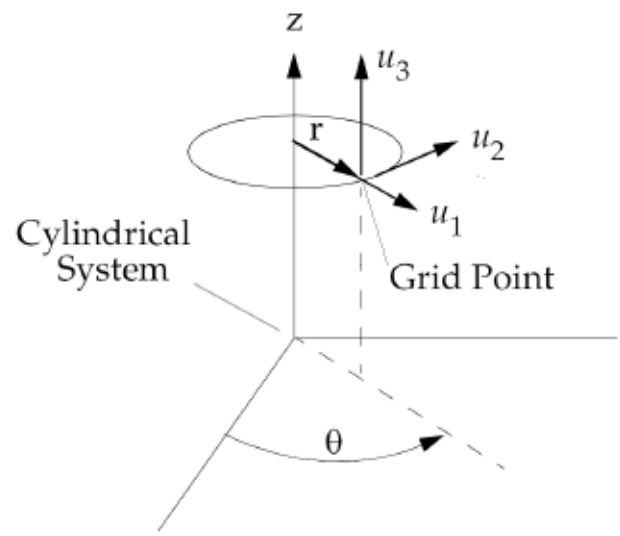


Figure 11-14. OFFSET Coordinates

9. Pin flags PA and PB will produce a fatal error in SOLs 106 and 129 when nonlinear material conditions are defined with the MATS1, MATTi, or CREEP entries. They can be used if the only nonlinearity is geometric.

REMARKS RELATED TO SOLS 601 AND 701:

1. BIT, W1A, W2A, W3A, W1B, W2B, W3B, SA, and SB are ignored.
2. CBEAM defines the same element as CBAR for SOLs 601 and 701.
3. Result output for this element is requested using Case Control command STRESS. However, element forces instead of stresses are output for CBEAM (and CBAR) elements.

CBEAR

Speed-Dependent Bearing Connection

Defines speed-dependent bearing connection. Valid for SOL 107-112 rotor dynamic analyses and SOL 101 maneuver load analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBEAR	EID	PID	GA	GB					

EXAMPLE:

CBEAR	1	2	100	101					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PBEAR entry. (Integer > 0)
GA, GB	Grid point identification number of bearing connection points. See Remark 3. (Integer > 0; GA ≠ GB)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. You must define a GROUP bulk entry for each rotor that has corresponding CBEAR entries. On each GROUP entry, list the CBEAR entries. Associate each GROUP entry with a rotor on the BRGSETi field on the ROTORD entry.

3. See the *Rotor Dynamics User's Guide* for information on modeling bearings including bearings that carry thrust loads in only one axial direction.

CBEND

Curved Beam or Pipe Element Connection

Defines a curved beam, curved pipe, or elbow element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

EXAMPLE:

CBEND	32	39	17	19	6.2	5.1	-1.2		

ALTERNATE FORMAT AND EXAMPLE:

CBEND	EID	PID	GA	GB	G0			GEOM	

CBEND	32	39	17	19	106				

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PBEND entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)

Field	Contents
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to end A. (Integer > 0; G0 ≠ GA or GB)
GEOM	Flag to select specification of the bend element. See Remark 3 (1 < Integer < 4)

REMARKS:

1. Element identification numbers must be unique with respect to all other element identification numbers.

2. For an additional explanation of the CBEND element, see the PBEND entry description. Figure 11-15 and Figure 11-16 define the element coordinate system and internal forces and moments.

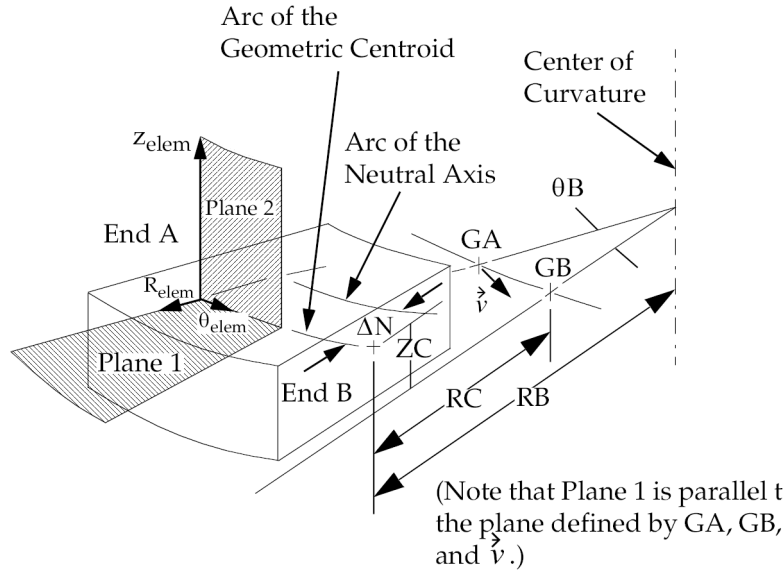


Figure 11-15. CBEND Element Coordinate System

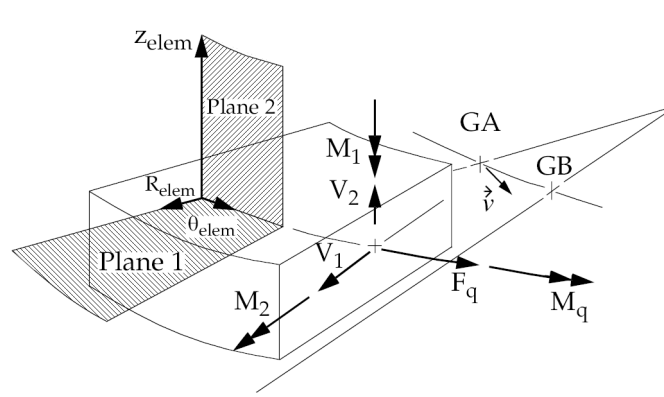
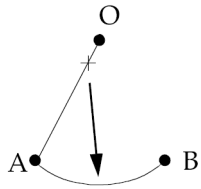
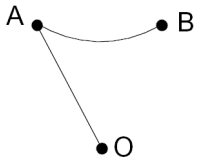
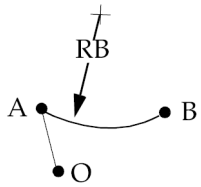
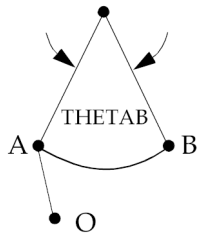


Figure 11-16. CBEND Element Internal Forces and Moments

3. The options for element connection to GA, GB using GEOM are shown below.

Table 11-1. GEOM Options

Configuration	GEOM	Description
	1	The center of curvature lies on the line AO (or its extension) or vector $\vec{v}$.
	2	The tangent of centroid arc at end A is parallel to line AO or vector $\vec{v}$. Point O (or vector $\vec{v}$) and the arc $\overline{AB}$ must be on the same side of the chord $\overline{AB}$.
	3	The bend radius (RB) is specified on the PBEND entry: Points A, B, and O (or vector $\vec{v}$) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$) lies on the opposite side of line AB from the center of the curvature.
	4	THETAB is specified on the PBEND entry. Points A, B, and O (or vector $\vec{v}$) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$) lies on the opposite side of line AB from the center of curvature.

CBUSH

Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

EXAMPLE 1:

NONCOINCIDENT GRID POINTS.

CBUSH	39	6	1	100	75				

EXAMPLE 2:

GB NOT SPECIFIED.

CBUSH	39	6	1					0	

EXAMPLE 3:

COINCIDENT GRID POINTS.

CBUSH	39	6	1	100				6	

EXAMPLE 4:

NONCOINCIDENT GRID POINTS WITH FIELDS 6 THROUGH 9 BLANK AND A SPRING-DAMPER OFFSET.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PBUSH entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark 6. (Integer GA > 0)
Xi	Components of orientation vector
	$\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector
	$\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to end A. See Remark 3 . (Integer > 0)

Field	Contents
CID	Element coordinate system identification. A 0 means the basic coordinate system. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 11-17 and Remark 3 . (Integer ≥ 0 or blank)
S	Location of spring damper. See Figure 11-17. (0.0 ≤ Real ≤ 1.0; Default = 0.5)
OCID	Coordinate system identification of spring-damper offset. See Remark 9 . (Integer ≥ -1; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 11-17.)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0. See Figure 11-18 and Remark 9 .. (Real)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Figure 11-17 shows the bush element geometry.

3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. For cylindrical or spherical coordinate systems, if GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis.

4. For noncoincident grids, when GO or X1, X2, X3 is given and no CID is specified,

   the line AB is the element x-axis and the orientation vector $\vec{v}$, lies in the x-y plane (similar to the CBEAM element).

5. For noncoincident grids, if neither GO or X1, X2, X3 is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both are specified on the PBUSH entry (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.

6. If GA and GB are coincident, or if GB is blank, then CID must be specified. When GB is blank, a grounded spring and damper is created at GA.

7. Only CBUSH elements in the residual structure that do not attach to any omitted degree-of-freedom can reference a PID identifying both a PBUSH entry and a PBUSHT entry.

8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.

9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0, then S is ignored and S1, S2, S3 are used.

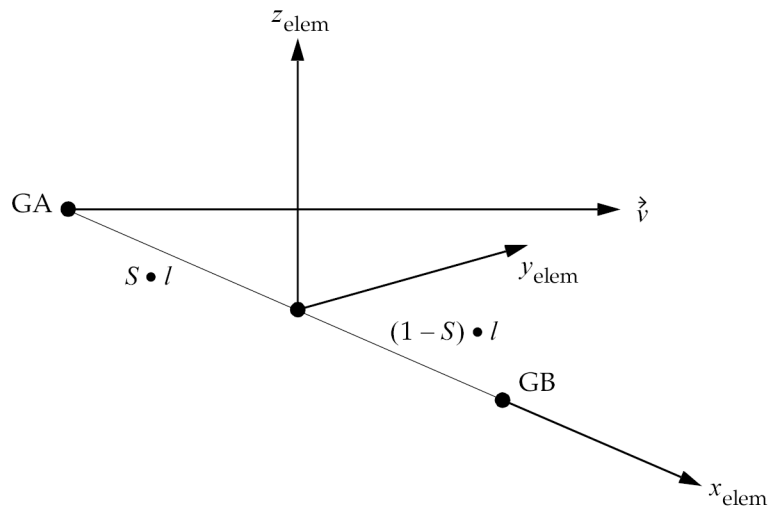


Figure 11-17. CBUSH Element

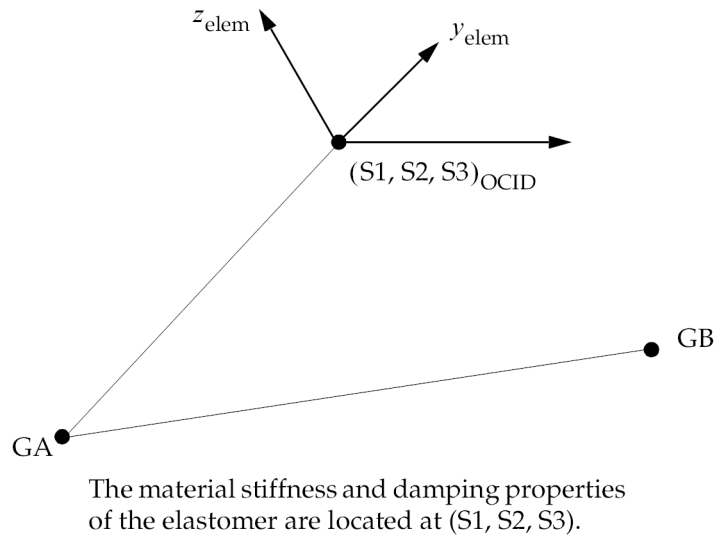


Figure 11-18. Definition of Offset S1, S2, S3

10. When CID ≥ 0, the element x-axis is set as in Remark 2 . This means that the element force is always computed as Ke. (UB − UA); if UA > UB, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation is tension and relative negative elongation is compression.

11. For this particular element, the effect of PARAM, G and the damping coefficient GE are included in the bushing stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters and Parameter Applicability Tables. For more details on element force calculation, see the NX Nastran Element Library.

12. In SOL 106, the STRESS case control command should be used to request the element force and stress output for the CBUSH and CBUSH1D elements.

REMARKS RELATED TO SOL 601:

1. S, OCID, S1, S2 and S3 are not supported.

CBUSH1D

Rod Type Spring-and-Damper Connection

Defines the connectivity of a one-dimensional spring and viscous damper element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CBUSH1D	EID	PID	GA	GB	CID				

EXAMPLE:

CBUSH1D	35	102	108	112					

FIELDS:

Field	Contents	Default Values
EID	Element identification number.	Required
PID	Property identification number of a PBUSH1D entry. (Integer > 0).	EID
GA	Grid point id of first grid.	Required
GB	Grid point id of second grid	Blank
CID	Coordinate system id. (Integer ≥ 0)	Blank

REMARKS:

1. For noncoincident grids GA ≠ GB and if CID is blank, the line GA to GB is the element axis. In geometric nonlinear analysis, the element axis (line GA to GB) follows the deformation of grids GA and GB. See Figure 11-19.

2. If CID ≥ 0 is specified, the x-axis of the CID coordinate system is the element axis. In geometric nonlinear analysis, the element axis (x-axis of CID) remains fixed for this case.

3. If GA and GB are coincident or if GB is blank, then CID ≥ 0 must be specified and the element axis is the x-axis of CID.

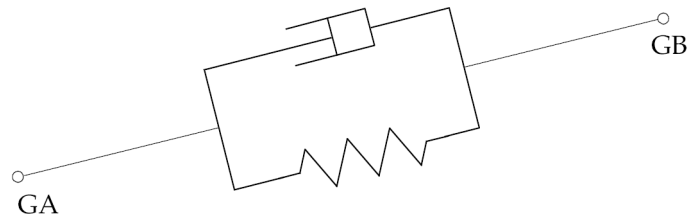


Figure 11-19. Spring and Damper Element

4. In SOL 106, the STRESS case control command should be used to request the element force and stress output for the CBUSH and CBUSH1D elements.

CCHOCK3

2D Chocking Triangular Element Connection

Defines a 2D chocking triangular element connection. Valid for SOL 401 axisymmetric analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CCHOCK3	EID	PID	G1	G2	G3		THETA		
					GAP1	GAP2	GAP3		

EXAMPLE:

CCHOCK3	19	9	1215	1216	1342				
						0.001	0.0		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCHOCK bulk entry. (Integer > 0; No default)
Gi	Grid point identification number of connection points. (Integer > 0; No default)
THETA	Material property orientation angle in degrees. See Remark 4. (Real; Default = 0.0)
GAPi	Initial gap opening at grid point Gi. See Remark 6. (Real or blank; See Remark 5 for default behavior)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system.

3. The basic coordinate system is the reference coordinate system for stress and strain output.

4. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 11-20 and 11-21 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. If GAPi is blank, the value for GAPT on the corresponding PCHOCK entry is used for the initial gap opening at grid point Gi. If GAPi and GAPT on the corresponding PCHOCK entry are both specified, the GAPi specification takes precedence. The software computes the total initial gap opening at grid point Gi as the product of GAPi (or GAPT) and the value for NGAP on the corresponding PCHOCK entry.

6. If the value for GAPi or, if applicable, the value for GAPT on the corresponding PCHOCK entry is negative, the software uses the value to calculate an initial compressive hoop strain.

7. For any grid point Gi selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the displacement coordinate system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis (φ-axis for a spherical system) of the displacement coordinate system must be in the same direction as the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system must be in the same direction as the Y-axis of the basic coordinate system.

8. GPSTRESS and GPSTRAIN output are not supported.

9. The behavior of chocking elements depend on whether large displacements are enabled.

   - When large displacements are not enabled, the gap status at the beginning of the analysis is used throughout the analysis and the stiffness does not reformulate unless plasticity is enabled. When plasticity is enabled, the stiffness reformulates to account for plasticity effects only.

   - When large displacements are enabled, the solution is iterative. The initial gap status is used for the first iteration. The gap status at the end of each iteration is updated and the stiffness is reformulated. When plasticity is enabled, the stiffness reformulates to account for plasticity effects, as well as changes in gap status.

     To enable large displacements, specify PARAM,LGDISP,+1 in the input file.

10. At any Gauss point location where there is no gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element. At any Gauss point location where there is a gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element with the E_θ, $v_{\theta r}$, and $v_{\theta z}$ elastic constants reduced by a factor of 1 x 10^6 so that the element stiffness becomes essentially identical to a plane stress formulation with σ_θ as the out-of-plane normal stress.

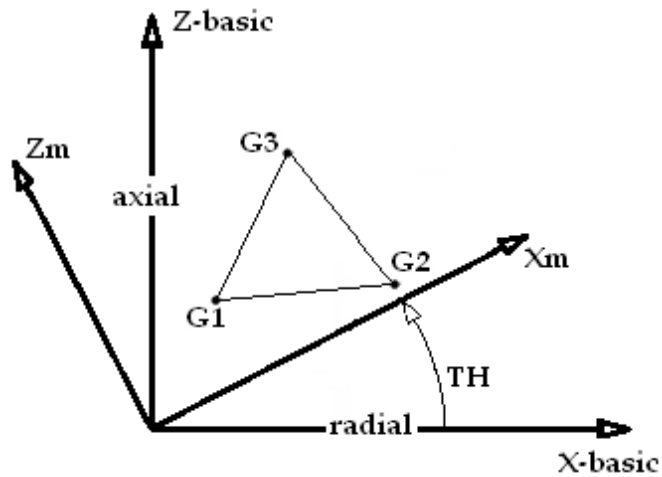


Figure 11-20. CCHOCK3 Element Geometry and Coordinate Systems, XZ-Plane

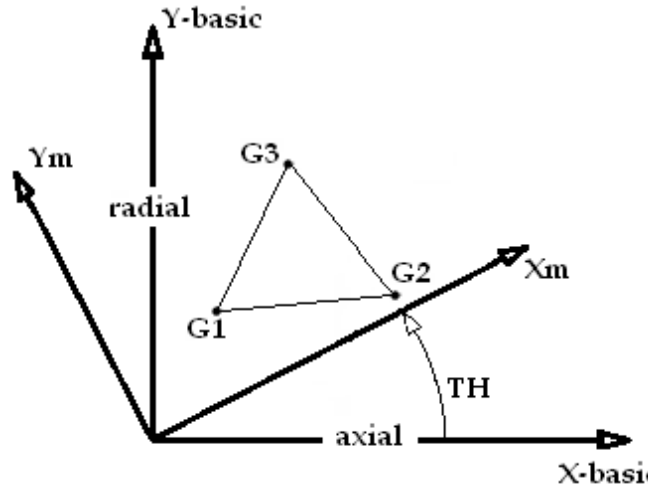


Figure 11-21. CCHOCK3 Element Geometry and Coordinate Systems, XY-Plane

CCHOCK4

2D Chocking Quadrilateral Element Connection

Defines a 2D chocking quadrilateral element connection. Valid for SOL 401 axisymmetric analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CCHOCK4	EID	PID	G1	G2	G3	G4	THETA		
					GAP1	GAP2	GAP3	GAP4	

EXAMPLE:

CCHOCK4	19	9	1215	1216	1342	1222			
					0.001		0.0	0.002	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCHOCK bulk entry. (Integer > 0; No default)
Gi	Grid point identification number of connection points. (Integer > 0; No default)
THETA	Material property orientation angle in degrees. See Remark 4. (Real; Default = 0.0)
GAPi	Initial gap opening at grid point Gi. See Remark 6. (Real or blank; See Remark 5 for default behavior)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8), must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system.

3. The basic coordinate system is the reference coordinate system for stress and strain output.

4. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 11-22 and 11-23 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. If GAPi is blank, the value for GAPT on the corresponding PCHOCK entry is used for the initial gap opening at grid point Gi. If GAPi and GAPT on the corresponding PCHOCK entry are both specified, the GAPi specification takes precedence. The software computes the total initial gap opening at grid point Gi as the product of GAPi (or GAPT) and the value for NGAP on the corresponding PCHOCK entry.

6. If the value for GAPi or, if applicable, the value for GAPT on the corresponding PCHOCK entry is negative, the software uses the value to calculate an initial compressive hoop strain.

7. For any grid point Gi selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the displacement coordinate system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis (φ-axis for a spherical system) of the displacement coordinate system must be in the same direction as the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system must be in the same direction as the Y-axis of the basic coordinate system.

8. GPSTRESS and GPSTRAIN output are not supported.

9. The behavior of chocking elements depend on whether large displacements are enabled.

   - When large displacements are not enabled, the gap status at the beginning of the analysis is used throughout the analysis and the stiffness does not reformulate unless plasticity is enabled. When plasticity is enabled, the stiffness reformulates to account for plasticity effects only.

   - When large displacements are enabled, the solution is iterative. The initial gap status is used for the first iteration. The gap status at the end of each iteration is updated and the stiffness is reformulated. When plasticity is enabled, the stiffness reformulates to account for plasticity effects, as well as changes in gap status.

     To enable large displacements, specify PARAM,LGDISP,+1 in the input file.

10. At any Gauss point location where there is no gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element. At any Gauss point location where there is a gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element with the E_θ, $v_{\theta r}$, and $v_{\theta z}$ elastic constants reduced by a factor of 1 x 10^6 so that the element stiffness becomes essentially identical to a plane stress formulation with σ_θ as the out-of-plane normal stress.

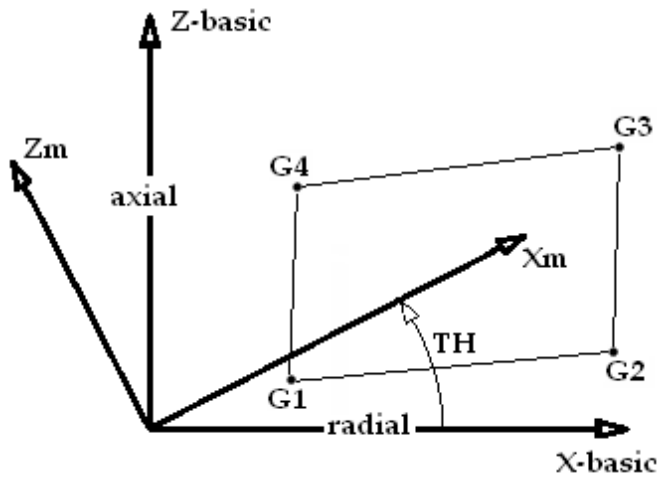


Figure 11-22. CCHOCK4 Element Geometry and Coordinate Systems, XZ-Plane

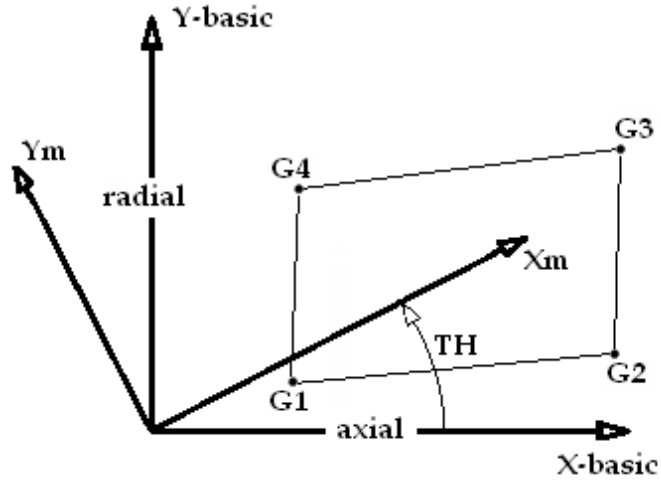


Figure 11-23. CCHOCK4 Element Geometry and Coordinate Systems, XY-Plane

CCHOCK6

2D Chocking Triangular Element Connection

Defines a 2D chocking triangular element connection. Valid for SOL 401 axisymmetric analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CCHOCK6	EID	PID	G1	G2	G3	G4	G5	G6	
			THETA		GAP1	GAP2	GAP3		

EXAMPLE:

CCHOCK6	237	1	1215	1217	1342	1216	1289	1290	
			30.0			0.001	0.0		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCHOCK bulk entry. (Integer > 0; No default)
Gi	Grid point identification number of connection points. (Integer > 0; No default)
THETA	Material property orientation angle in degrees. See Remark 4. (Real; Default = 0.0)
GAPi	Initial gap opening at grid point Gi. See Remark 6. (Real or blank; See Remark 5 for default behavior)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8), must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system.

3. The basic coordinate system is the reference coordinate system for stress and strain output.

4. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 11-24 and 11-25 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. If GAPi is blank, the value for GAPT on the corresponding PCHOCK entry is used for the initial gap opening at grid point Gi. If GAPi and GAPT on the corresponding PCHOCK entry are both specified, the GAPi specification takes precedence. The software computes the total initial gap opening at grid point Gi as the product of GAPi (or GAPT) and the value for NGAP on the corresponding PCHOCK entry.

6. If the value for GAPi or, if applicable, the value for GAPT on the corresponding PCHOCK entry is negative, the software uses the value to calculate an initial compressive hoop strain.

7. For any grid point Gi selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the displacement coordinate system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis (φ-axis for a spherical system) of the displacement coordinate system must be in the same direction as the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system must be in the same direction as the Y-axis of the basic coordinate system.

8. GPSTRESS and GPSTRAIN output are not supported.

9. The behavior of chocking elements depend on whether large displacements are enabled.

   - When large displacements are not enabled, the gap status at the beginning of the analysis is used throughout the analysis and the stiffness does not reformulate unless plasticity is enabled. When plasticity is enabled, the stiffness reformulates to account for plasticity effects only.

   - When large displacements are enabled, the solution is iterative. The initial gap status is used for the first iteration. The gap status at the end of each iteration is updated and the stiffness is reformulated. When plasticity is enabled, the stiffness reformulates to account for plasticity effects, as well as changes in gap status.

     To enable large displacements, specify PARAM,LGDISP,+1 in the input file.

10. At any Gauss point location where there is no gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element. At any Gauss point location where there is a gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element with the E_θ, $v_{\theta r}$, and $v_{\theta z}$ elastic constants reduced by a factor of 1 x 10^6 so that the element stiffness becomes essentially identical to a plane stress formulation with σ_θ as the out-of-plane normal stress.

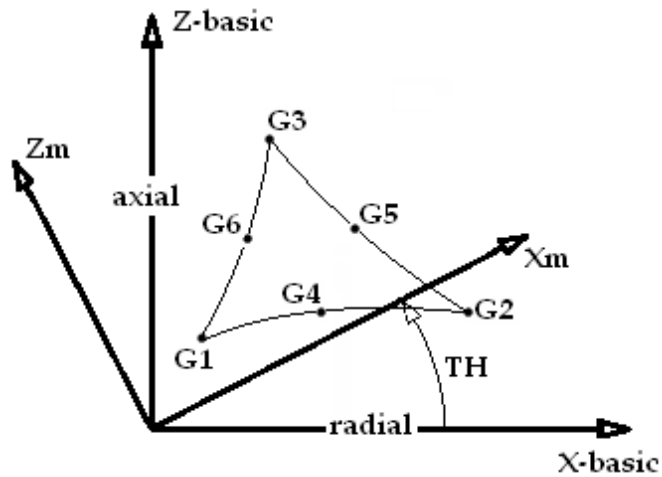


Figure 11-24. CCHOCK6 Element Geometry and Coordinate Systems, XZ-Plane

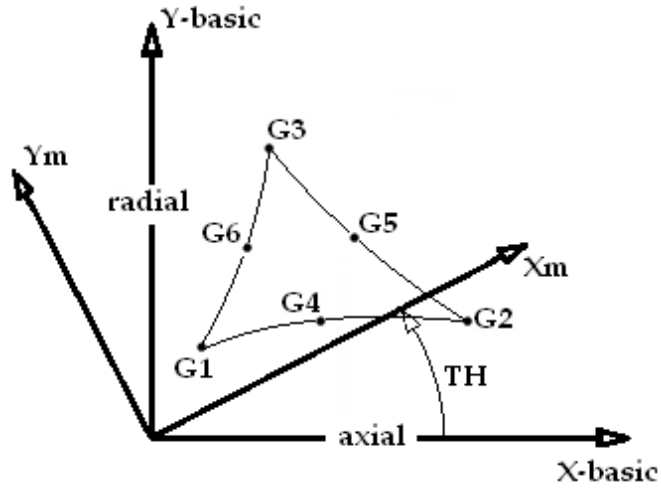


Figure 11-25. CCHOCK6 Element Geometry and Coordinate Systems, XY-Plane

CCHOCK8

2D Chocking Quadrilateral Element Connection

Defines a 2D chocking quadrilateral element connection. Valid for SOL 401 axisymmetric analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CCHOCK8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	THETA		GAP1	GAP2	GAP3	GAP4	

EXAMPLE:

CCHOCK8	19	9	1215	1217	1442	1440	1216	1886	
	1441	1385	30.0		0.001		0.0	0.002	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCHOCK bulk entry. (Integer > 0; No default)
Gi	Grid point identification number of connection points. (Integer > 0; No default)
THETA	Material property orientation angle in degrees. See Remark 4. (Real; Default = 0.0)
GAPi	Initial gap opening at grid point Gi. See Remark 6. (Real or blank; See Remark 5 for default behavior)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8), must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system.

3. The basic coordinate system is the reference coordinate system for stress and strain output.

4. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 11-26 and 11-27 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. If GAPi is blank, the value for GAPT on the corresponding PCHOCK entry is used for the initial gap opening at grid point Gi. If GAPi and GAPT on the corresponding PCHOCK entry are both specified, the GAPi specification takes precedence. The software computes the total initial gap opening at grid point Gi as the product of GAPi (or GAPT) and the value for NGAP on the corresponding PCHOCK entry.

6. If the value for GAPi or, if applicable, the value for GAPT on the corresponding PCHOCK entry is negative, the software uses the value to calculate an initial compressive hoop strain.

7. For any grid point Gi selected on axisymmetric, plane stress, plane stain, or chocking elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the displacement coordinate system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis (φ-axis for a spherical system) of the displacement coordinate system must be in the same direction as the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system must be in the same direction as the Y-axis of the basic coordinate system.

8. GPSTRESS and GPSTRAIN output are not supported.

9. The behavior of chocking elements depend on whether large displacements are enabled.

   - When large displacements are not enabled, the gap status at the beginning of the analysis is used throughout the analysis and the stiffness does not reformulate unless plasticity is enabled. When plasticity is enabled, the stiffness reformulates to account for plasticity effects only.

   - When large displacements are enabled, the solution is iterative. The initial gap status is used for the first iteration. The gap status at the end of each iteration is updated and the stiffness is reformulated. When plasticity is enabled, the stiffness reformulates to account for plasticity effects, as well as changes in gap status.

     To enable large displacements, specify PARAM,LGDISP,+1 in the input file.

10. At any Gauss point location where there is no gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element. At any Gauss point location where there is a gap, the contribution to the elemental stiffness matrix for the chocking element from that Gauss point is based on the axisymmetric formulation of the chocking element with the E_θ, $v_{\theta r}$, and $v_{\theta z}$ elastic constants reduced by a factor of 1 x 10^6 so that the element stiffness becomes essentially identical to a plane stress formulation with σ_θ as the out-of-plane normal stress.

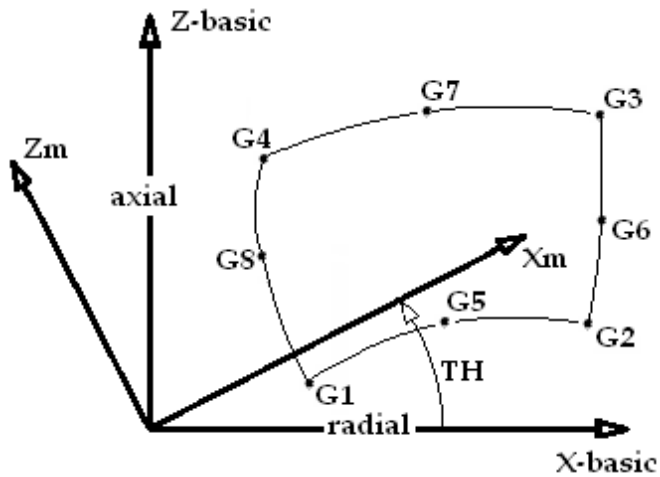


Figure 11-26. CCHOCK8 Element Geometry and Coordinate Systems, XZ-Plane

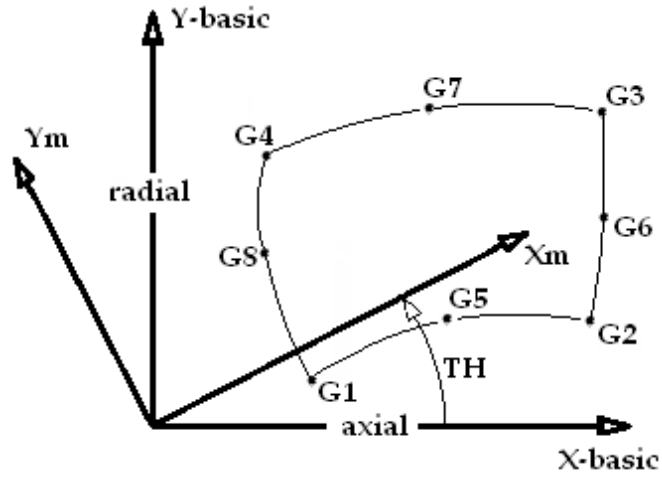


Figure 11-27. CCHOCK8 Element Geometry and Coordinate Systems, XY-Plane

CDAMP1

Scalar Damper Connection

Defines a scalar damper element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

EXAMPLE:

CDAMP1	19	6	0		23	2			

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PDAMP property entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 apply to grid points.)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a

grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. The two connection points (G1, C1) and (G2, C2), must be distinct.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

REMARKS RELATED TO SOLS 601 AND 701:

G1 and G2 must be grid points, i.e., they cannot be scalar points.

CDAMP2

Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

EXAMPLE:

CDAMP2	16	2.98	32	1					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 apply to grid points.)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. The two connection points (G1, C1) and (G2, C2), must be distinct.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

REMARKS RELATED TO SOLS 601 AND 701:

G1 and G2 must be grid points, i.e., they cannot be scalar points.

CDAMP3

Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

EXAMPLE:

CDAMP3	16	978	24	36					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PDAMP entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

CDAMP4

Scalar Damper Property and Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points, without reference to a material or property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

EXAMPLE:

CDAMP4	16	-2.6	4	9					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
B	Scalar damper value. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

CDAMP5

Scalar Damper with Material Property

Defines a damping element that refers to a material property entry and connection to grid or scalar points. This element is intended for heat transfer analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDAMP5	EID	PID	G1	G2					

EXAMPLE:

CDAMP5	1	4	10	20					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Identification number of a PDAMP5 property entry. (Integer > 0; Default = EID)
G1, G2	Grid or scalar point identification numbers. (Integer ≥ 0 and G1 ≠ G2)

REMARKS:

1. G1 or G2 may be blank or zero indicating a constraint.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. CDAMP5 generates a lumped heat capacity in heat transfer analysis.

4. A scalar point specified on CDAMP5 need not be defined on an SPOINT entry.

CDUMi

Dummy Element Connection

Defines a dummy element (1 ≤ i ≤ 9)

FORMAT:

1	2	3	4	5	6	7	8	9	10
CDUMi	EID	PID	G1	G2	G3	G4	-etc.-		
	A1	A2	-etc.-						

EXAMPLE:

CDUM2	114	108	2	5	6	8	11		
	2.4		3.E4	2		50			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PDUMi entry. See Remark 2 . (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0, G1 ≠ G2 ... ≠ GN)
Ai	Additional fields. (Real or Integer)

REMARKS:

1. The user must write the associated element subroutines for matrix generation, stress recovery, etc., and perform a link edit to replace the dummy routines.

2. If no property entry is required, PID may contain the material identification number.

3. Additional entries are defined in the user-written element routines.

4. CDUM1 is replaced by the CTRIAX6 element. If CDUM1 is used, User Fatal Message 307 will be issued.

CELAS1

Scalar Spring Connection

Defines a scalar spring element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

EXAMPLE:

CELAS1	2	6			8	1			

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.

If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.

2. In solutions 153 and 159, the software calculates heat conduction for CELAS1 elements as

   $q = K \times \Delta T$

   where K is the value entered on the PELAS entry. Both G1 and G2 must be defined, and C1 and C2 must be “1” when grid point ID’s are entered for G1 and G2.

3. Element identification numbers should be unique with respect to all other element identification numbers.

4. The two connection points (G1, C1) and (G2, C2) must be distinct.

5. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

7. If Gi refers to a grid point, then Ci refers to degree-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

8. The effect of PARAM,G and the damping coefficient GE (defined on the PELAS entry) are included in the spring stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters and Parameter Applicability Tables. For more details on element force and stress calculation, see the NX Nastran Element Library.

REMARKS RELATED TO SOLS 601 AND 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. In SOL 701, this spring element is not considered in the critical time step size calculation because it has no mass.

CELAS2

Scalar Spring Property and Connection

Defines a scalar spring element without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

EXAMPLE:

CELAS2	28	6.2+3	32		19	4			

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. See Remarks 2 and 11. (Real)
G1, G2	Geometric grid point or scalar identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)
GE	Damping coefficient. See Remarks 9 and 10. (Real)
S	Stress coefficient. (Real)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a

grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.

2. In solutions 153 and 159, the software calculates heat conduction for CELAS2 elements as

   $q = K \times \Delta T$

   where K is the value entered on the CELAS2 entry. Both G1 and G2 must be defined, and C1 and C2 must be “1” when grid point ID’s are entered for G1 and G2.

3. Element identification numbers should be unique with respect to all other element identification numbers.

4. The two connection points (G1, C1) and (G2, C2) must be distinct.

5. The element stress is computed by multiplying the stress coefficient S with the recovered element force.

6. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library.*

7. A scalar point specified on this entry need not be defined on an SPOINT entry.

8. If Gi refers to a grid point, then Ci refers to degree-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

10. The effect of PARAM, G and the damping coefficient GE are included in the spring stiffness for force and stress calculation in a frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters and Parameter Applicability Tables. For more details on element force calculation see the NX Nastran Element Library.

11. Rotational stiffness should be specified as moment per radian.

REMARKS RELATED TO SOLS 601 AND 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. GE is ignored.

3. In SOL 701, this spring element is not considered in the critical time step size calculation because it has no mass.

CELAS3

Scalar Spring Connection to Scalar Points Only

Defines a scalar spring element that connects only to scalar points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

EXAMPLE:

CELAS3	19	2	14	15					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. In solutions 153 and 159, the software calculates heat conduction for CELAS3 elements as

   $q = K \times \Delta T$

   where K is the value entered on the PELAS entry. Both S1 and S2 must be defined.

7. The effect of PARAM,G and the damping coefficient GE (defined on the PELAS entry) are included in the spring stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters and Parameter Applicability Tables. For more details on element force calculation, see the NX Nastran Element Library.

CELAS4

Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

EXAMPLE:

CELAS4	42	6.2-3	2						

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. See Remarks 8 and 9. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The structural damping coefficient GE is not available with CELAS4. The effect of PARAM,G is included in the spring stiffness for force calculations in

frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. See Parameters and Parameter Applicability Tables.

4. No stress coefficient is available with CELAS4.

5. Only one scalar spring element may be defined on a single entry.

6. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

7. A scalar point specified on this entry need not be defined on an SPOINT entry.

8. Rotational stiffness should be specified as moment per radian.

9. In solutions 153 and 159, the software calculates heat conduction for CELAS4 elements as

   $q = K \times \Delta T$

   where K is the value entered on the CELAS4 entry. Both S1 and S2 must be defined.

CFAST

Defines Shell Patch Connection

Defines a shell patch connection with direct stiffness input.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

EXAMPLE USING PROP:

1	2	3	4	5	6	7	8	9	10
CFAST	10	25	PROP	2	4	105			

EXAMPLE USING ELEM:

1	2	3	4	5	6	7	8	9	10
CFAST	10	25	ELEM	36	46	105			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PFAST bulk data entry (Integer > 0; Default = EID)

Field	Contents
TYPE	Determines if shell element patches A and B are defined by element or property identification numbers. (Character) See Remark 1. If TYPE = ‘PROP’, IDA,IDB define pshell property identification numbers. If TYPE = ‘ELEM’, IDA,IDB define shell element identification numbers.
IDA,IDB	Property id (for type PROP) or Element id (for type ELEM) defining patches A and B. IDA and IDB should be unique. (Integer > 0)
GS	Grid point used to determine the location of the connection on patches A and B, only when GA,GB are blank or 0. See Remark 2 . (Integer > 0 or blank)
GA,GB	Grid point used to determine the location of the connection on patches A and B. See Remark 2 . (Integer > 0 or blank)
XS,YS,ZS	Determines the coordinate location of the connection on patches A and B, only if GA,GB,GS are all 0 or blank. See Remark 2 . (Real or blank)

REMARKS:

1. The CFAST element defines a weld like connection between two shell element patches. The CFAST element TYPE = “PROP” and “ELEM” are similar to the CWELD element TYPE = “PARTPAT” and “ELPAT”, respectively, except that the CFAST element stiffness is entered directly on the KTi and KRi fields of the PFAST entry. The CWELD element stiffness is derived from an axial rod of diameter D, where D is specified on the PWELD bulk entry. The diameter D specified on the PFAST entry is only used to determine the CFAST connectivity on the patches (grid points GHA1-GHA4 on patch A and GHB1-GHB4 on patch B). See the remarks on the CWELD bulk entry for the CFAST element connectivity information.

2. GA/GB, or GS, or XS/YS/ZS, in that order of precedence, determine the location(s) of the CFAST connection on surface patches A and/or B. The connection location on patch A is a specification of grid point GA, or a projection of GS or XS/YS/ZS normal to surface patch A. The location on patch B is a specification of grid point GB, or a projection of GS or XS/YS/ZS normal to surface patch B. When GS or XS/YS/ZS are used, a normal projection must exist in order to define a valid connection element. GS or XS/YS/ZS do not need to

lie on either surface patch A or B. If GS or XS/YS/ZS are used to define the connection location, grid points are internally created at the connection location with an ID. The numbering of the internal IDs is describe in Remark 3.

3. When GA and GB are specified, the displacements for these grids will be printed/stored as requested on the DISPLACEMENT case control command. If these points are not specified, they will be generated by the software, and the displacements for these grids can only be written to the .f06 file with grid IDs calculated as follows:
   Grid ID = Value of SYSTEM(178) + (CWELD ID) + (end ID)
   where the end ID is 1 or 2 for the two ends. SYSTEM(178) defaults to 101,000,000.
   The new grid IDs are only used for labeling the displacement output and do not exist in the analysis or internal tables. This displacement result is not written to the .op2 or punch file.

CGAP

Gap Element Connection

Defines a gap or friction element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3	CID	

EXAMPLE:

CGAP	17	2	110	112	5.2	0.3	-6.1		

ALTERNATE FORMAT AND EXAMPLE:

CGAP	EID	PID	GA	GB	G0			CID	
CGAP	17	2	110	112	13				

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PGAP entry. (Integer > 0; Default = EID)
GA, GB	Connected grid points at ends A and B. (Integers > 0; GA ≠ GB)
X1, X2, X3	Components of the orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. See Remark 2 . (Real)

Field	Contents
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. See Remark 2 . (Integer > 0)
CID	Element coordinate system identification number. CID must be specified if GA and GB are coincident (distance from GA to GB<10^{-4}), or if an interference condition is desired . See Remarks 2 and 6. (Integer ≥ 0 or blank)

REMARKS:

1. The CGAP element is treated as a nonlinear gap element when used in the nonlinear solutions 106, 129, 153, and 159, in which the gap conditions update as the nonlinear solution iterates.

   For linear solutions (solution 101 and consecutive solutions 103, 111 and 112) with contact defined (BCSET card exists), CGAP elements are treated as linear contact elements if the system cell (412) OLDGAPS is set to 0 (default). If the system cell OLDGAPS is set to 1, the CGAP will be treated the same as a linear spring element (as described below). See the chapter on surface contact in the *NX Nastran User's Guide* for more information.

   For linear solutions without contact defined, and for all other solution sequences, the CGAP will be treated the same as a linear spring element, and remains linear with the initial stiffness. This stiffness depends on the value for the initial gap opening (U0 field in the PGAP entry).

2. The gap element coordinate system is defined by one of two following methods:

   - If the coordinate system (CID field) is specified, the element coordinate system is established using that coordinate system, in which the element x-axis is in the T1 direction and the y-axis in the T2 direction. The orientation vector $\vec{v}$ will be ignored in this case. The basic coordinate system is used when CID=0.

     If an interference condition is desired, the CID positive x-axis (gap direction) should point from the "GB side" to the "GA-side" of the gap.

   - If the CID field is blank and the grid points GA and GB are not coincident (distance from A to B ≥ 10−4), then the line AB is the element x-axis and the orientation vector $\vec{v}$ lies in the x-y plane. See Figure 28 .

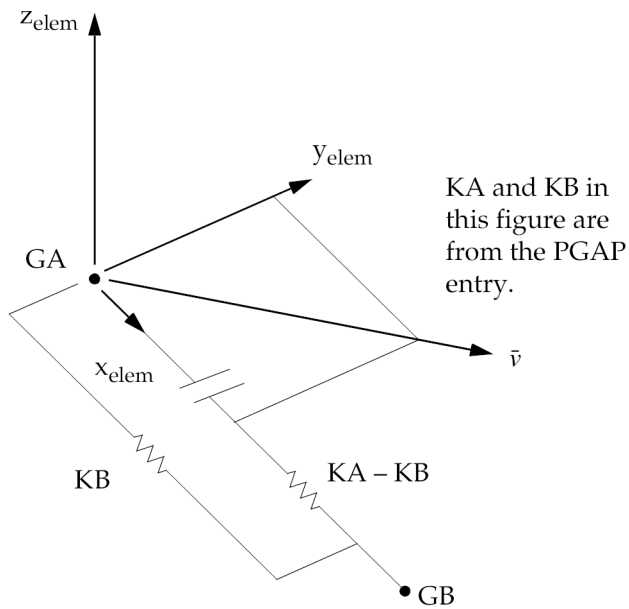


Figure 11-28. CGAP Element Coordinate System

3. The element coordinate system does not rotate as a result of deflections.

4. Initial gap openings are specified on the PGAP entry and not derived from the separation distance between GA and GB.

5. Forces, which are requested with the STRESS Case Control command, are output in the element coordinate system. F_x is positive for compression.

6. If CID is being used to define the element coordinate system and the CID refers to either a cylindrical or spherical coordinate system, then grid GA will be used to define the system. If grid GA lies on the z-axis of the cylindrical or spherical coordinate system it is recommended that a different coordinate system be used for this element.

7. See PARAM,CDITER for an alternative approach.

REMARKS RELATED TO SOLS 601 AND 701:

1. CGAP is simulated with a nonlinear spring element in SOLs 601 and 701, i.e., the transverse properties of this element are ignored. Please see SOL 601 and 701 remarks in the PGAP entry.

2. X1, X2, and X3 are ignored.

CHACAB

Acoustic Absorber Element Connection

Defines the acoustic absorber element in coupled fluid-structural analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHACAB	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

EXAMPLE:

CHACAB	95	12	1	2	5	7	8	9	
	24	23							

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PACABS entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1-G4 must be given in consecutive order about one quadrilateral face. G5-G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.

3. The mass is lumped to the face formed by grid points G5-G8 and G17-G20 and defined to be in contact with the fluid. The opposite face has no mass contribution due to the absorber element. Also, the face in contact with the fluid has only translational stiffness in the direction normal to the face.

4. Absorber modeling requirements:

   - The grid points defining G1-G20 must be structural grids. That is, the CD field on the GRID entry must *not* include a “-1”.

   - Grid points G1-G4, and if they exist, G9-G12 should be connected to the structure mesh.

   - Grids G5-G8, and if they exist, G17-G20 should be offset normal from the structure mesh. The offset amount is not significant to the absorber formulation and can be small but nonzero.

   - The element face formed by grids G5-G8, and if they exist, G17-G20 must be in proximity to the fluid face in order for the fluid-structure coupling to occur.

   - The edge grid points, G9-G20 are optional. They can be used when connecting to a structure mesh which includes edge grid points. If the ID of any edge connection grid point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. It is recommended that the edge grid points if they exist be located within the middle third of the edge.

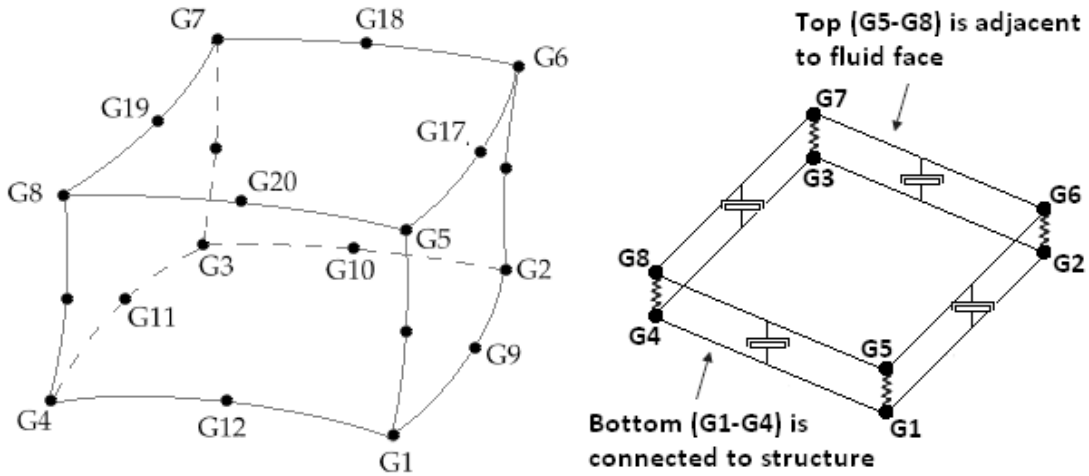


Figure 11-29. CHACAB Element Connection

CHACBR

Acoustic Barrier Element Connection

Defines the acoustic barrier element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHACBR	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

EXAMPLE:

CHACBR	95	12	1	2	5	7	8	9	
	24	23							

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PACBAR entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1-G4 must be given in consecutive order about one quadrilateral face. G5-G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.

3. The face consisting of grid points G1-G4 and G9-G12 if defined is assumed to be the backing that corresponds to MBACK on the PACBAR entry.

4. The face consisting of grid points G5-G8 and G17-G20 if defined is assumed to be the septum that corresponds to MSEPTM on the PACBAR entry. The grid points connected to this face have only translational stiffness in the direction normal to the face.

5. Barrier modeling requirements:

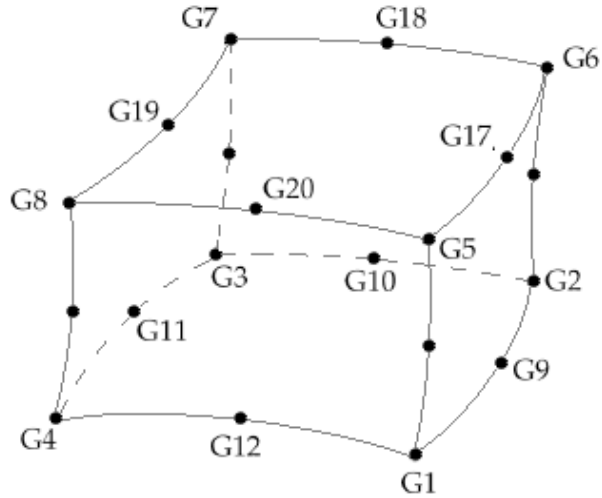
   - The grid points defining G1-G20 must be structural grids. That is, the CD field on the GRID entry must *not* include a “-1”.

   - Grid points G1-G4, and if they exist, G9-G12 should be connected to the structure mesh.

   - Grids G5-G8, and if they exist, G17-G20 should be offset normal from the structure mesh. The offset amount is not significant to the barrier formulation and can be small but nonzero.

   - The element face formed by grids G5-G8, and if they exist, G17-G20 must be in proximity to the fluid face in order for the fluid-structure coupling to occur.

   - The edge grid points G9-G20 are optional. They can be used when connecting to a structure mesh which includes edge grid points. If the ID of any edge connection grid point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. It is recommended that the edge grid points if they exist be located within the middle third of the edge.



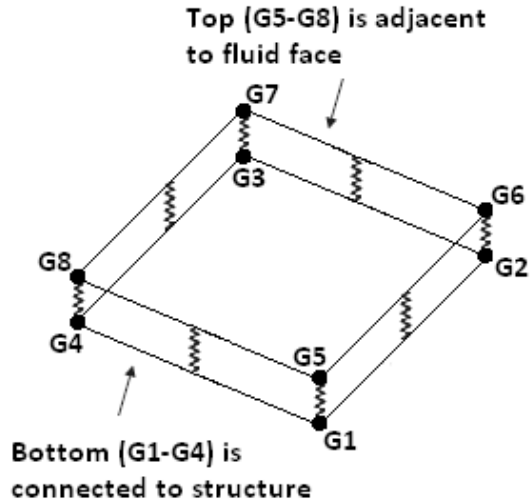


Figure 11-30. CHACBR Element Connection

CHBDYE

Geometric Surface Element Definition (Element Form)

Defines a boundary condition surface element with reference to a heat conduction element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		

EXAMPLE:

CHBDYE	2	10	1	3	3	2	2		

FIELDS:

Field	Contents
EID	Surface element identification number for a specific side of a particular element. See Remarks 1 and 10. (Unique Integer > 0 among all elements)
EID2	A heat conduction element identification number. (Integer > 0)
SIDE	A consistent element side identification number. See Remarks 6 , 7 , 8, and 9. (1 ≤ Integer ≤ 6)
IVIEWF	A VIEW entry identification number for the front face of the surface element. (Integer > 0, see Remark 2 for the default.)
IVIEWB	A VIEW entry identification number for the back face of the surface element. (Integer > 0, see Remark 2 for the default.)
RADMIDF	RADM identification number for the front face of surface element. (Integer ≥ 0, see Remark 2 for the default.)

Field	Contents
RADMIDB	RADM identification number for the back face of the surface element. (Integer ≥ 0, see Remark 2 for the default.)

REMARKS:

1. EID is a unique elemental ID associated with a particular surface element. EID2 identifies the general heat conduction element being considered for this surface element.

2. The defaults for IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank both on the CHBDYE entry and the BDYOR entry, then the default is zero.

3. For the front face of shell elements, the right-hand rule is used as one progresses around the element surface from G1 to G2 to ... Gn. For the edges of shell elements or the ends of line elements, an outward normal is used to define the front surface. For example, when a radiation boundary condition is defined on SIDE=1 of a shell element, a positive EID indicates that IVIEWF is relative to the top of the shell.

4. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.

5. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.

6. Side selection for solid elements:

   The sides of the solid elements are numbered consecutively according to the order of the grid point numbers on the solid element entry. The sides of solid elements are either quadrilaterals or triangles. For each element type, tabulate the grid points (gp) at the corners of each side.

Table 11-2. 8-node or 20-node CHEXA

side	gp	gp	gp	gp
1	4	3	2	1
2	1	2	6	5
3	2	3	7	6

Table 11-2. 8-node or 20-node CHEXA

side	gp	gp	gp	gp
4	3	4	8	7
5	4	1	5	8
6	5	6	7	8

Table 11-3. CPENTA

side	gp	gp	gp	gp
1	3	2	1	
2	1	2	5	4
3	2	3	6	5
4	3	1	4	6
5	4	5	6	

Table 11-4. CPYRAM

side	gp	gp	gp	gp
1	4	3	2	1
2	1	2	5	
3	2	3	5	
4	3	4	5	
5	4	1	5	

Table 11-5. CTETRA

side	gp	gp	gp
1	3	2	1
2	1	2	4
3	2	3	4
4	3	1	4

7. Side selection for shell elements:

SIDE=1 selects the face area of the element defined by the corner grids in the connectivity. SIDE = 2 through 4 for a TRI or 2 through 5 for a QUAD select the individual edge areas around the perimeter such that SIDE=2 selects the edge between the first and second grid points, SIDE=3 selects the edge between the second and the third, and so on. See "CHBDYG" for surface type definition. The thickness is that of the shell element. Note that midside grids are ignored. See Remark 3.

8. Side selection for line elements:

   SIDE=1 selects the outside surface area of the element between the end grid points excluding the end areas. SIDE=2 and SIDE=3 select the end areas at the first and second grid points in the element connectivity, respectively. The area calculation is consistent with the element geometry.

9. Side selection for axisymmetric elements:

   The axisymmetric elements CTRAX3, CTRAX6, CQUADX4, and CQUADX8 have revolved sides. SIDE=1 is determined by the element connectivity in the table below. The other sides are determined by the same connectivity pattern around the perimeter.

Table 11-6. Axisymmetric rev sides

Element	Number of rev sides	SIDE=1
CTRAX3	3	G1 and G2
CTRAX6	3	G1, G4, and G2
CQUADX4	4	G1 and G2
CQUADX8	4	G1, G5, and G2

10. The EID field on all CHBDYi entries is entered as positive integer, but the EID referenced on certain load or boundary condition entries can be entered as a +EID or -EID to represent the front or back of the CHBDYi entry, respectively. For example, entering a -EID on a QVECT entry applies a heat flux opposite to the CHBDYi normal orientation vector. The table below summarizes the thermal load and boundary conditions, if they support CHBDYi entires, and if so, if they support a negative EID.

Loads	Description	CHBDYi Supported	Negative EID Supported
QVECT	Directional heat flux from a distant source.	Yes	Yes
QVOL	Volumetric internal heat generation.	No	
QHBDY	Heat flux applied to an area defined on grid points.	No	

QBDY1	Heat flux applied to surface elements.	Yes	Yes
QBDY2	Heat flux applied to grid points associated with a surface element.	Yes	No
QBDY3	Heat flux applied to surface elements with control node capability.	Yes	Yes
SLOAD	Power into a grid or scalar point.	No	
NOLIN1	Nonlinear transient load as a tabular function.	No	
NOLIN2	Nonlinear transient load as a product of two variables.	No	
NOLIN3	Nonlinear transient load as a positive variable raised to a power.	No	
NOLIN4	Nonlinear transient load as a negative variable raised to a power.	No	
Boundary Conditions			
CONV	CONV Free convection	Yes	No
CONVM	CONVM Forced convection (fluid “element”)	Yes, but CHBDYP only	No
RADBC	Radiation exchange with space	Yes	Yes*
RADSET	Radiation exchange within an enclosure	No	
RADLST	Listing of Enclosure Radiation Faces	Yes	Yes*

* RADBC and RADLST entries do support a negative EID, although the front and back IVIEWi and RADMIDi fields already allow for this control, and a negative EID is typically not needed. IVIEWF and RADMIDF are associated with the CHBDYi normal orientation vector and IVIEWB and RADMIDB with the opposite. For radiation problems, if the RADMIDF or RADMIDB is zero, default radiant properties assume perfect black body behavior.

REMARKS RELATED TO SOL 601:

1. CHBDYE defines convection or radiation boundary condition on a side of 3D solid, shell or axisymmetric element for SOL 601,153 or SOL 601,159 analysis.

2. IVIEWF and IVIEWB are ignored.

3. RADMIDB applies only for radiation boundary condition applied on shell element.

4. If more than one CHBDYE or CHBDYG entry defines the same element side with the same material property and ambient grid point, only one of the entries will be used. A warning will be issued in such cases.

CHBDYG

Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

EXAMPLE:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

FIELDS:

Field	Contents
EID	Surface element identification number. (Unique Integer > 0 among all elemental entries)
TYPE	Surface type. See Remark 3 . (Character)
IVIEWF	A VIEW entry identification number for the front face. (Integer > 0; see Remark 2 for the default.)
IVIEWB	A VIEW entry identification number for the back face. (Integer > 0; see Remark 2 for the default.)
RADMIDF	RADM identification number for front face of the surface element. (Integer > 0; see Remark 2 for the default.)
RADMIDB	RADM identification number for the back face ofthe surface element. (Integer ≥ 0; see Remark 2 for the default.)

Field	Contents
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

REMARKS:

1. EID is a unique ID associated with a particular surface element as defined by the grid points.

2. The defaults for TYPE, IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYG entry and the BDYOR entry, then the default is zero.

3. TYPE specifies the kind of element surface; allowed types are: REV, AREA3, AREA4, AREA6, and AREA8. See Figure 11-31, Figure 11-33, and Figure 11-34.

   - TYPE = REV

     The “REV” type, which is used with elements CTRAX3, CTRAX6, CQUADX4, and CQUADX8, has two primary grid points whose location depends on the element orientation.

     o When the elements are oriented on the XZ plane, the two primary grid points must lie on the XZ plane of the basic coordinate system with X > 0. A midside grid point G3 is optional and supports convection or heat flux from the edge of elements CTRAX6 and CQUADX8. The defined area is a conical section with Z as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux. Automatic view factor calculations with VIEW data are not supported for the REV option.

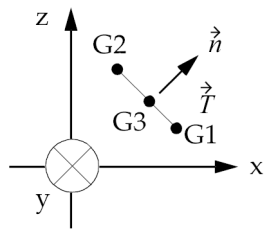


Figure 11-31. Normal Vector for CHBDYG Element of Type “REV”, XZ plane orientation

The unit normal lies in the XZ plane, and is given by

$$\vec{n} = (\vec{e}_y \times \vec{T}) / \left|\vec{e}_y \times \vec{T}\right| .$$

$\vec{e}_y$ is the unit vector in the Y direction.

o When the elements are oriented on the XY plane, the two primary grid points must lie on the XY plane of the basic coordinate system with Y > 0. A midside grid point G3 is optional and supports convection or heat flux from the edge of elements CTRAX6 and CQUADX8. The defined area is a conical section with X as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux. Automatic view factor calculations with VIEW data are not supported for the REV option.

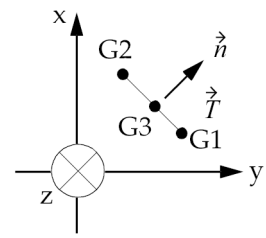


Figure 11-32. Normal Vector for CHBDYG Element of Type “REV”, XY plane orientation

The unit normal lies in the XY plane, and is given by

$$\vec{n} = (\vec{e}_Z \times \vec{T}) / \left|\vec{e}_Z \times \vec{T}\right|$$

$\vec{e}_Z$ is the unit vector in the Z direction.

- TYPE = AREA3, AREA4, AREA6, or AREA8

  These types have three and four primary grid points, respectively, that define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property entry is required for convection, radiation, or thermal vector flux.

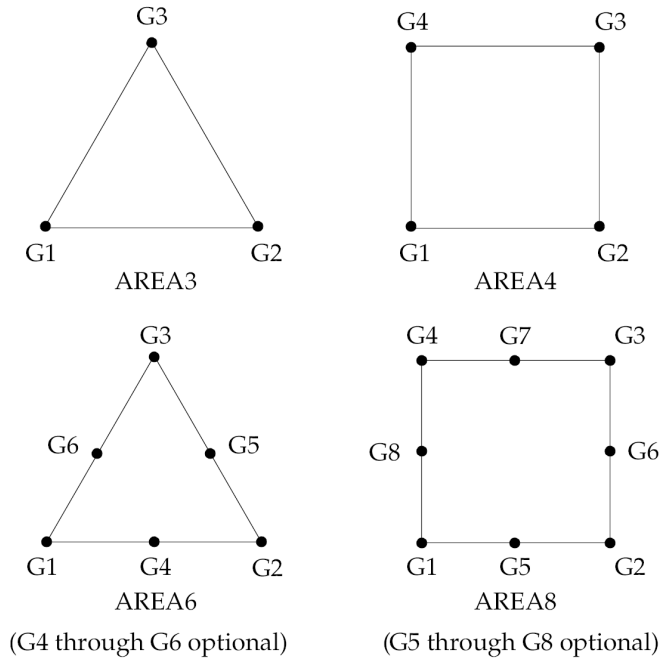


Figure 11-33. TYPE Examples

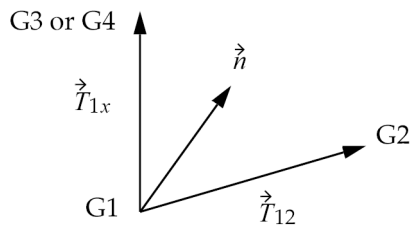


Figure 11-34. Normal Vector for CHBDYG Element of Types "AREAi"

The unit normal vector is given by

$$\vec{n} = \frac{(\vec{T}_{12} \times \vec{T}_{1x})}{\left|\vec{T}_{12} \times \vec{T}_{1x}\right|}$$

(G3 is used for triangles, and G4 is used for quadrilaterals.)

4. For defining the front face, the right-hand rule is used on the sequence G1 to G2 to ... Gn of grid points.

5. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.

6. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.

7. See the remarks on the CHBDYE bulk entry for application of boundary conditions using CHBDYG entries and a discussion of front and back faces.

REMARKS RELATED TO SOL 601:

1. CHBDYG defines convection or radiation boundary condition using grid points on a face of 3D solid or shell element or on an edge of axisymmetric element for SOL 601,153 or SOL 601,159 analysis.

2. IVIEWF and IVIEWB are ignored.

3. RADMIDB applies only for radiation boundary condition applied on shell element.

4. If more than one CHBDYE or CHBDYG entry defines the same element side with the same material property and ambient grid point, only one of the entries will be used. A warning will be issued in such cases.

CHBDYP

Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

EXAMPLE:

CHBDYP	2	5	POINT	2	2	101		500	
	3	3			0.0	0.0	1.0		

FIELDS:

Field	Contents
EID	Surface element identification number. (Unique Integer > 0 among all element identification numbers)
PID	PHBDY property entry identification numbers. (Integer > 0)
TYPE	Surface type. See Remark 3 . (Character).
IVIEWF	VIEW entry identification number for the front face. (Integer > 0 or blank)
IVIEWB	VIEW entry identification number for the back face. (Integer > 0 or blank)
G1, G2	Grid point identification numbers of grids bounding the surface. (Integer > 0)

Field	Contents
GO	Orientation grid point. (Integer ≥ 0; Default = 0)
RADMIDF	RADM entry identification number for the front face. (Integer ≥ 0 or blank)
RADMIDB	RADM entry identification number for the back face. (Integer ≥ 0 or blank)
GMID	Grid point identification number of a midside node if it is used with the line type surface element.
CE	Coordinate system for defining orientation vector. (Integer ≥ 0; Default = 0)
Ei	Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank)

REMARKS:

1. EID is a unique ID associated with a particular surface element as defined by the grid point(s).

2. The defaults for PID, TYPE, IVIEWF, IVIEWB, GO, RADMIDF, RADMIDB, CE, and Ei may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYP entry and the BDYOR entry, then the default is zero.

3. TYPE specifies the kind of element surface; the allowed types are: “POINT,” “LINE,” “ELCYL,” “FTUBE,” and “TUBE.” For TYPE = “FTUBE” and TYPE = “TUBE,” the geometric orientation is completely determined by G1 and G2; the GO, CE, E1, E2, and E3 fields are ignored.

   - TYPE = “POINT”

     TYPE = “POINT” has one primary grid point, requires a property entry, and the normal vector Vi must be specified if thermal flux is to be used.

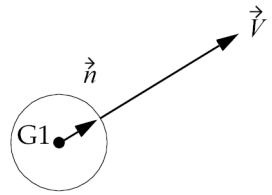


Figure 11-35. Normal Vector for CHBDYP Element of Type “POINT” (See Remarks 4 and 5.)

The unit normal vector is given by

$$\vec{n} = \vec{V} / \left|\vec{V}\right|$$

where

$\vec{V}$ is specified in the Ei field and given in the basic system at the referenced grid point. See Remarks 4 and 5 for the determination of $\vec{V}$.

- TYPE = “LINE,” “FTUBE,” or “TUBE”

  The TYPE = “LINE” type has two primary grid points, requires a property entry, and the vector is required. TYPE = “FTUBE” and TYPE = “TUBE” are similar to TYPE = “LINE” except they can have linear taper with no automatic view factor calculations. GMID is an option for the TYPE = “LINE” surface element only and is ignored for TYPE = “FTUBE” and “TUBE”.

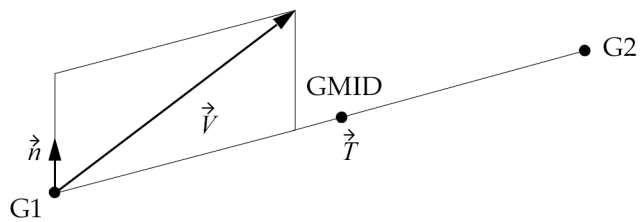


Figure 11-36. Normal Vector for CHBDYP Element with TYPE=“LINE”,TYPE=“FTUBE”, or TYPE=“TUBE”(See Remarks 4 and 5.)

The unit normal lies in the plane $\vec{V}$ and $\vec{T}$, is perpendicular to $\vec{T}$, and is given by:

$$\vec{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{\left|\vec{T} \times (\vec{V} \times \vec{T})\right|}$$

- TYPE = “ELCYL”

  TYPE = “ELCYL” (elliptic cylinder) has two connected primary grid points and requires a property entry. The vector must be nonzero. Automatic view factor calculations are not available.

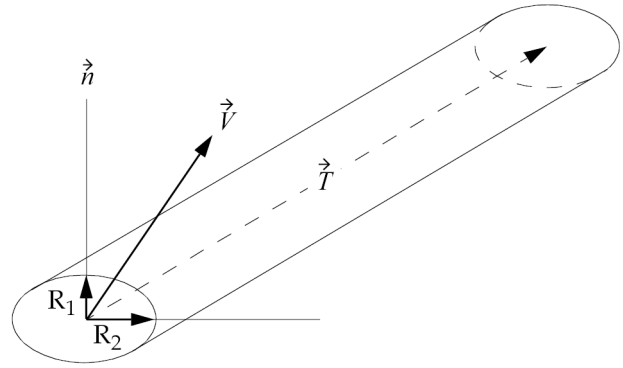


Figure 11-37. Normal Vector for CHBDYP Element of TYPE=“ELCYL”(See Remarks 4 and 5.)

The same logic is used to determine $\vec{n}$ as for TYPE = LINE. The “radius” R_1 is in the $\vec{n}$ direction, and R2 is the perpendicular to $\vec{n}$ and $\vec{T}$ (see fields 7 and 8 of the PHBDY entry).

4. For TYPE = “POINT,” TYPE = “LINE,” and TYPE = “ELCYL,” geometric orientation is required. The required information is sought in the following order:
   - If GO > 0 is found on the CHBDYP entry, it is used.
   - Otherwise, if a nonblank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
   - If neither of the aboveis found, the same information is sought in the same way from the BDYOR entry.
   - If none of the above apply, a warning message is issued.
5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.

- If GO > zero:

  For a TYPE = “POINT” surface, the normal to the front face is the vector from G1 to GO. For the TYPE = “LINE” surface, the plane passes through G1, G2, GO and the right-hand rule is used on this sequence to get the normal to the front face. For a TYPE = “ELCYL” surface the first axis of the ellipse lies on the G1, G2, GO plane, and the second axis is normal to this plane. For TYPE = “FTUBE” or “TUBE” surface, no orientation is required, and GO is superfluous.

- If GO is zero:

  For a TYPE = “POINT” surface, the normal to the front face is the orientation vector. For the TYPE = “LINE” surface, the plane passes through G1, G2, and the orientation vector; the front face is based on the right-hand rule for the vectors G2-G1 and the orientation vector. For a TYPE = “ELCYL” surface, the first axis of the ellipse lies on the G1, G2, orientation vector plane, and the second axis is normal to this plane.

6. The continuation entry is optional.

7. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.

8. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the following surface element entries: CHBDYE, CHBDYG, or CHBDYP.

CHEXA

Six-Sided Solid Element Connection

Defines the connections of the six-sided solid element with eight to twenty grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

EXAMPLE:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	0	0	30	31	53	54	
	55	56	57	58	59	60			

FIELDS:

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID, PLSOLID, PCOMPS, or PMIC entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

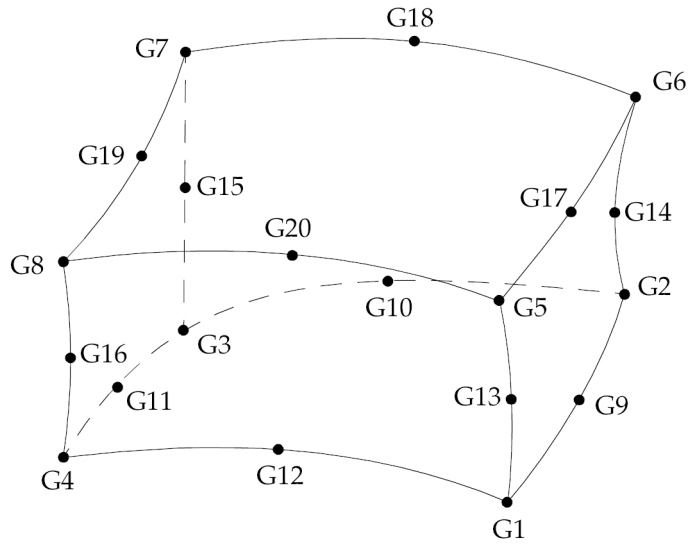


Figure 11-38. CHEXA Element Connection

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.

3. The edge points G9 to G20 are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.

4. Components of stress are output in the material coordinate system. See the remarks on the PSOLID bulk entry for hyperelastic and nonlinear exceptions. See the remarks on the PCOMPS bulk entry for composite laminate exception.

5. The second continuation is optional.

6. Except when used as a hyperelastic element or as a composite laminate solid element, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.

   R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.

   S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.

   T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

   The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system, a 3 x 3 positive-definite symmetric matrix would be produced.)

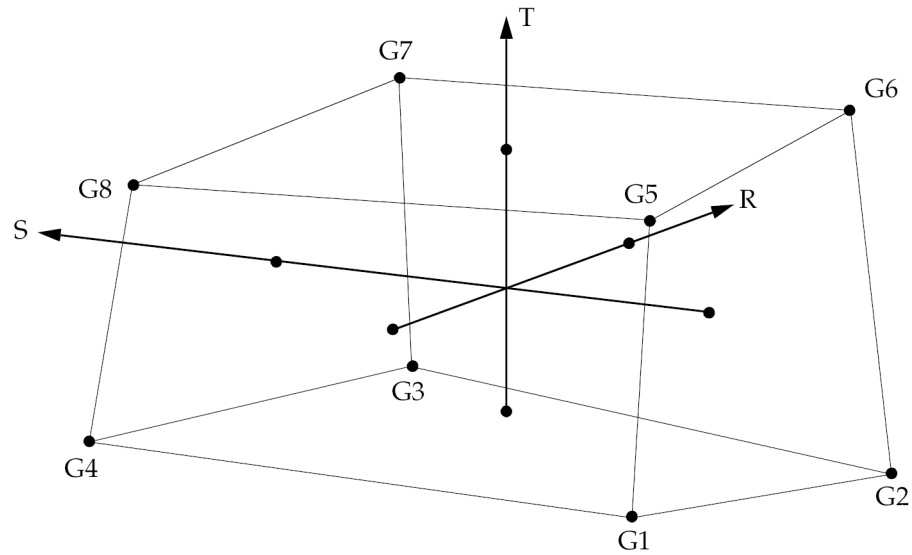


Figure 11-39. CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.

8. For hyperelastic elements, the plot codes are specified under the CHEXAFD element name in “Item Codes”.

9. By default, all twelve edges of the element are considered straight unless any of G9 through G20 are specified.

10. Collapsed CHEXA elements are supported for use in crack simulations using SOL 401 and a third-party software. To allow for collapsed CHEXA elements, specify PARAM,COLPHEXA,YES. Note that the collapsed CHEXA element is not supported in a glue or contact region.

    Any face of a CHEXA element can be collapsed to an edge. The edge of the collapsed face represents the crack front.

    For example, Figure 11-40 shows the CHEXA element of Figure 11-38 with the G2–G14–G6–G18–G7–G15–G3–G10 face collapsed so that the G2–G14–G6 edge and the G3–G15–G7 edge become the crack front. Alternately, the G2–G14–G6–G18–G7–G15–G3–G10 face could be collapsed so that the G2–G10–G3 edge and the G6–G18–G7 edge would become the crack front.

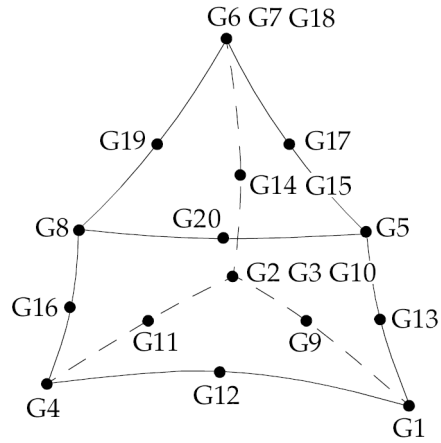


Figure 11-40. Collapsed CHEXA Element

Two options are available for specifying a CHEXA element with a collapsed face:

- In Format 1, 15 unique grid IDs are specified in the 20 grid ID fields of the CHEXA bulk entry. Format 1 is typically used for elastic material models. With Format 1, mid-side grids can move to the quarter-span locations closest to the crack front.

  For the collapsed CHEXA element shown in Figure 11-40, the Format 1 specification is as follows:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G2	G4	G5	G6	
	G6	G8	G9	G2	G11	G12	G13	G14	
	G14	G16	G17	G6	G19	G20			

where the same grid ID is entered in the G2, G3, and G10 fields, another grid ID is entered in both the G14 and G15 fields, and another grid ID is entered in the G6, G7, and G18 fields.

- In Format 2, 20 unique grid IDs are specified in the 20 grid ID fields of the CHEXA bulk entry. However, eight of the grid IDs do not have unique coordinates. Format 2 is typically used for elasto-plastic material models. With Format 2, mid-side grids remain at the mid-span locations.

  For the collapsed CHEXA element shown in Figure 11-40, the Format 2 specification is as follows:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

  where the grids entered in the G2, G3, and G10 fields would share the same coordinates, the grids entered in the G14 and G15 fields would share the same coordinates, and the grids entered in the G6, G7, and G18 fields would share the same coordinates. Unlike Format 1 where grids in the CHEXA element connectivity are merged, Format 2 does not merge coincident grids in the CHEXA element connectivity. Thus, these grids can move independently of one another.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 601, only elements with 8 or 20 grid points are allowed, i.e., either all edge points G9 to G20 are specified or no edge points are specified. For SOL 701, only elements with 8 grid points are allowed.

2. For SOL 601, 20-node CHEXA elements may be converted to 27-node CHEXA elements (6 additional nodes on the centroid of the six faces and 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. 27-node CHEXA elements are especially effective in the analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

CHEXCZ

Six-Sided Cohesive Element Connection

Defines the connections of the six-sided cohesive element with eight to twenty grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CHEXCZ	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

EXAMPLE:

CHEXCZ	15	1	3	4	5	6	7	8	
	9	10		0	30	31	53	54	
	55	56	57	58	59	60			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLCZ. (Integer > 0; No default)
Gi	Grid point identification numbers of connection points. (Integer > 0 or blank; See Remark 3 for default behavior)

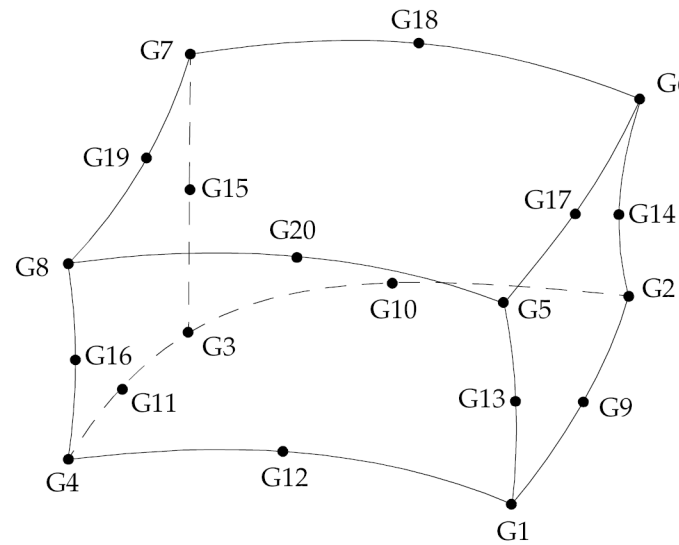


Figure 11-41. CHEXCZ Element Connection

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. This is the bottom face of the element. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, G7 opposite G3, and G8 opposite G4. This is the top face of the element. The normal direction is defined relative to the bottom surface.

3. The edge points G9 to G20 are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.

4. If G15 through G20 are omitted, the second continuation line is not required.

5. For best results, locate edge grid points within the middle third of the edge.

6. For information regarding the material coordinate system, see PSOLCZ.

CLOAD

Static Load Combination for Superelement Loads (Superposition)

Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).

FORMAT:

1	2	3	4	5	6	7	8	9	10
CLOAD	CID	S	S1	IDV1	S2	IDV2	S3	IDV3	
	S4	IDV4	-etc.-						

EXAMPLE:

CLOAD	25	1.0	25.0	10	-1.0	101	2.2-1	604	
	-62.0	62							

FIELDS:

Field	Contents
CID	Combination identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
IDVi	Identification numbers of load vectors (EXCITEID of a selected LSEQ entry) calculated for a superelement loads entry. (Integer > 0)

REMARKS:

1. The CLOAD entry must be selected in the residual solution subcase of the case control with CLOAD = CID and must be used if loads are applied to upstream superelements in SOL 106 or 153.

2. The load vector defined is given by

$$\{P\} = S\sum_{i} Si\{P_{\mathrm{IDVi}}\}$$

3. The IDVi field refers to a previously calculated load vector for the superelement via the LSEQ approach. That is, a LOADSET keyword must have been selected in Case Control that in turn refers to one or more LSEQ entries in the Bulk Data Section. The IDVi refers to the EXCITEID of such LSEQ entries. For more details, see the “LSEQ” and the “LOADSET”.

4. In the CID or IDV fields, a CLOAD entry may not reference an identification number defined by another CLOAD entry.

CMASS1

Scalar Mass Connection

Defines a scalar mass element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

EXAMPLE:

CMASS1	32	6	2	1					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PMASS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid or scalar point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS3 entry.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. The two connection points (G1, C1) and (G2, C2) must not be coincident.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

7. Scalar elements input coupled mass matrices when the second pair of fields is entered. When uncoupled point masses are desired input only the first pair of fields. When a coupled mass matrix is requested the submatrix added has M on the diagonal, and -M on the off-diagonal. The element is not checked for internal constraints, which is the user’s responsibility if desired. There are instances where elements with internal constraints are desired, although not frequently. To identify the presence of internal constraints caused by coupled mass, inspect GPWG output, OLOAD output due to GRAV loads, and rigid body modes of free structures. Some forms of coupled mass will cause coupling of rigid body translational mass terms in GPWG output, and poor rigid body modes in modal analysis.

REMARKS RELATED TO SOLS 601 AND 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. If G1 and G2 are specified, CMASS1 models a mass between G1 and G2 in SOL 601. Note that this is different from how it is treated in other solution sequences. Please see the *Advanced Nonlinear Theory and Modeling Guide*.

CMASS2

Scalar Mass Property and Connection

Defines a scalar mass element without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2			

EXAMPLE:

CMASS2	32	9.25	6	1					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
M	Value of the scalar mass. (Real)
G1, G2	Geometric grid or scalar point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point.)

REMARKS:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS4 entry.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. The two connection points (G1, C1) and (G2, C2) must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for Gi and Ci.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

7. See Remark 7 for “CMASS1”.

REMARKS RELATED TO SOLS 601 AND 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. If G1 and G2 are specified, CMASS1 models a mass between G1 and G2 in SOL 601. Note that this is different from how it is treated in other solution sequences. Please see the *Advanced Nonlinear Theory and Modeling Guide*.

CMASS3

Scalar Mass Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

EXAMPLE:

CMASS3	13	42	62						

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PMASS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMASS4

Scalar Mass Property and Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

EXAMPLE:

CMASS4	23	14.92		23					

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
M	Scalar mass value. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

REMARKS:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate. This is the usual case.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.

4. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

5. A scalar point specified on this entry need not be defined on an SPOINT entry.

Chapter 12: Bulk Data Entries CO—CY

CONM1

Concentrated Mass Element Connection, General Form

Defines a 6 x 6 symmetric mass matrix at a geometric grid point.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

EXAMPLE:

CONM1	2	22	2	2.9	6.3				
	4.8	28.6							
		28.6						28.6	

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number for the mass matrix. (Integer ≥ 0)
Mij	Mass matrix values. (Real)

REMARKS:

1. For a less general means of defining concentrated mass at grid points, see the CONM2 entry description.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. The mass matrix has the form:

$$\begin{bmatrix} M_{11} & & & & & \\ M_{21} & M_{22} & & \text{symmetric} & & \\ M_{31} & M_{32} & M_{33} & & & \\ M_{41} & M_{42} & M_{43} & M_{44} & & \\ M_{51} & M_{52} & M_{53} & M_{54} & M_{55} & \\ M_{61} & M_{62} & M_{63} & M_{64} & M_{65} & M_{66} \end{bmatrix}$$

REMARKS RELATED TO SOLS 601 AND 701:

1. Only the diagonal terms of the mass matrix Mii ($1 \le i \le 6$) are used.

2. CID is ignored, i.e., mass components Mii are assumed to be in the displacement coordinate system of grid point G.

CONM2

Concentrated Mass Element Connection, Rigid Body Form

Defines a concentrated mass at a grid point.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

EXAMPLE:

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. For CID of -1 see X1, X2, X3 below. (Integer ≥ -1; Default = 0)
M	Mass value. (Real)
X1, X2, X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real)
Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (For I11, I22, and I33; Real ≥ 0.0; for I21, I31, and I32; Real)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.

3. The continuation is optional.

4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.

5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & \\ & M & & \text{symmetric} & & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -132 & I33 \end{bmatrix}$$

where

$$M = \int \rho dV$$

$$I11 = \int \rho(x_2^2 + x_3^2)dV$$

$$I22 = \int \rho(x_1^2 + x_3^2)dV$$

$$I33 = \int \rho(x_1^2 + x_2^2)dV$$

$$I21 = \int \rho(x_1 x_2)dV$$

$$I31 = \int \rho(x_1 x_3)dV$$

$$I32 = \int \rho(x_2 x_3)dV$$

and

x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4.

The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is nonpositive definite, since this may cause fatal errors in dynamic analysis modules.

6. If CID ≥ 0, then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.

7. See “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide* for a definition of coordinate system terminology.

REMARKS RELATED TO SOLS 601 AND 701:

1. X1, X2, and X3 are ignored, i.e., the center of gravity of the mass is assumed to be at the grid point.

2. CID is ignored, i.e., mass moments of inertia components Iii (1 ≤ i ≤ 3) are assumed to be in the displacement coordinate system of grid point G.

3. I21, I31, and I32 are ignored. The displacement coordinate system for grid point G should coincide with the principal axes of the concentrated mass.

CONROD

Rod Element Property and Connection

Defines a rod element without reference to a property entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

EXAMPLE:

CONROD	2	16	17	4	2.69				

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient for torsional stress determination. (Real)
NSM	Nonstructural mass per unit length. (Real)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. For structural problems, MID must reference a MAT1 material entry.

3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. J, C, and NSM are ignored.

2. For structural problems, MID must reference a MAT1, MATSMA, MATVE or MATPLCY material entry for SOL 601, and a MAT1 or MATPLCY material entry for SOL 701.

3. For heat transfer problems, MID must reference a MAT4 entry.

4. CONROD defines a truss element with no torsional stiffness, i.e., only axial force is transmitted by the element.

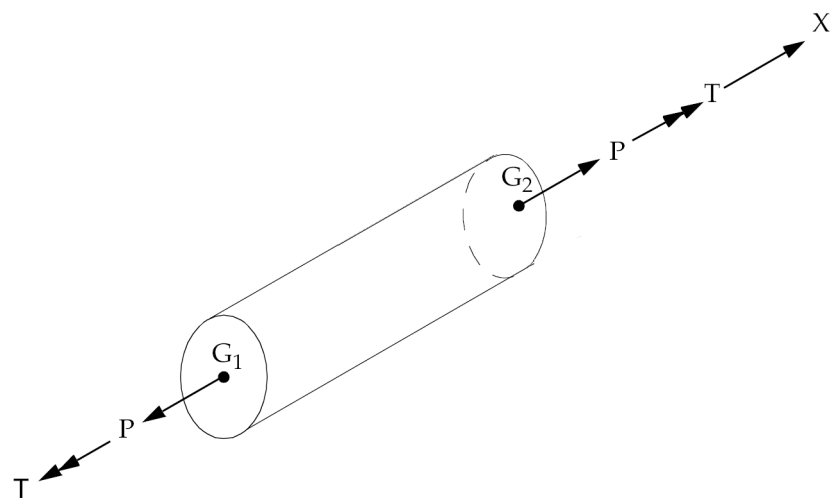


Figure 12-1. CONROD Element Forces and Moments

CONV

Heat Boundary Element Free Convection Entry

Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
	TA5	TA6	TA7	TA8					

EXAMPLE:

CONV	2	101	3	201	301				

FIELDS:

Field	Contents
EID	CHBDYG, CHBDYE, or CHBDYP surface element identification number. (Integer > 0)
PCONID	Convection property identification number of a PCONV entry. (Integer > 0)
FLMND	Point for film convection fluid property temperature. (Integer ≥ 0; Default = 0)
CNTRLND	Control point for free convection boundary condition. (Integer ≥ 0; Default = 0)
TAi	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2 through TA8; Default for TA2 through TA8 is TA1.)

REMARKS:

1. The basic exchange relationship can be expressed in one of the following forms:
   - $q = H \cdot (T - \text{TAMB})^{\text{EXPF}}(T - \text{TAMB})$, CNTRLND = 0
   - $q = (H \cdot u_{\text{CNTRLND}})(T - \text{TAMB})^{\text{EXPF}}(T - \text{TAMB})$, CNTRLND ≠ 0
   - $q = H \cdot (T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$, CNTRLND = 0
   - $q = (H \cdot u_{\text{CNTRLND}})(T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$, CNTRLND ≠ 0

     EXPF is specified on the PCONV entry.

     (See "PCONV" for additional clarification of forms.)
2. The continuation entry is not required.
3. CONV is used with an CHBDYi (CHBDYG, CHBDYE, or CHBDYP) entry having the same EID.
4. The temperature of the film convection point provides the look up temperature to determine the convection film coefficient. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperature, or the ambient temperature, as defined in the FORM field of the PCONV Bulk Data entry.
5. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If midside ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
6. See the Bulk Data entry, "PCONV" , for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.

REMARKS RELATED TO SOL 601:

1. The basic exchange relationship can be expressed in one of the following forms:
   - $q = H \cdot (T - \text{TAMB})$, CNTRLND = 0
   - $q = (H \cdot u_{\text{CNTRLND}})(T - \text{TAMB})$, CNTRLND ≠ 0
2. Only one ambient point TA1 is supported. TA2 to TA8 are ignored. The temperature on TA1 must be prescribed on a TEMPBC or SPC entry.

3. FLMND is ignored.

4. CNTRLND may be used to model time-dependent convection heat transfer coefficient. The temperature on CNTRLND must be prescribed on a TEMPBC or SPC entry.

5. H cannot be both time-dependent and temperature-dependent at the same time. Hence, temperature on CNTRLND cannot be time-dependent when H is made temperature-dependent by a MATT4 entry.

CONVM

Heat Boundary Element Forced Convection Entry

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
CONVM	EID	PCONID	FLMND	CNTMDOT	TA1	TA2			

EXAMPLE:

CONVM	101	1	201	301	20	21			

FIELDS:

Field	Contents
EID	CHBDYP element identification number. (Integer > 0)
PCONID	Convection property identification number of a PCONVM entry. (Integer > 0)
FLMND	Point used for fluid film temperature. (Integer ≥ 0; Default = 0)
CNTMDOT	Control point used for controlling mass flow. (Integer > 0)
TA1, TA2	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2; Default for TA2 is TA1.)

REMARKS:

1. CONVM is used with an CHBDYP entry of type FTUBE having the same EID.

2. The temperature of the fluid film point may be specified to determine the material properties for the fluid. If FLMND = 0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperatures, or the ambient temperature, as defined in the FORM field of the PCONVM Bulk Data entry.

3. CNTMDOT must be set to the desired mass flow rate (mdot) to effect the advection of energy downstream at an mdot · C_p · T rate. In addition to the effect that mdot has on the transfer of thermal energy in the streamwise direction, this control point value is also used in computing the tube Reynolds number and subsequently the forced convection heat transfer coefficient if requested. This enables the fluid stream to exchange heat with its surroundings.

4. If only the first ambient point is specified, then the second ambient point is assumed to have the same temperature.

5. See the Bulk Data entry, “PCONVM” , for an explanation of the mathematical relationships available for forced convection and the reference temperature for fluid material properties.

CORD1C

Cylindrical Coordinate System Definition, Form 1

Defines a cylindrical coordinate system using three grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

EXAMPLE:

CORD1C	3	16	32	19					

FIELDS:

Field	Contents
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A; G1B ≠ G2B ≠ G3B;)

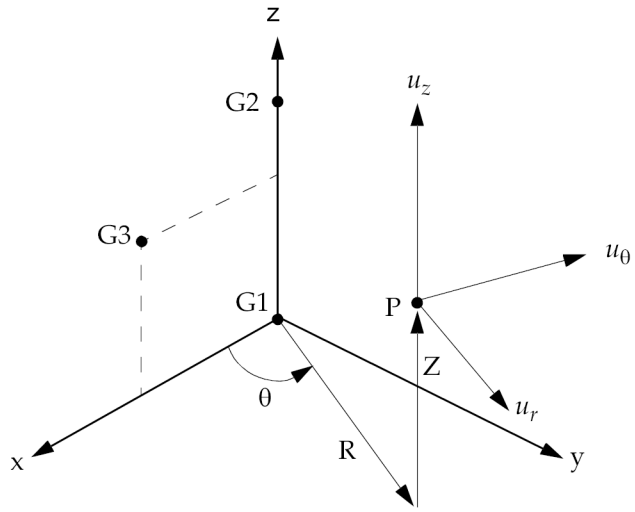


Figure 12-2. CORD1C Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be non-colinear and not coincident.

4. The location of a grid point (P in Figure 12-2) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.

5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).

6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide*.

CORD1R

Rectangular Coordinate System Definition, Form 1

Defines a rectangular coordinate system using three grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

EXAMPLE:

CORD1R	3	16	32	19					

FIELDS:

Field	Contents
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

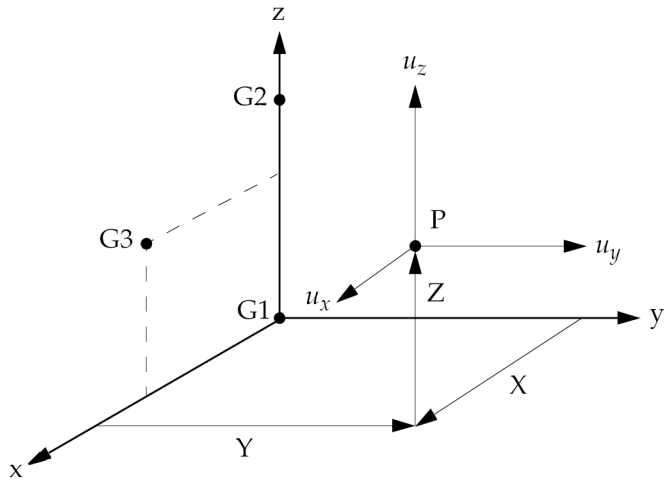


Figure 12-3. CORD1R Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The three grid points GiA (or GiB) must be noncolinear and not coincident.

4. The location of a grid point (P in Figure 12-3) in this coordinate system is given by (X, Y, Z).

5. The displacement coordinate directions at P are shown above by (u_x, u_u, u_z).

CORD1S

Spherical Coordinate System Definition, Form 1

Defines a spherical coordinate system by reference to three grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

EXAMPLE:

CORD1S	3	16	32	19					

FIELDS:

Field	Contents
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

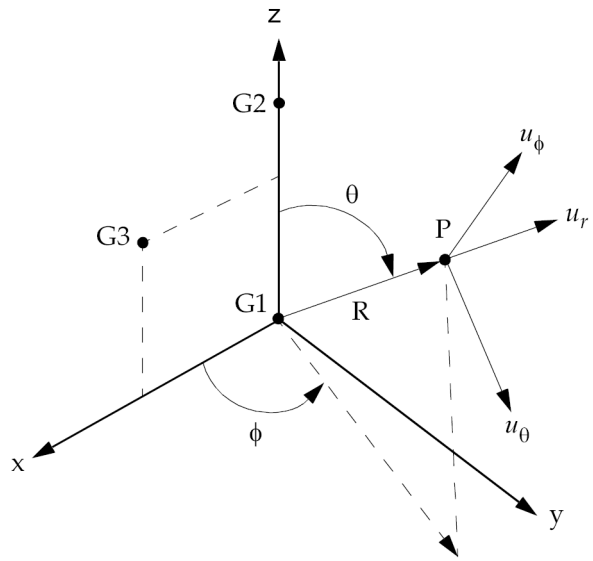


Figure 12-4. CORD1S Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with a definition that does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.

4. The location of a grid point (P in Figure 12-4) in this coordinate system is given by (R, θ, φ) where θ and φ are measured in degrees.

5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_φ).

6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide*.

CORD2C

Cylindrical Coordinate System Definition, Form 2

Defines a cylindrical coordinate system using the coordinates of three points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

EXAMPLE:

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

FIELDS:

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

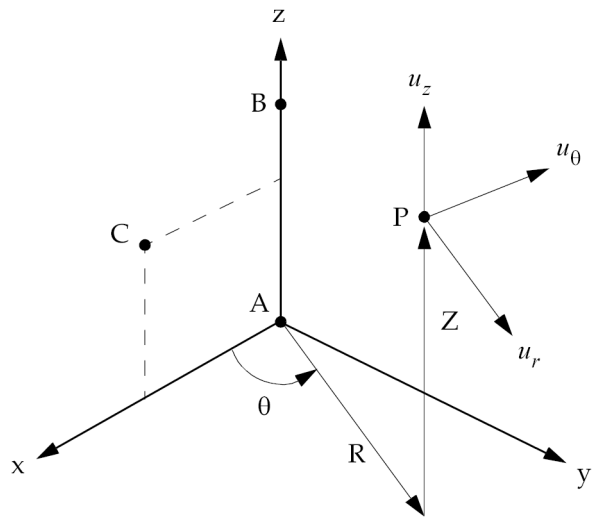


Figure 12-5. CORD2C Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.

2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Non-colinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

3. The continuation entry is required.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in Figure 12-5) in this coordinate system is given by (R, θ, Z), where θ is measured in degrees.

6. The displacement coordinate directions at P are dependent on the location of P as shown above by u_r, u_θ, u_z).

7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide*.

8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD2R

Rectangular Coordinate System Definition, Form 2

Defines a rectangular coordinate system using the coordinates of three points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

EXAMPLE:

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

FIELDS:

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0, which is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

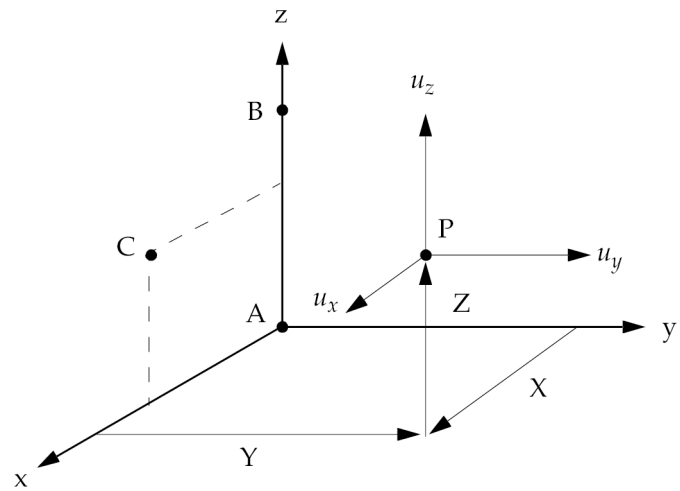


Figure 12-6. CORD2R Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must be unique.

2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.

3. The continuation entry is required.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in the Figure 12-6) in this coordinate system is given by (X, Y, Z).

6. The displacement coordinate directions at P are shown by (u_x, u_y, u_z).

7. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD2S

Spherical Coordinate System Definition, Form 2

Defines a spherical coordinate system using the coordinates of three points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

EXAMPLE:

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

FIELDS:

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

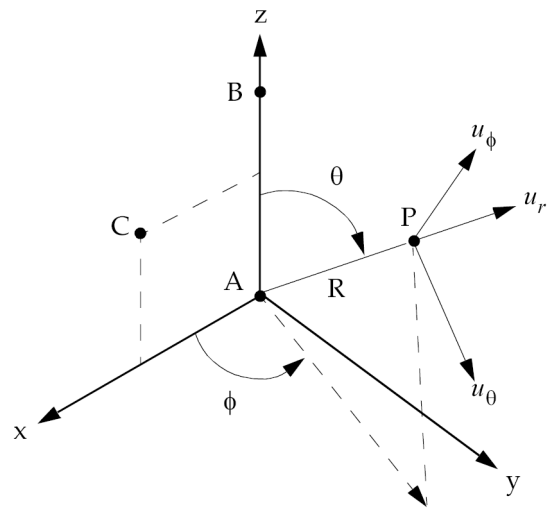


Figure 12-7. CORD2S Definition

REMARKS:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must all be unique.

2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

3. The continuation entry is required.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in Figure 12-7) in this coordinate system is given by (R, θ, φ), where θ and φ are measured in degrees.

6. The displacement coordinate directions at P are shown above by (u_r, u_θ, u_φ).

7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide*.

8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD3G

General Coordinate System

Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. The CORD3G entry is used with the MAT9 entry to orient material principal axes for 3-D composite analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CORD3G	CID	METHOD		THETAID1	THETAID2	THETAID3	CIDREF		

EXAMPLE:

CORD3G	100	E313		110	111	112	0		

FIELDS:

Field	Contents
CID	Coordinate system identification number. See Remark 1. (Integer > 0)
METHOD	E313 or S321 for Euler angle rotation in 313 sequence or space-fixed rotation in 321 sequence. See Remark 2. (Character; Default = “E313”)
THETAID	Identification number for DEQATN Bulk Data entry which defines the three angles (in radians) measured from reference coordinates to the general material coordinate system. See Remark 3. (Integer > 0)
CIDREF	Identification number for the coordinate system from which the orientation of the general coordinate system is defined. (Integer > 0)

REMARKS:

1. CID must be unique with respect to all other coordinate systems. CID cannot be referenced on GRID entries.

2. Three Euler angles specify the rotation of the CORD3G coordinate axes (xyz) with respect to the local Cartesian coordinate axes (XYZ) in CIDREF as follows: first rotate about Z-axis by q_1, next rotate about rotated x-axis by q_2, and then rotate about rotated z-axis by q_3. On the other hand, the space-fixed rotations in 321 sequence specify all the rotations about the fixed coordinate axes: first rotate about Z by q_1, next about Y by q_2, then about X by q_3.

3. The three rotations define a coordinate transformation which transforms position vectors in the reference coordinate system into the general coordinate system.

CPENTA

Five-Sided Solid Element Connection

Defines the connections of a five-sided solid element with six to fifteen grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

EXAMPLE:

CPENTA	112	2	3	15	14	4	103	115	
	5	16	8				120	125	
	130								

FIELDS:

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID, PLSOLID, PCOMPS, or PMIC entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

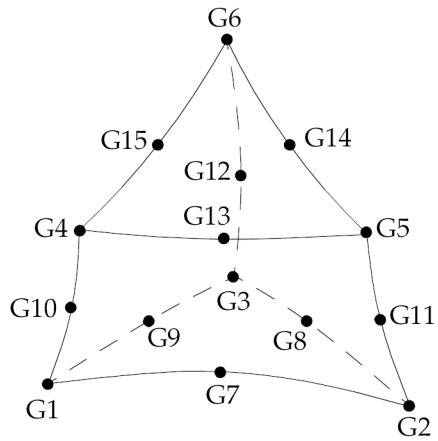


Figure 12-8. CPENTA Element Connection

REMARKS:

1. Element ID numbers must be unique with respect to all other element ID numbers.

2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.

3. The edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.

4. Components of stress are output in the material coordinate system. See remarks on the PSOLID bulk entry for hyperelastic and nonlinear exceptions. See remarks on the PCOMPS bulk entry for composite laminate exception.

5. Except when used as a hyperelastic element or as a composite laminate solid element, the element coordinate system for the CPENTA element is defined as follows: The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

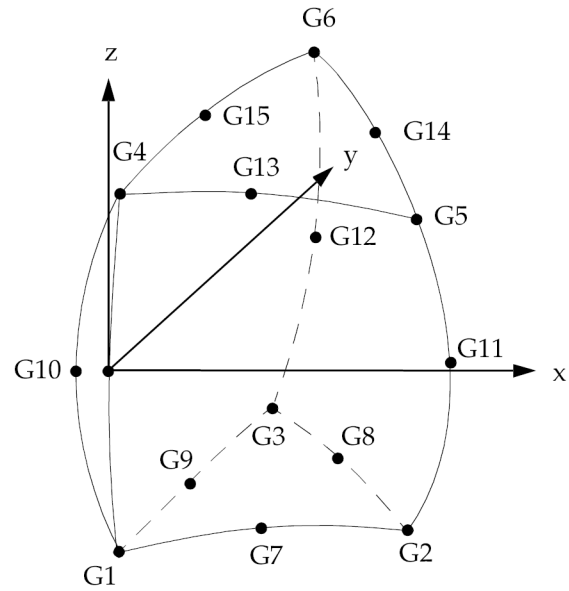


Figure 12-9. CPENTA Element Coordinate System

6. It is recommended that the edge grid points be located within the middle third of the edge.

7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in “Item Codes”.

8. By default, all of the nine edges of the element are considered straight unless any of G7 through G15 are specified.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 601, only elements with 6 or 15 grid points are allowed, i.e., either all edge points G7 to G15 are specified or no edge points are specified. For SOL 701, only elements with 6 grid points are allowed.

2. For SOL 601, 15-node CPENTA elements may be converted to 21-node CPENTA elements (5 additional nodes on the centroid of the five faces and 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. 21-node CPENTA elements are more effective than 15-node CPENTA for analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

CPENTCZ

Five-Sided Cohesive Element Connection

Defines the connections of a five-sided cohesive element with six to fifteen grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPENTCZ	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

EXAMPLE:

CPENTCZ	112	2	3	15	14	4	103	115	
	5	16	8	0			120	125	
	130								

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLCZ entry. (Integer > 0; No default)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank; See Remark 3 for default behavior)

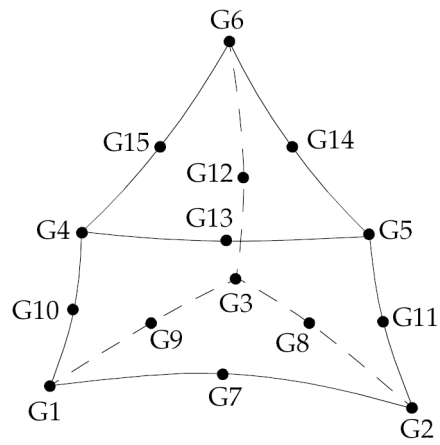


Figure 12-10. CPENTCZ Element Connection

REMARKS:

1. Element identification numbers must be unique with respect to all other element identification numbers.

2. Grid points G1 through G3 must define one triangular face. This is the bottom face. G4 through G6 must be on the opposite face with G4 opposite G1, G5 opposite G2, and G6 opposite G3. This is the top face. The normal direction is defined relative to the bottom surface.

3. The edge points G7 to G15 are optional. Any or all of them may be omitted. If the ID of any edge connection point is left blank or set to zero (G10, G11, and G12 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be omitted.

4. If all edge grid points are omitted, the first and second continuation lines are not required. If G15 is the only edge grid point that is omitted, the second continuation line is not required.

5. For best results, locate edge grid points within the middle third of the edge.

6. For information regarding the material coordinate system, see PSOLCZ.

CPLSTN3

Plane Strain Triangular Element Connection

Defines a plane strain triangular element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTN3	EID	PID	G1	G2	G3	THETA			

EXAMPLE:

CPLSTN3	111	203	31	74	75	30.0			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE, PLPLANE, or PGPLSN entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 8. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

4. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

5. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

6. GPSTRESS or GPSTRAIN output is not supported.

7. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

8. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 11 and 12 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

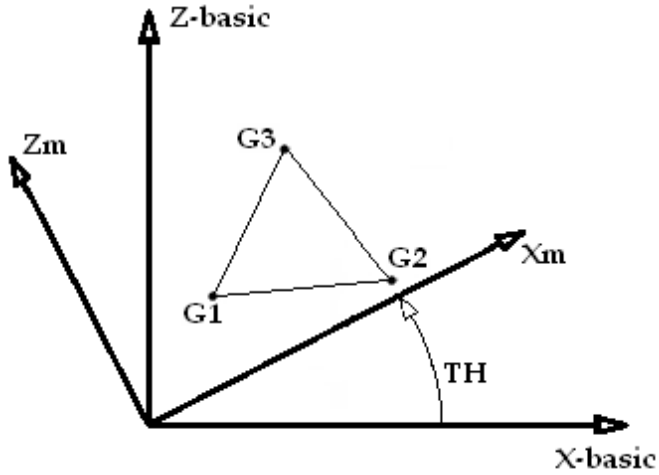


Figure 12-11. CPLSTN3 Element Geometry and Coordinate Systems, XZ-Plane

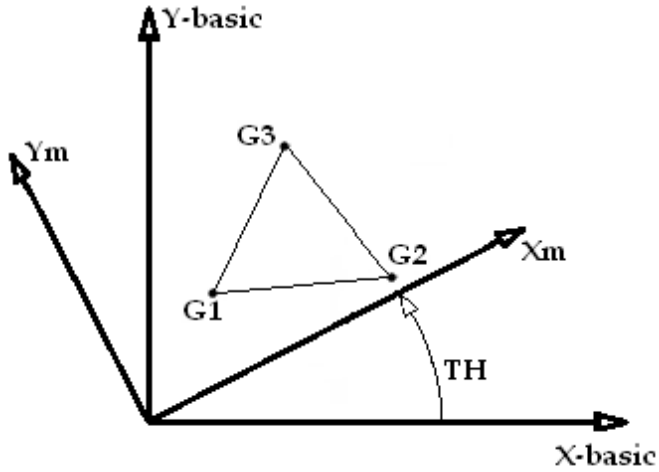


Figure 12-12. CPLSTN3 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then

the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTN4

Plane Strain Quadrilateral Element Connection

Defines a plane strain quadrilateral element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTN4	EID	PID	G1	G2	G3	G4	THETA		

EXAMPLE:

CPLSTN4	111	203	31	74	75	32	15.0		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE, PLPLANE, or PGPLSN entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 11. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

4. All interior angles must be less than 180º.

5. The element coordinate system is the basic coordinate system.

6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

10. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

    - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

    - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

11. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 13 and 14 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

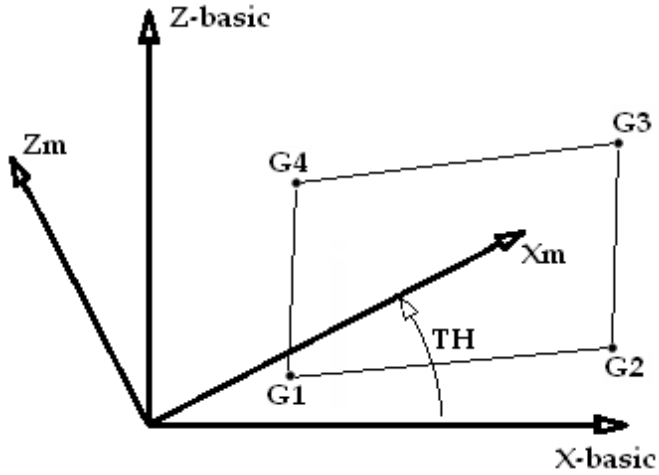


Figure 12-13. CPLSTN4 Element Geometry and Coordinate Systems, XZ-Plane

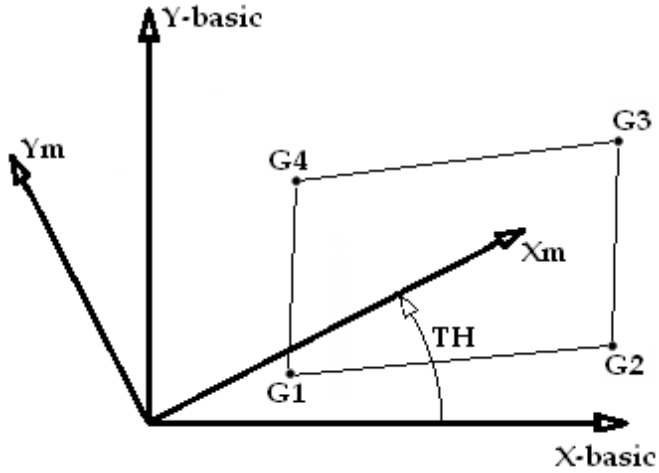


Figure 12-14. CPLSTN4 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. Incompatible modes are used for this element. Incompatible modes may be turned off by specifying ICMODE=0 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTN6

Plane Strain Triangular Element Connection

Defines a plane strain triangular element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTN6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA								

EXAMPLE:

CPLSTN6	302	3	31	33	71	32	51	52	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE, PLPLANE, or PGPLSN entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. G1, G2, G3 are the corner grid points and G2, G4, G6 are the midside grid points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 10. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. Grid points G1 through G6 must be numbered as shown in CTRIA6.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 15 and 16 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

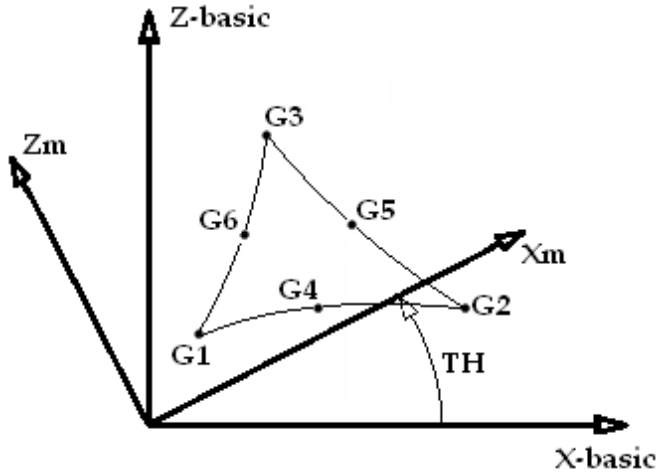


Figure 12-15. CPLSTN6 Element Geometry and Coordinate Systems, XZ-Plane

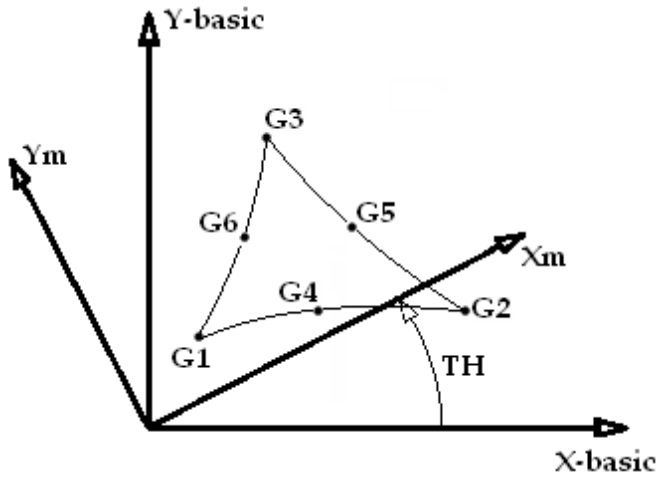


Figure 12-16. CPLSTN6 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 6-node triangular elements may be converted to 7-node triangular elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTN8

Plane Strain Quadrilateral Element Connection

Defines a plane strain quadrilateral element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTN8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	THETA						

EXAMPLE:

CPLSTN8	207	3	31	33	73	71	32	51	
	53	72	45.0						

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE, PLPLANE, or PGPLSN entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. G1 to G4 are the corner grid points and G5 to G8 are the midside grid points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 10. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. Grid points G1 through G8 must be numbered as shown in CQUAD8.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 17 and 18 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

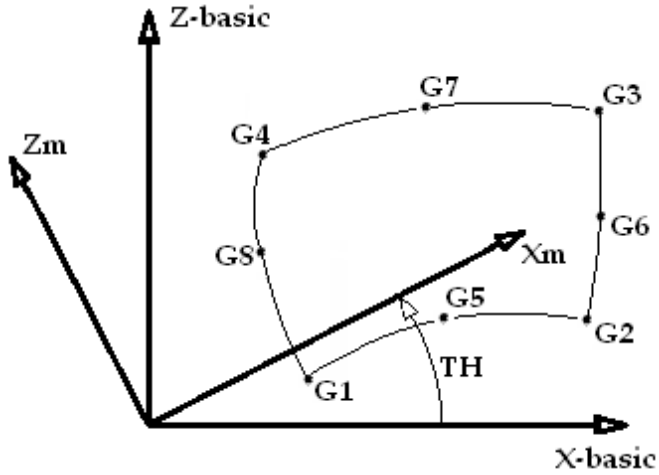


Figure 12-17. CPLSTN8 Element Geometry and Coordinate Systems, XZ-Plane

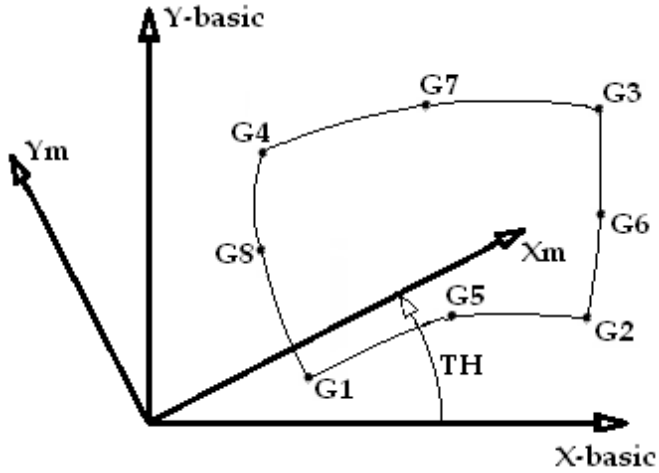


Figure 12-18. CPLSTN8 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. The 9-node plane strain element is more effective in the analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTS3

Plane Stress Triangular Element Connection

Defines a plane stress triangular element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTS3	EID	PID	G1	G2	G3		THETA		
				TFLAG	T1	T2	T3		

EXAMPLE:

CPLSTS3	111	203	31	74	75		30.0		
				1	1.2	1.2	1.2		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 10. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)

Field	Contents
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. T1, T2, and T3 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

- If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 19 and 20 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

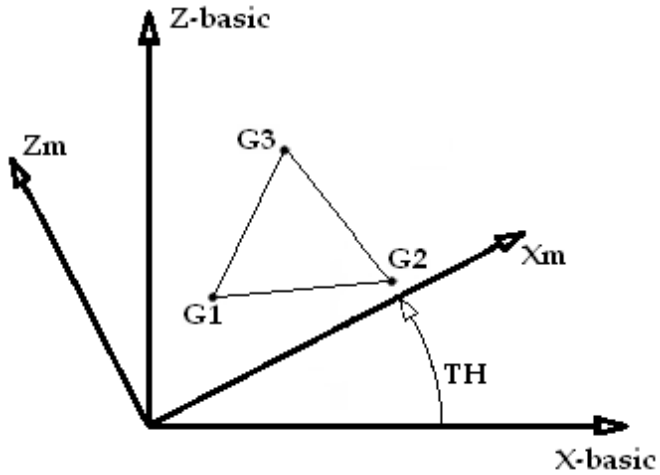


Figure 12-19. CPLSTS3 Element Geometry and Coordinate Systems, XZ-Plane

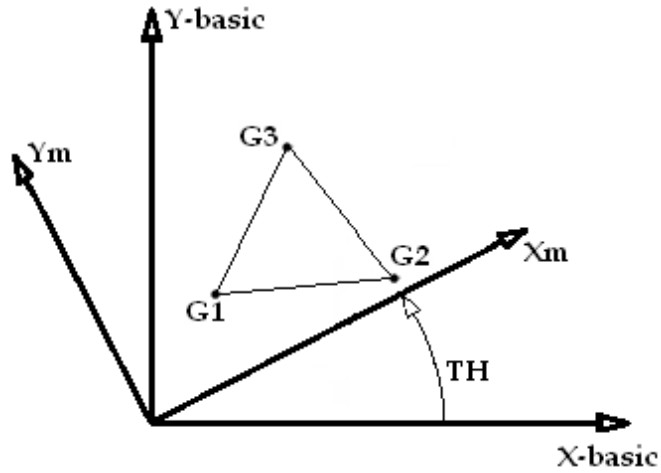


Figure 12-20. CPLSTS3 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTS4

Plane Stress Quadrilateral Element Connection

Defines a plane stress quadrilateral element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTS4	EID	PID	G1	G2	G3	G4	THETA		
				TFLAG	T1	T2	T3	T4	

EXAMPLE:

CPLSTS4	111	203	31	74	75	32	30.0		
					0.8	0.8	0.8	0.8	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 10. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)

Field	Contents
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. T1, T2, T3 and T4 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

- If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 21 and 22 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

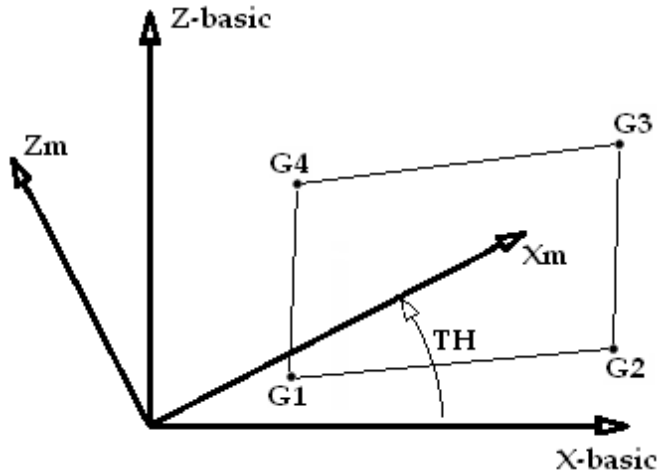


Figure 12-21. CPLSTS4 Element Geometry and Coordinate Systems, XZ-Plane

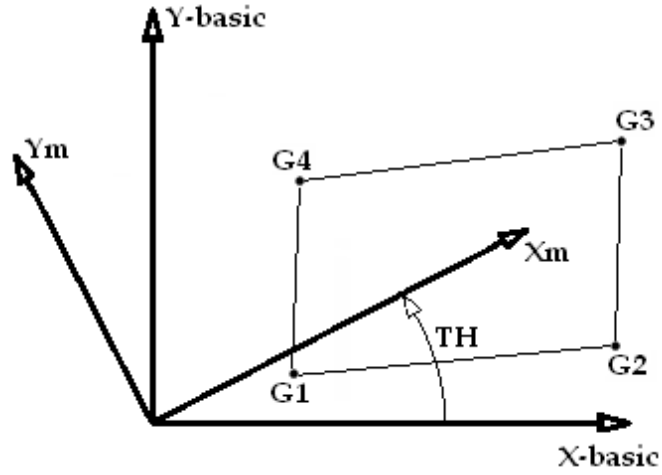


Figure 12-22. CPLSTS4 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. Incompatible modes are used for this element. Incompatible modes may be turned off by specifying ICMODE=0 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTS6

Plane Stress Triangular Element Connection

Defines a plane stress triangular element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTS6	EID	PID	G1	G2	G3	G4	G5	G6	
			THETA	TFLAG	T1	T2	T3		

EXAMPLE:

CPLSTS6	302	3	31	33	71	32	51	52	
			25.0	1	0.2	0.2	0.2		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 11. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)

Field	Contents
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. T1, T2, and T3 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. Grid points G1 through G6 must be numbered as shown in CTRIA6.

5. The element coordinate system is the basic coordinate system.

6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

10. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

    - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the

Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

- If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

11. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 23 and 24 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

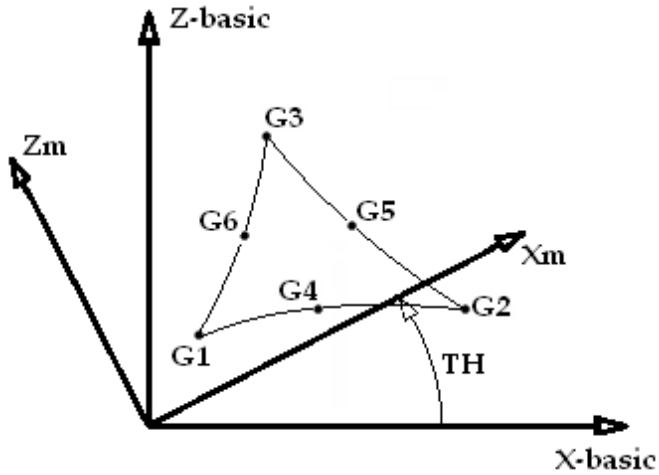


Figure 12-23. CPLSTS6 Element Geometry and Coordinate Systems, XZ-Plane

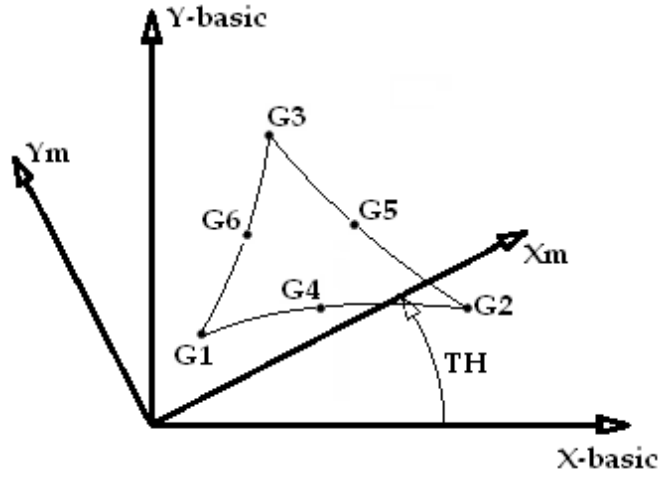


Figure 12-24. CPLSTS6 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 6-node triangular elements may be converted to 7-node triangular elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPLSTS8

Plane Stress Quadrilateral Element Connection

Defines a plane stress quadrilateral element for use in linear or nonlinear analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPLSTS8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	THETA	TFLAG	T1	T2	T3	T4	

EXAMPLE:

CPLSTS8	207	3	31	33	73	71	32	51	
	53	72	70.0	1	0.5	0.5	0.5	0.5	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. See Remark 11. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)

Field	Contents
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of all axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. Grid points G1 through G8 must be numbered as shown in CQUAD8.

4. T1, T2, T3 and T4 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

5. The element coordinate system is the basic coordinate system.

6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

10. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

    - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the

Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

- If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement coordinate system to be parallel with the Y-axis of the basic coordinate system.

11. For orthotropic materials, the THETA angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 25 and 26 show the positive sense of the THETA angle when the element is defined on the XZ-plane and XY-plane, respectively.

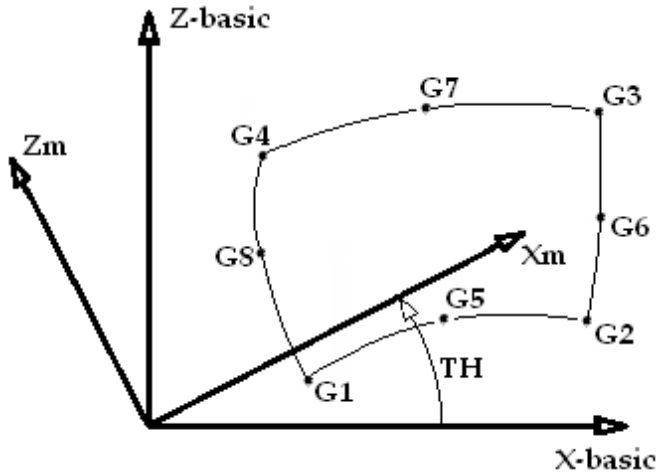


Figure 12-25. CPLSTS8 Element Geometry and Coordinate Systems, XZ-Plane

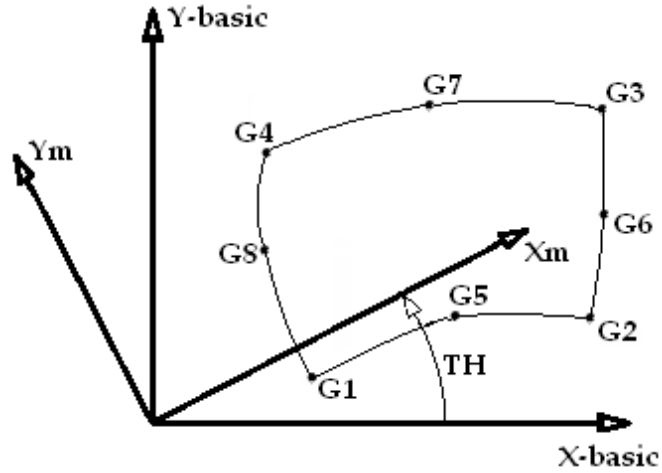


Figure 12-26. CPLSTS8 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CPYRAM

Five-Sided Solid Element Connection

Defines the connection of the five-sided solid element with five to thirteen grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CPYRAM	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

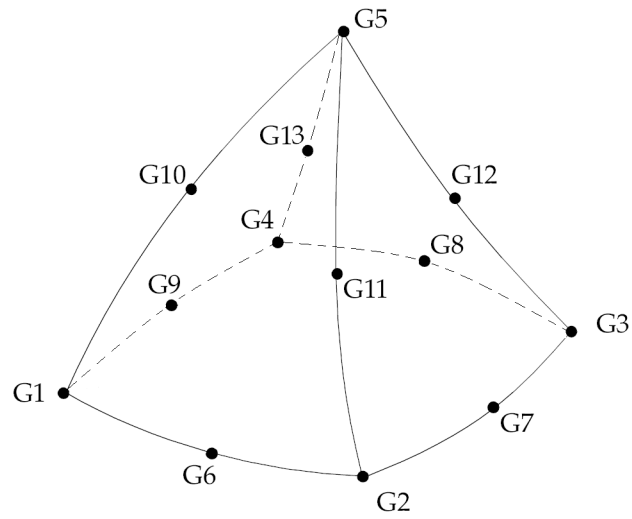


Figure 12-27. CPYRAM Element Connection

EXAMPLE:

CPYRAM	111	203	31	32	33	34	35	36	
	37	38	39	40	41	42	43		

CPYRAM

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID, PLSOLID (SOL 601 only), or PMIC entry. (Integer > 0; Default=EID)
Gi	Grid point identification numbers of connected points. (Unique integers > 0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. The other four faces are triangles. G5 is the vertex and must be opposite with the quadrilateral face.

3. The edge points G6 to G13 are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.

4. Components of stress are output in the material coordinate system.

5. The CPYRAM element coordinate system is the same as the basic coordinate system.

6. It is recommended that the edge grids be located within the middle third of the edge.

7. The CPYRAM element does not support hyperelasticity in SOL 106 and 129.

8. By default, all eight edges of the element are considered straight unless any of G6 through G13 is specified.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 701, only elements with 5 grid points are allowed.

2. For SOL 601, 13-node CPYRAM elements may be converted to 14-node CPYRAM elements (1 additional node on the centroid of the quadrilateral face of the element) by specifying ELCV=1 in the NXSTRAT entry. 14-node CPYRAM elements are more effective than 13-node CPYRAM for analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

CQUAD

Fully Nonlinear Plane Strain Element

Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9						

EXAMPLE:

CQUAD	111	203	31	74	75	32			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PLPLANE entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer ≥ 0 or blank)
G9	Identification number of center grid point. Optional. (Integer ≥ 0 or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G9 must be numbered as shown and must lie on a plane.
3. It is recommended that the edge points be located within the middle third of the edge.
4. Plot codes are specified under the CQUADFD element name in “Item Codes” .
5. Stresses and strains are output in the coordinate system identified by the CID field of the PLPLANE entry.

REMARKS RELATED TO SOL 601:

1. Only elements with 4, 8, or 9 grid points are allowed.
2. Stresses and strains are output in the basic coordinate system, i.e., CID field of the PLPLANE entry is assumed to be 0.
3. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. The 9-node element is more effective in the analysis of incompressible rubber-like materials.

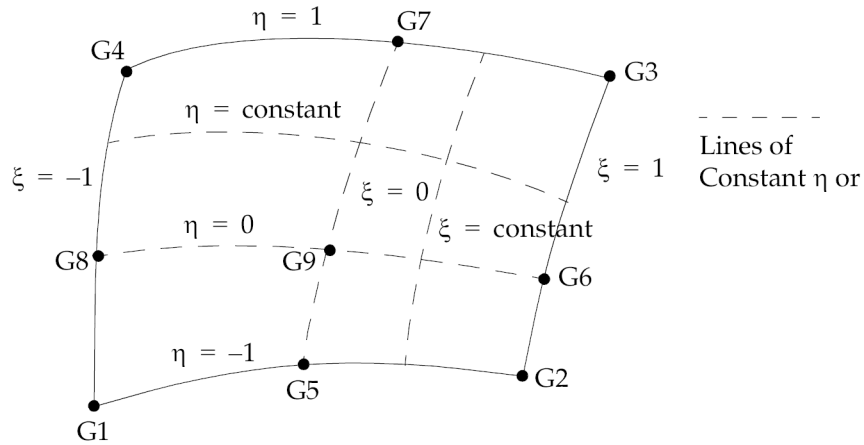


Figure 12-28. CQUAD Element Coordinate System

CQUAD4

Quadrilateral Plate Element Connection

Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

EXAMPLE:

CQUAD4	111	203	31	74	75	32	2.6	0.3	
		1	1.77	2.04	2.09	1.80			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCOMP, PCOMPG, PLPLANE, PMIC, or PSHELL entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique.)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. See Figure 12-30. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. MCID is ignored for hyperelastic elements. See Remark 12. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)

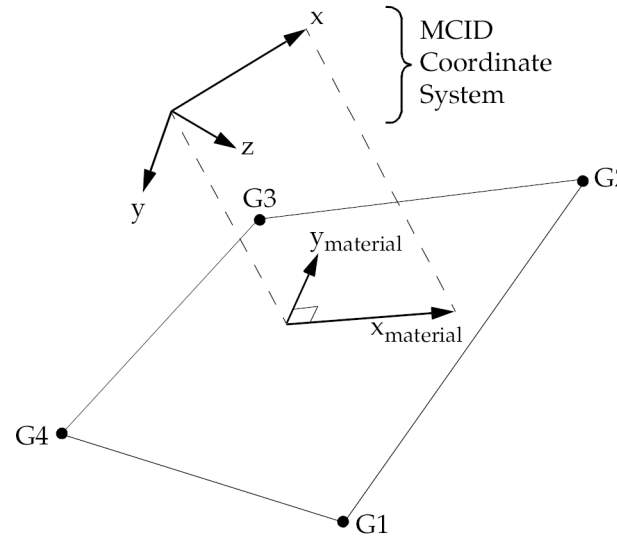


Figure 12-29. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. ZOFFS is ignored for hyperelastic elements. See Remarks 8 and 9. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)
Ti	Thickness of element at grid points G1 through G4 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. Ti is ignored for hyperelastic elements. See Remark 6. (Real > 0.0 or blank. See Remark 4 for the default.)

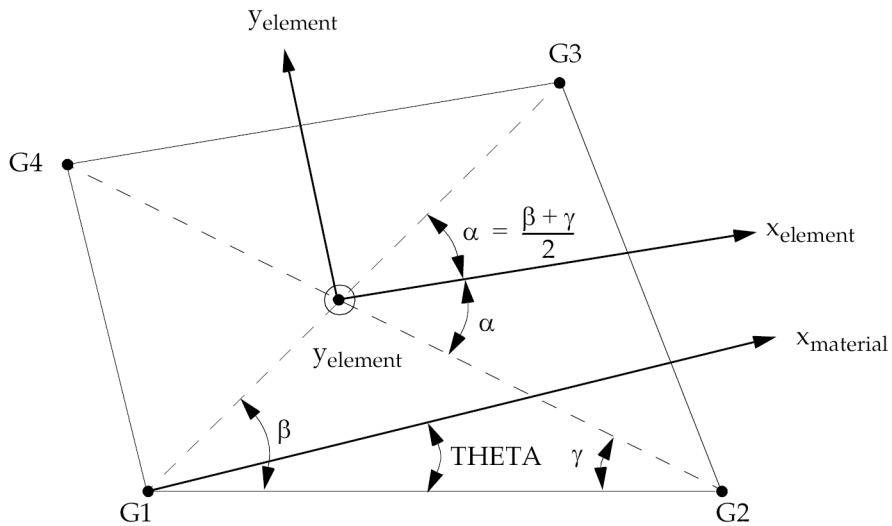


Figure 12-30. CQUAD4 Element Geometry and Coordinate Systems

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

3. All interior angles must be less than 180°.

4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

5. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

6. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

7. The reference coordinate system for the output of stress, strain and element force depends on the element type.

- For CQUAD4 elements which are not hyperelastic, the reference coordinate system for output is the element coordinate system. For SOL 106 geometric nonlinear analysis, the nonlinear stresses are output in deformed element coordinate system.

- For hyperelastic elements, the stress and strain are output according to CID on the PLPLANE entry.

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CQUAD4 entry.

   ZOFFS should not be used when differential stiffness is calculated in solutions 105 and 200, and solutions 103, 107 - 112 using STATSUB, since the differential stiffness calculation does not include the offset vectors.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

9. ZOFFS has the following caveats in nonlinear solutions 106 and 129:

   - For geometry nonlinear only, the differential stiffness calculation does not include the offset vectors. The exclusion of the offset in the differential stiffness will have an adverse effect on convergence, however, if the solution converges, the results will be correct. In a nonlinear solution, equilibrium (and convergence) is achieved when internal and external forces balance. The offset is correctly accounted for in the internal and external force calculations even though it is not accounted for in the differential stiffness.

   - ZOFFS should not be defined on elements which use MATS1 nonlinear material definitions.

   - ZOFFS is ignored for hyperelastic elements.

   - ZOFFS should not be used in nonlinear solutions 106 and 129 in combination with thermal loads.

10. For finite deformation hyperelastic analysis, the plot codes are given by the CQUADFD element name in “Item Codes”.

11. By default, all four edges of the element are considered straight.

12. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element

face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

13. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

REMARKS RELATED TO SOLS 601 AND 701:

1. ZOFFS is ignored.

2. Incompatible modes are used for this element if it is used as a shell element. Incompatible modes may be turned off by specifying ICMODE=0 in the NXSTRAT entry. For modeling shell elements, CQUAD4 is most effective.

3. Incompatible modes are not used for this element if it used as a plane strain element (SOL 601 only). The use of CQUAD or CQUAD8 to model 9-node or 8-node plane strain elements is recommended.

4. For both linear and nonlinear analysis, stress and strain results at grid points may be requested for single-ply elements by specifying STRESS(CORNER) case control command.

5. For geometry nonlinear analysis, the nonlinear stresses are written in undeformed element coordinate system.

CQUAD8

Curved Quadrilateral Shell Element Connection

Defines a curved quadrilateral shell or plane strain element with eight grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID	ZOFFS	
	TFLAG								

EXAMPLE:

CQUAD8	207	3	31	33	73	71	32	51	
	53	72	0.125	0.025	0.030	0.025	30.	0.03	

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG, or PLPLANE entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer ≥ 0 or blank)

Field	Contents
Ti	Thickness of element at grid points G1 through G4 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. Ti is ignored for hyperelastic elements. See Remark 6. (Real > 0.0 or blank. See Remark 4 for the default.)
THETA	Material property orientation angle in degrees. See Figure 12-32. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. MCID is ignored for hyperelastic elements. See Remarks 3 and 12). (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)

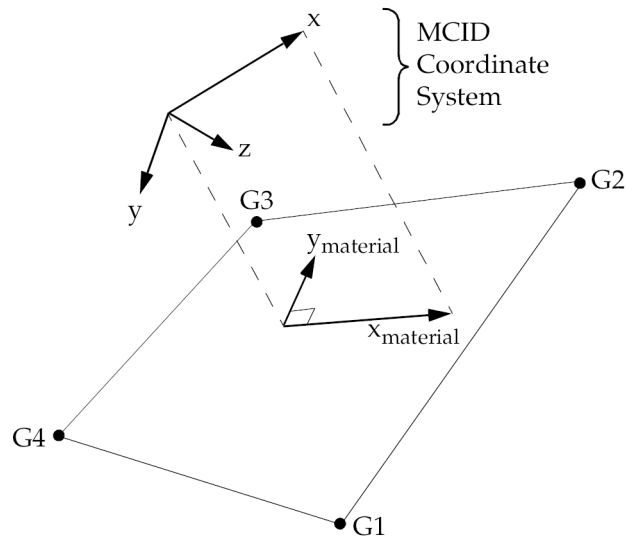


Figure 12-31. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 8. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)

REMARKS:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.

2. Grid points G1 to G8 must be numbered as shown in Figure 12-32.

3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between $x_{material}$ and the line of constant xi. THETA is calculated by projecting the x-axis of the MCID coordinate system onto the surface of the element. Because the shape of a CQUAD8 can vary from one integration point to the next, the direction of xi can also vary, causing the resulting material coordinate definition to vary. When using orthotropic or anisotropic materials which depend on consistent material coordinate directions, it is recommended to use the QUAD4 element.

4. T1, T2, T3 and T4 are optional. If they are not supplied, they will be set equal to the value of T on the PSHELL entry.

5. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

6. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

7. It is recommended that the mid-side grid points be located within the middle third of the edge. If the edge point is located at the quarter point, the program may fail with a divide-by-zero error, or the calculated stresses will be meaningless.

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CQUAD8 entry.

   ZOFFS should not be used when differential stiffness is calculated in solutions 105 and 200, and solutions 103, 107 - 112 using STATSUB, since the differential stiffness calculation does not include the offset vectors.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

9. If all mid-side grid points are deleted, then the element will be excessively stiff and the transverse shear forces incorrect. A User Warning Message is printed, and a CQUAD4 element is recommended instead. If the element is hyperelastic, then it is processed identically to the hyperelastic CQUAD4 element.

10. For a description of the element coordinate system, see “Plate and Shell Elements” in the *NX Nastran Element Library*. Stresses and strains are output in the local coordinate system identified by x_l and y_l in Figure 12-32. However, for hyperelastic elements, the stress and strain are output in the coordinate system identified by the CID field on the PLPLANE entry.

11. For hyperelastic elements the plot codes are specified under the CQUADFD element name in “Item Codes”.

12. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

13. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

REMARKS RELATED TO SOL 601:

1. Only elements with 4 or 8 grid points are allowed. That is, either G5 to G8 are all specified or none are specified.

2. ZOFFS is ignored.

3. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. As a plane strain element, the 9-node element is more effective in the analysis of incompressible media and inelastic materials including rubber-like materials, elasto-plastic materials, and materials with Poisson’s ratio close to 0.5. As a shell element, the 9-node element is also more effective than the 8-node element. For example, the 9-node element support of large strain.

4. In a nonlinear analysis, both linear and nonlinear stress/strain data blocks are output for CQUAD8 elements in the op2 file. Note that in a future release, the output of linear stress/strain data blocks for CQUAD8 elements may be discontinued for a nonlinear analysis.

5. For geometry nonlinear analysis, the nonlinear stresses are written in undeformed element coordinate system.

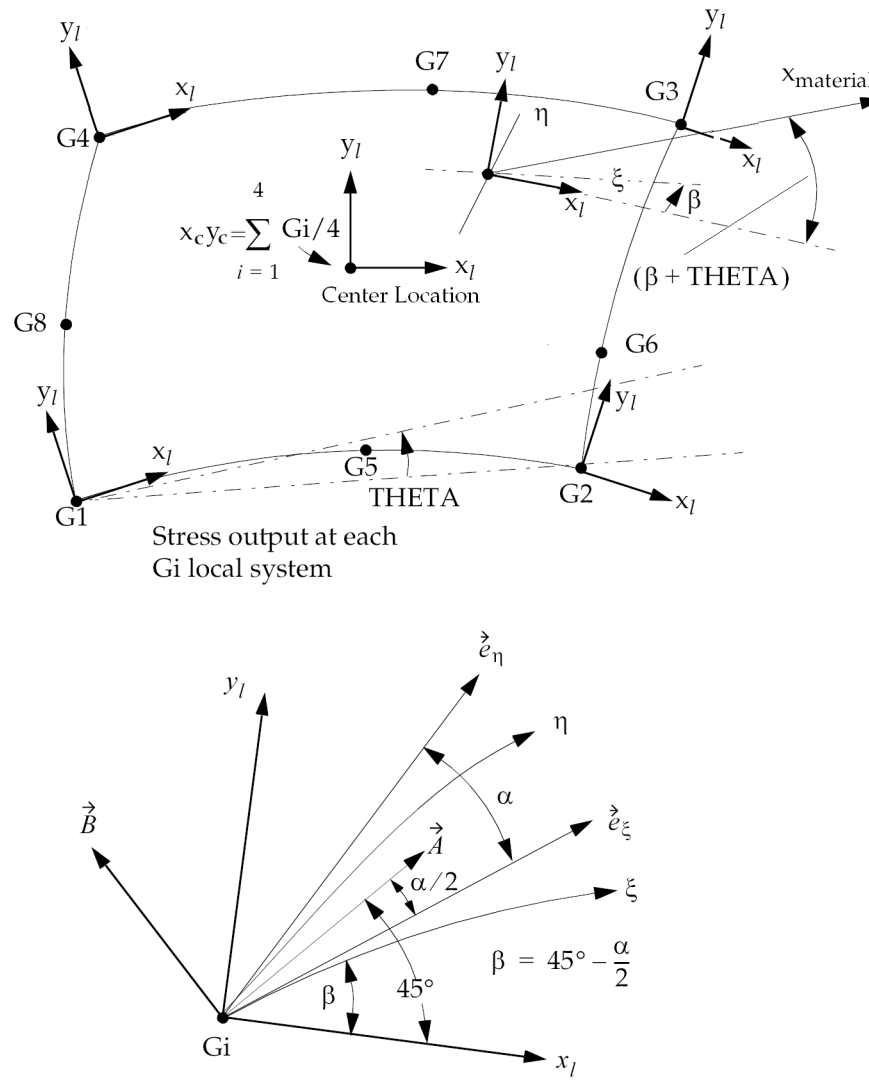


Figure 12-32. CQUAD8 Element Geometry and Coordinate Systems

where:

$\vec{e}_\eta$ is tangent to η at Gi.

$\vec{e}_\xi$ is tangent to ξ at Gi.

$\vec{A}$ is formed by bisection of $\vec{e}_\eta$ and $\vec{e}_\xi$.

$\vec{B}$ and $\vec{A}$ are perpendicular.

y_l is formed by bisection of $\vec{A}$ and $\vec{B}$.

x_l is perpendicular to y_l.

CQUADR

Quadrilateral Plate Element Connection

Defines an isoparametric membrane and bending quadrilateral plate element. This element is less sensitive to initial distortion and extreme values of Poisson's ratio than the CQUAD4 element. It is a companion to the CTRIAR element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

EXAMPLE:

CQUADR	82	203	31	74	75	32	2.6		
			1.77	2.04	2.09	1.80			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP, or PCOMPG entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. See Figure 12-34. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. See Remark 13. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)

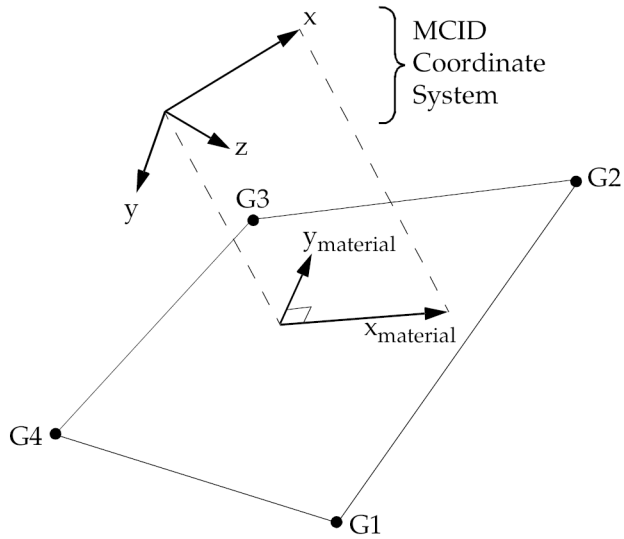


Figure 12-33. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 8. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)
Ti	Thickness of element at grid points G1 through G4 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. See Remark 6. (Real > 0.0 or blank. See Remark 4 for the default.)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

3. All the interior angles must be less than 180°.

4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

5. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

6. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

7. Stresses and strains are output in the element coordinate system at the centroid unless the CORNER or CUBIC entry is specified on the STRESS case control card. Specifying CORNER or CUBIC will result in stresses at grid points G1 through G4.

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CQUADR entry.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

9. The rotational degrees-of-freedom normal to the element are active in the element formulation and must not be constrained unless at a boundary. Inaccurate results will be obtained if they are constrained.

10. As compared to the CQUAD4 element, the CQUADR element:

    - Accounts for normal rotational DOF to provide improved membrane accuracy.

    - Is less sensitive to high aspect ratios and values of Poisson’s ratio near 0.5.

11. The CQUADR element is not supported in SOL 401. The CQUADR element is supported in all other linear structural and heat transfer solutions. If used in SOL 106 or 129, the CQUADR element behaves linear elastically in a material nonlinear analysis and the stiffness does not reformulate in a geometric nonlinear analysis.

12. The CTRIAR element is the triangular companion to the CQUADR element and should be used instead of CTRIA3 or CTRIA6.

13. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

14. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

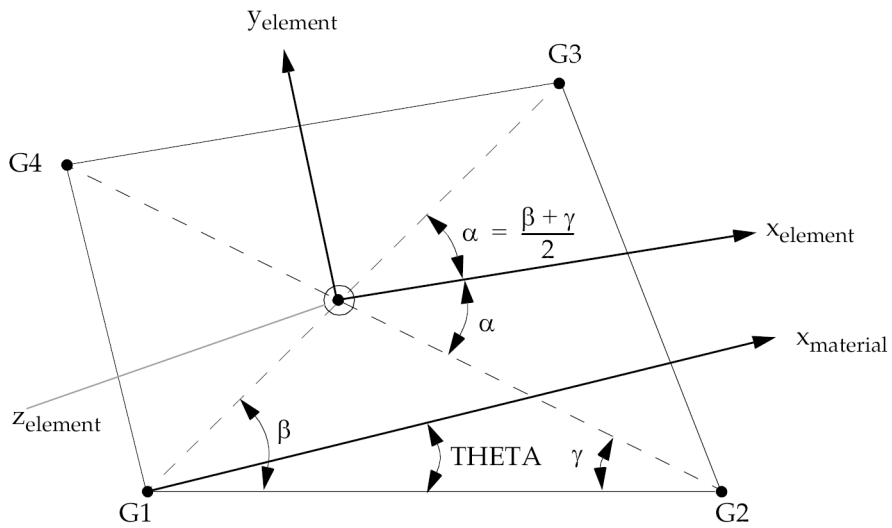


Figure 12-34. CQUADR Element Geometry and Coordinate Systems

REMARKS RELATED TO SOL 601:

1. ZOFFS is ignored.

2. CQUADR can only be used in linear analysis. It is recommended that CQUAD4 be used instead of CQUADR in SOL 601 analysis. If the CQUADR element is used in SOL 601, the CQUADR element behaves linear elastically in a material nonlinear analysis and the stiffness does not reformulate in a geometric nonlinear analysis. The CQUADR element is not supported in SOL 701.

3. For geometric nonlinear analysis, the nonlinear stresses are written in the undeformed element coordinate system.

CQUADX4

Axisymmetric Quadrilateral Element Connection

Defines an isoparametric and axisymmetric quadrilateral cross-section ring element for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUADX4	EID	PID	G1	G2	G3	G4	TH		

EXAMPLE:

CQUADX4	203	100	10	20	30	40	30.0		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PMIC, PSOLID or PLSOLID entry. (Integer > 0; Default=EID)
Gi	Grid point identification numbers of connected points. (Unique integers > 0)
TH	Material property orientation angle in degrees. See Remark 4. TH is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of the axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), the plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and the plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. When axisymmetric elements are defined on the XZ-plane, X is the radial direction, and Z is the axial direction. The grid points defining these elements must have X ≥ 0. See Figure 36.

   When axisymmetric elements are defined on the XY-plane, Y is the radial direction, and X is the axial direction. The grid points defining these elements must have Y ≥ 0. See Figure 37.

4. For orthotropic materials, the TH angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 36 and 37 show the positive sense of the TH angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. The initial element coordinate system is the basic coordinate system.

6. See the remarks on the PSOLID entry for information on stress and strain output coordinate systems.

7. The CQUADX4, CQUADX8, CTRAX3, and CTRAX6 elements behave linearly in SOL 106 and 129 when not hyperelastic.

8. For hyperelastic element, the plot codes are specified under the CQUADX4FD element name in Item Codes Overview.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement

coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. A concentrated load (e.g. FORCE entry) at Gi is the total of the force around the circumference. Reaction force and applied load output are the same. For example, to apply a distributed load of 135.0 Newton/mm on a single grid where the radius is 0.5 mm:

    The value entered on a FORCE entry = (Distributed force * 2 * π * Radius)

    = 135.0 N/mm * 2 * π * 0.5 mm

    = 424.1 N

    See system cell 587 for information on the pre-version 10 axisymmetric element behavior.

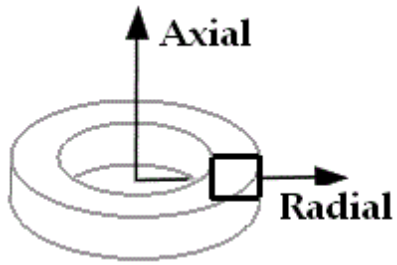


Figure 12-35. CQUADX4 Element Idealization

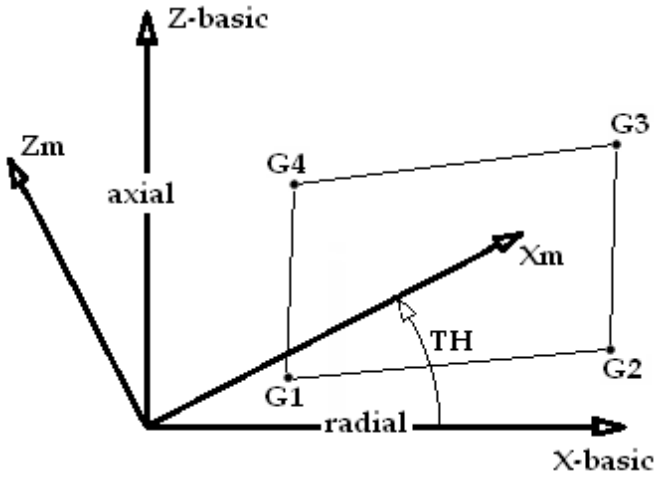


Figure 12-36. CQUADX4 Element Geometry and Coordinate Systems, XZ-Plane

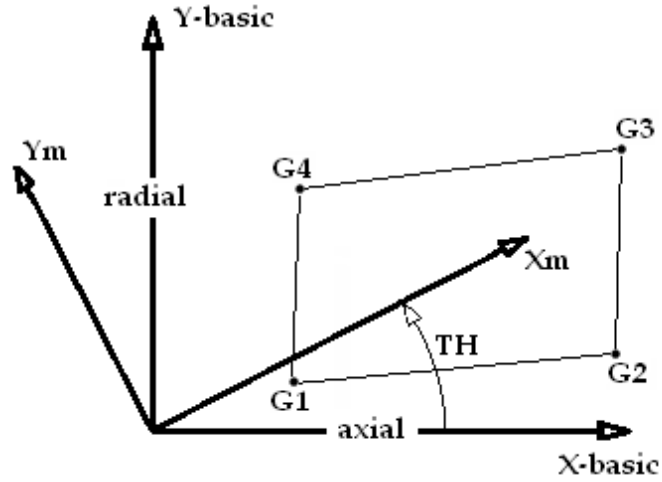


Figure 12-37. CQUADX4 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. When TH is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CQUADX8

Axisymmetric Quadrilateral Element Connection

Defines an isoparametric and axisymmetric quadrilateral cross-section ring element with midside nodes for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CQUADX8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	TH						

EXAMPLE:

CQUADX8	203	100	10	20	30	40	50	60	
	70	80	30.0						

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID or PLSOLID entry. (Integer > 0; Default=EID)
Gi	Grid point identification numbers of connected points (midside grids cannot be eliminated). (Unique integers > 0)
TH	Material property orientation angle in degrees. See Remark 4. TH is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of the axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), the plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and the plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. When axisymmetric elements are defined on the XZ-plane, X is the radial direction, and Z is the axial direction. The grid points defining these elements must have X ≥ 0. See Figure 39.

   When axisymmetric elements are defined on the XY-plane, Y is the radial direction, and X is the axial direction. The grid points defining these elements must have Y ≥ 0. See Figure 40.

4. For orthotropic materials, the TH angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 39 and 40 show the positive sense of the TH angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. The initial element coordinate system is the basic coordinate system.

6. See the remarks on the PSOLID entry for information on stress and strain output coordinate systems.

7. The CQUADX4, CQUADX8, CTRAX3, and CTRAX6 elements behave linearly in SOL 106 and 129 when not hyperelastic.

8. For hyperelastic element, the plot codes are specified under the CQUADX8FD element name in Item Codes Overview.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement

coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. A concentrated load (e.g. FORCE entry) at Gi is the total of the force around the circumference. Reaction force and applied load output are the same. For example, to apply a distributed load of 135.0 Newton/mm on a single grid where the radius is 0.5 mm:

    The value entered on a FORCE entry = (Distributed force * 2 * π * Radius)

    = 135.0 N/mm * 2 * π * 0.5 mm

    = 424.1 N

    See system cell 587 for information on the pre-version 10 axisymmetric element behavior.

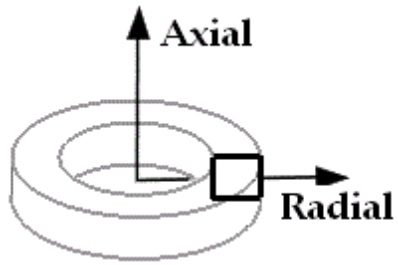


Figure 12-38. CQUADX8 Element Idealization

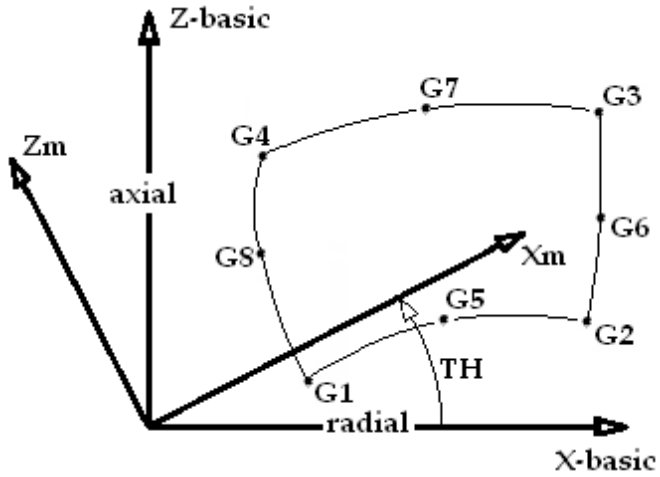


Figure 12-39. CQUADX8 Element Geometry and Coordinate Systems, XZ-Plane

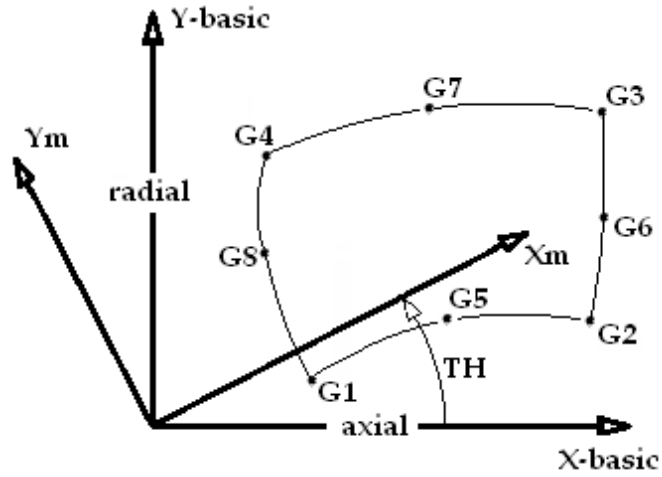


Figure 12-40. CQUADX8 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. When TH is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CRAC2D

Two-Dimensional Crack Tip Element

Defines a two-dimensional crack tip element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CRAC2D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

EXAMPLE:

CRAC2D	114	108	2	5	6	8	7	11	
	12	14	16	17		20	22		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PRAC2D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0, G11 through G18 may be blank.)

REMARKS:

1. The following is a dummy element and requires the presence of one Bulk Data entry of the form:

ADUM8	18	0	5	0	CRAC2D				

2. The element should be planar. Significant deviations will produce fatal errors.

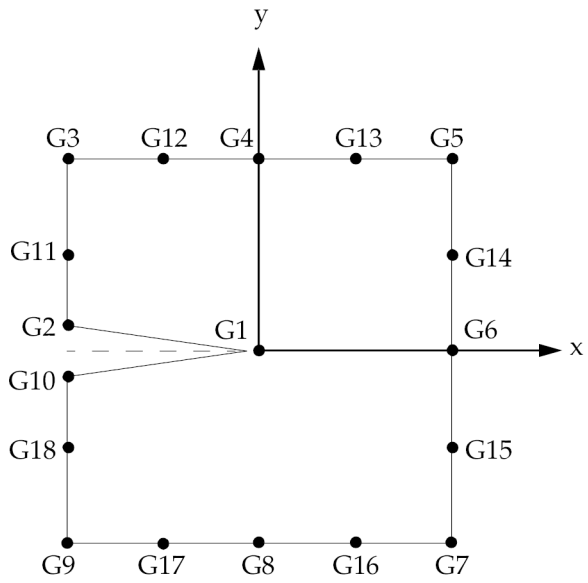
3. Grid points G1 through G10 are required while grid points G11 through G18 are optional for the quadrilateral form of the element.

4. The stresses and stress intensity factors are calculated assuming that G2 and G10 are coincident. Deviations from this will produce erroneous results.

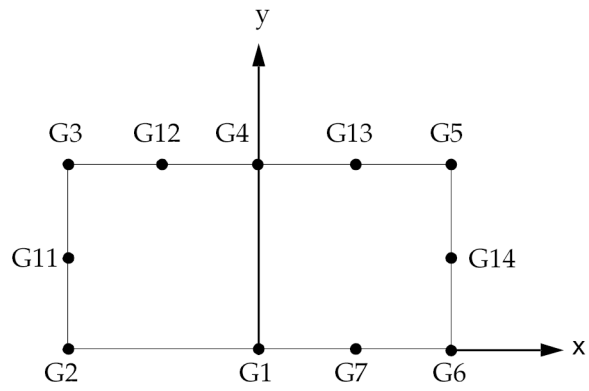
5. For the symmetric half-crack option, grid points G1 through G7 are required while grid points G11 through G14 are optional. Grid points G8 through G10 and G15 through G18 must not be present for this option.

6. The ordering conventions for the full-crack and half-crack options are shown in Figure 12-41.

7. The stress output is interpreted as described in “Two-Dimensional Crack Tip Element (CRAC2D)” in the *NX Nastran Element Library*.



(a) Full Crack Option



(b) Symmetric Half-Crack Option

Figure 12-41. CRAC2D Element Connection for Full and Symmetric Options

CRAC3D

Three-Dimensional Crack Tip Element

Defines a three-dimensional crack tip element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CRAC3D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20	G21	G22	
	G23	G24	G25	G26	G27	G28	G29	G30	
	G31	G32	G33	G34	G35	G36	G37	G38	
	G39	G40	G41	G42	G43	G44	G45	G46	
	G47	G48	G49	G50	G51	G52	G53	G54	
	G55	G56	G57	G58	G59	G60	G61	G62	
	G63	G64							

EXAMPLE:

CRAC3D	113	101	2	5	7	8	4	10	
	11	14	15	17		3	6	9	
	12		16		102	105	107	108	
	104	110	111	114	115	117		103	
	106	109	112		116		202	205	
	207	208	204	210	211	214	215	217	
	225	226							

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PRAC3D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0)

REMARKS:

1. The following is a dummy element and requires the presence of one Bulk Data entry of the form:

ADUM9	64	0	6	0	CRAC3D				

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. This element, including grid point numbering conventions, is shown in Figure 12-42 and Figure 12-43. grid points G1 through G10, and G19 through G28 are required; midside and surface grid points G11 through G18, G29 through G36, and G37 through G64 are optional. Either all or none of grid points G37 through G46 should be present. A fatal error message will be issued for partial connectivity.

4. For the symmetric half-crack option grid points G1 through G7, and G19 through G25 are required, whereas grid points G11 through G14, G29 through G32, and G37 through G42 are optional. Grid points G8 through G10, G15 through G18, G26 through G28, G33 through G36, G43 through G46, G51 through G55, and G60 through G64 should not be specified to invoke this option.

5. It is recommended that both the faces (formed by grid points G1 through G18 and grid points G19 through G36) and the midplane (formed by grid points G37 through G46) be planar. It is also recommended that midside grid points G37 through G46 be located within the middle third of the edges.

6. The midside nodes on both the faces should be defined in pairs. For example, if grid point G11 is not defined, then grid point G29 should not be defined and vice versa.

7. The stresses and stress intensity factors are calculated with the assumptions that grid points G2 and G10, G20 and G28, and G38 and G46 are coincident. Deviation from this condition will produce erroneous results.

8. The stress output is interpreted as described in “Three-Dimensional Crack Tip Element (CRAC3D)” in the *NX Nastran Element Library*.

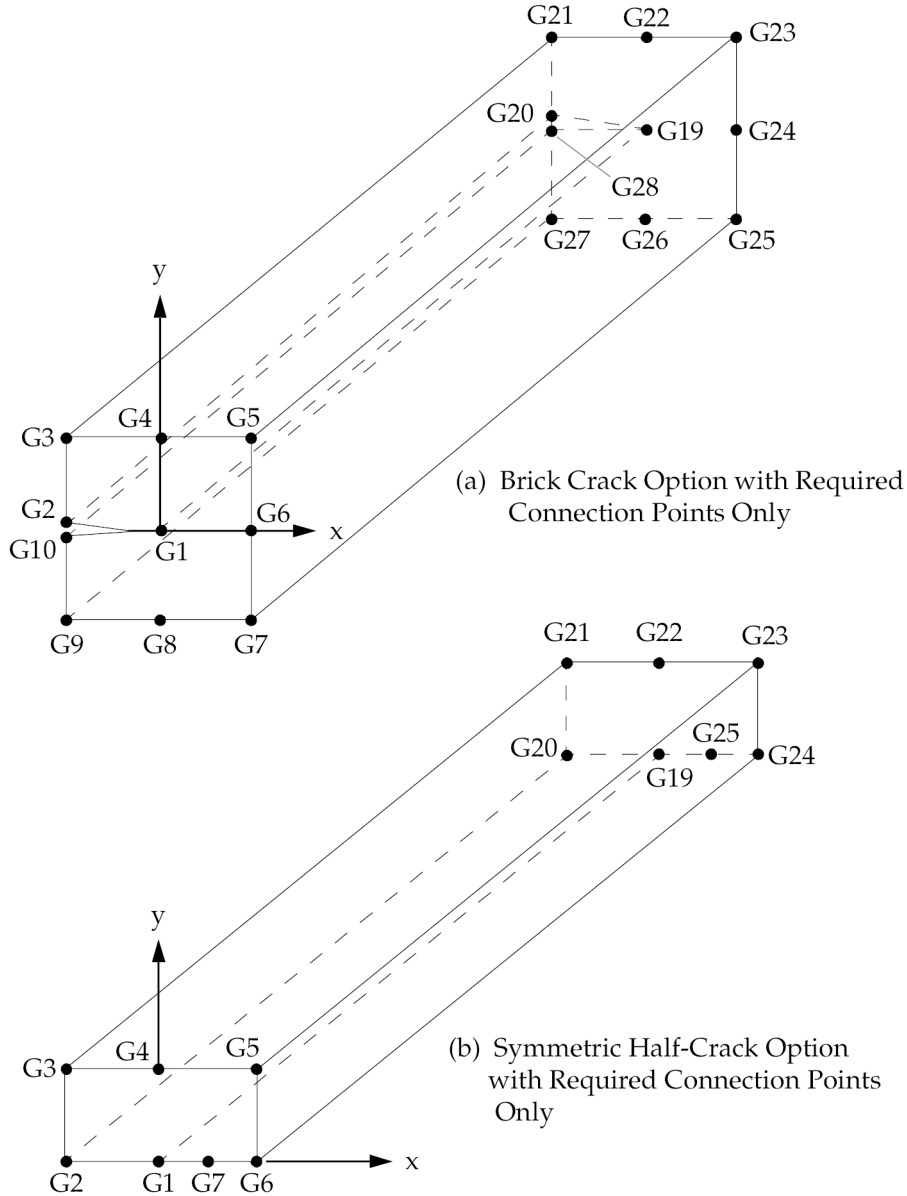


Figure 12-42. CRAC3D Solid Crack Tip Element with Required Connection Points Only

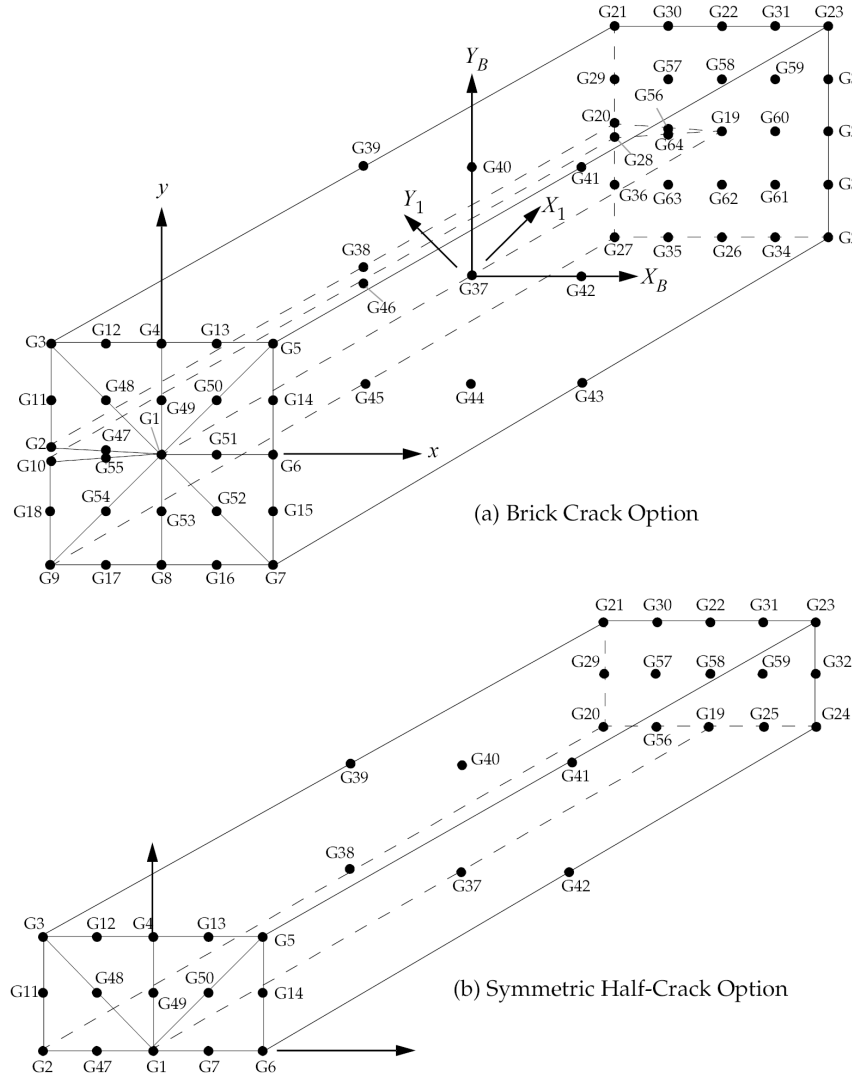


Figure 12-43. CRAC2D Solid Crack Tip Element with All Connection Points

CRAKTP

Crack Tip Specification

Specifies information related to a crack tip.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CRAKTP	SID	NR							
	G1	VCEV1	G3	VCEV2	G3	VCEV3	G4	VCEV4	
	G5	VCEV5	etc.						

EXAMPLE:

CRAKTP	1	3							
	11	1	12	2	13	3	16	4	
	17	9	52	8	57	9			

FIELDS:

Field	Contents
SID	Crack tip identification number. (Integer > 0)
NR	Number of rings to compute j-integral. (Integer > 0)
Gi	Grid point identification number for the crack tip. (Integer > 0)
VCEVi	Virtual crack extension vector identification number. (Integer > 0)

CREEP

Creep Characteristics

Defines creep characteristics based on experimental data or known empirical creep law. This entry will be activated if a MAT1, MAT2, MAT8, MAT9, or MAT11 entry with the same MID is used and the NLPARM entry is prepared for creep analysis (except for SOL 601). The creep formulation is principally suited for isotropic materials and, in general, when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CREEP	MID	T0	EXP	FORM	TIDKP	TIDCP	TIDCS	THRESH	
	TYPE	a	b	c	d	e	f	g	

EXAMPLE:

CREEP	8	1100.		CRLAW					
	121	6.985-6	2.444	7.032-4	0.1072	6.73-9	0.1479	3.0	

FIELDS:

Field	Contents
MID	Material identification number of a MAT1, MAT2, MAT8, MAT9, or MAT11 entry. (Integer > 0)
T0	Reference temperature at which creep characteristics are defined. See Remark 2. (Real; Default = 0.0)
EXP	Temperature-dependent term, $e^{(-\Delta H/(R \cdot T0))}$, in the creep rate expression. See Remark 2. (0.0 < Real ≤ 1.0; Default = 1.0E-9)
FORM	Form of the input data defining creep characteristics. (Character: “CRLAW” for empirical creep law, or “TABLE” for tabular input data of creep model parameters.)

Field	Contents
TIDKP, TIDCP, TIDCS	Identification number of a TABLES1 entry, which defines the creep model parameters $K_p(\sigma)$, $C_p(\sigma)$, and $C_s(\sigma)$, respectively. See Remarks 3 through 5. (Integer > 0)
THRESH	Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young's modulus. (0.0 < Real < 1.0E-3; Default = 1.0E-5)
TYPE	Identification number of the empirical creep law type. See Remark 1. (Integer: 111, 112, 121, 122, 211, 212, 221, 222, or 300)
a through g	Coefficients of the empirical creep law specified in TYPE. Continuation should not be specified if FORM = "TABLE". See Remark 1. (Real)

REMARKS:

1. Two classes of empirical creep law are available. **Creep Law Class 1** is expressed as:

$$\varepsilon^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t$$

Equation 12-1.

Parameters $A(\sigma)$, $R(\sigma)$, and $K(\sigma)$ are specified in the following form, as recommended by the Oak Ridge National Laboratory:

Parameter	Function 1	Digit	Function 2	Digit
$A(\sigma)$	$a\sigma^b$	i=1	$ae^{b\sigma}$	i=2
$R(\sigma)$	$ce^{d\sigma}$	j=1	$c\sigma^d$	j=2
$K(\sigma)$	$e \cdot [\sinh(f\sigma)]^g$	k=1	$ee^{f\sigma}$	k=2

TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines $A(\sigma) = a\sigma^b$, $R(\sigma) = c\sigma^d$, and $K(\sigma) = ee^{f\sigma}$.

Creep Law Class 2 (TYPE=300) is expressed as:

$$\varepsilon^{c}(\sigma, t) = a\sigma^{b} t^{d}$$

Equation 12-2.

where the values of b and d must be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

2. Creep law coefficients a through g are usually determined by least squares fit of experimental data, obtained under a constant temperature. This reference temperature at which creep behavior is characterized must be specified in the T0 field if the temperature of the structure is different from this reference temperature. The conversion of the temperature input (°F or °C) to °K (degrees Kelvin) must be specified in the PARAM,TABS entry as follows:

   PARAM,TABS,273.16 (If Celsius is used.)

   PARAM,TABS,459.69 (If Fahrenheit is used.)

   When the correction for the temperature effect is required, the temperature distribution must be defined in the bulk entries TEMP, TEMPP1, and/or TEMPRB, which are selected by the Case Control command TEMP(LOAD) = SID within the subcase.

   From the thermodynamic considerations, the creep rate is expressed as:

$$\dot{\varepsilon}^{C} = \dot{\varepsilon}_{A}(e^{-\Delta H / RT})$$

Equation 12-3.

where:

ΔH = energy of activation

R = gas constant (= 1.98 cal/mole °K)

T = absolute temperature (°K)

$\dot{\varepsilon}_A$ = strain/sec per activation

If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by the following factor:

$$\frac{\dot{\varepsilon}^c}{\dot{\varepsilon}_0^c} = \mathrm{EXP}^{\left(\frac{T0}{T} - 1\right)}$$

Equation 12-4.

where:

$\dot{\varepsilon}^c$ = corrected creep rate

$\dot{\varepsilon}_0^c$ = creep rate at T0

$\mathrm{EXP}^{\left(\frac{T0}{T} - 1\right)}$ = correction factor

3. If the creep model parameters K_p, C_p, and C_s are to be specified with FORM = “TABLE”, then TABLES1 entries (with IDs that appear in TIDxx fields) must be provided in the bulk section. In this case, the continuation should not be specified.

4. Creep model parameters K_p, C_p, and C_s represent parameters of the uniaxial rheological model as shown in Figure 12-44.

   Tabular values (X_i, Y_i) in the TABLES1 entry correspond to (σ_i, K_{pi}), (σ_i, C_{pi}), and (σ_i, C_{si}) for the input of K_p, C_p, and C_s, respectively. For linear viscoelastic materials, parameters K_p, C_p, and C_s are constant and two values of σi must be specified for the same value of K_{pi}, C_{pi}, and C_{si}.

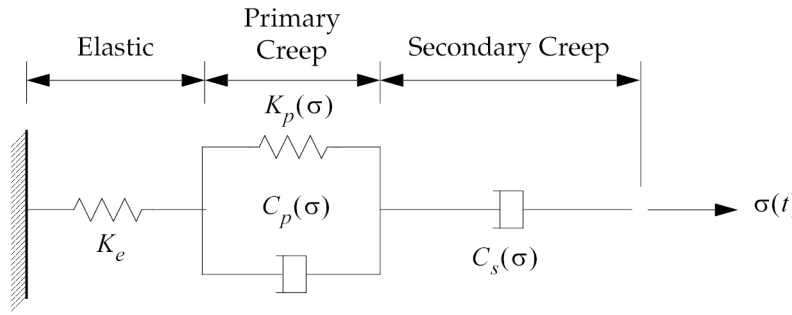


Figure 12-44. CREEP Parameter Idealization

Creep model parameters, as shown in Figure 12-45 through Figure 12-47 below, must have positive values. If the table look-up results in a negative value, the value will be reset to zero and a warning message (TABLE LOOK-UP RESULTS IN NEGATIVE VALUE OF CREEP MODEL PARAMETER IN ELEMENT ID=****) will be issued.

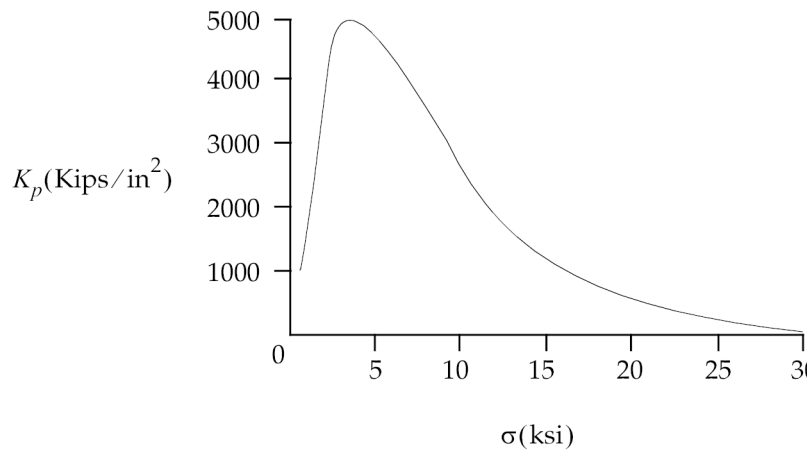


Figure 12-45. K_p Versus σ Example for CREEP

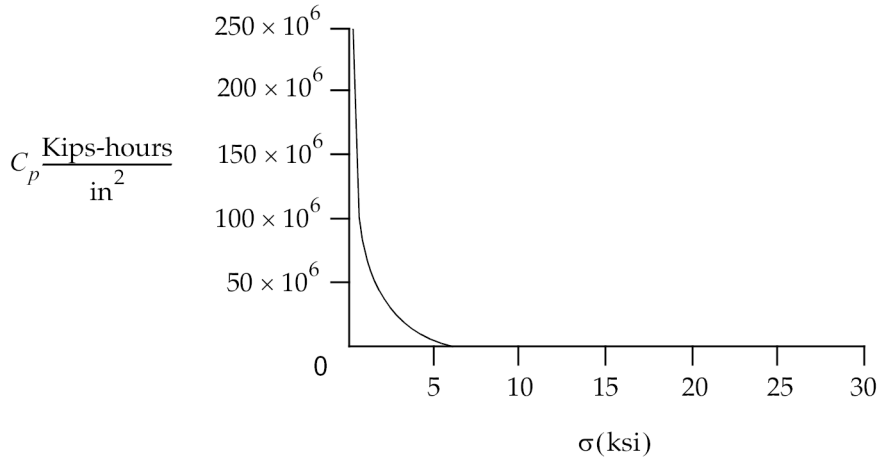


Figure 12-46. C_p Versus σ Example for CREEP

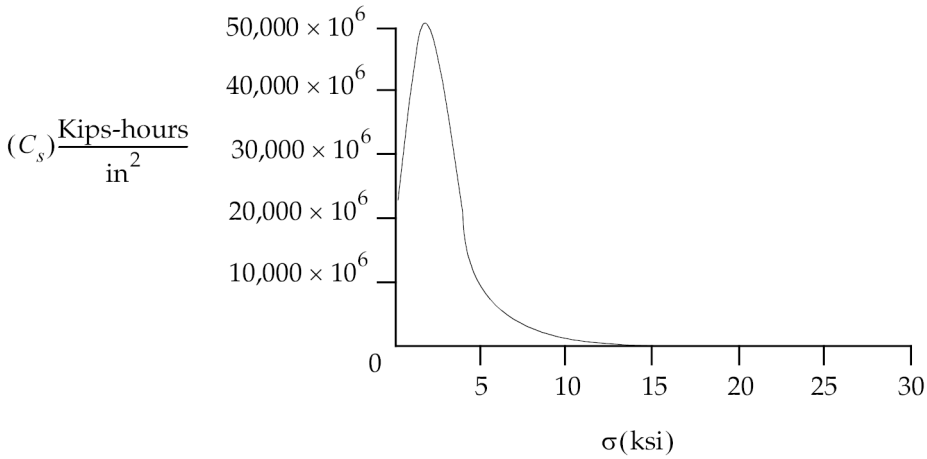


Figure 12-47. C_s Versus σ Example for CREEP

5. Creep analysis requires an initial static solution at *t = 0*, which can be obtained by specifying a subcase that requests an NLPARM entry with DT = 0.0.

REMARKS RELATED TO SOL 601:

1. MID is restricted to the material identification number of a MAT1 entry. CREEP and MAT1 entries model an elastic-creep material with only elastic strain and creep strain effects. CREEP, MAT1 and MATS1 entries model a plastic-creep material with elastic, plastic and creep strain effects.

2. Only FORM=CRLAW is allowed.

3. T0, EXP, TIDKP, TIDCP, TIDCS, and THRESH are ignored.

4. Only TYPE=300 and TYPE=222 are supported.

5. For TYPE=300, the coefficients a, b and d may be made temperature-dependent by the MATTC entry.

CROD

Rod Element Connection

Defines a tension-compression-torsion element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

EXAMPLE:

CROD	12	13	21	23					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PMIC or PROD entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See “CONROD” for alternative method of rod definition.
3. Only one element may be defined on a single entry.

REMARKS RELATED TO SOLS 601 AND 701:

CROD defines a truss element with no torsional stiffness, i.e., only axial force is transmitted by the element.

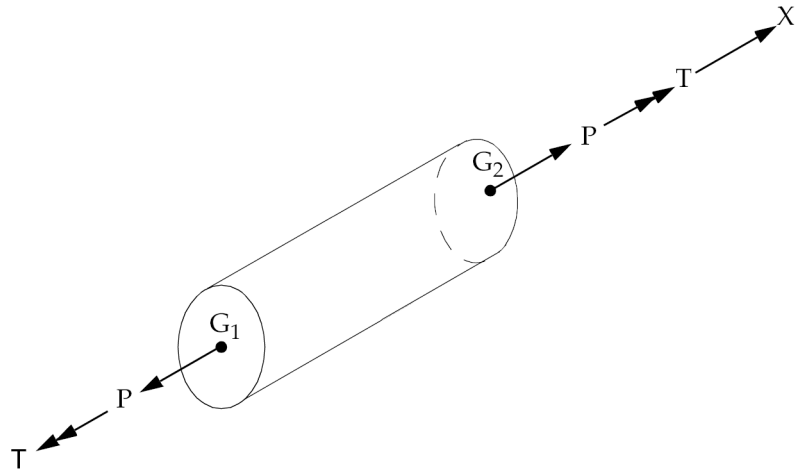


Figure 12-48. CROD Element Internal Forces and Moments

CSET

Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

CSET	124	1	5	23	6	16			

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. CSET and BNDFREE entries are equivalent.
2. If there are no CSETi or BSETi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BSETi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees-of-freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSET1

Free Boundary Degrees-of-Freedom, Alternate Form of CSET Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

EXAMPLE:

CSET1	124	1	5	7	6	9	12	122	
	127								

ALTERNATE FORMATS AND EXAMPLE:

CSET1	C	ID1	“THRU”	ID2					
CSET1	3	6	THRU	32					

CSET1		“ALL”							
CSET1		ALL							

FIELDS:

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)

REMARKS:

1. CSET1 and BNDFREE1 entries are equivalent.

2. If there are no CSETi or BSETi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BSETi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees-of-freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.

4. If PARAM,AUTOSPC is YES then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSHEAR

Shear Panel Element Connection

Defines a shear panel element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

EXAMPLE:

CSHEAR	3	6	1	5	3	7			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHEAR entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ≠ G3 ≠ G4)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.

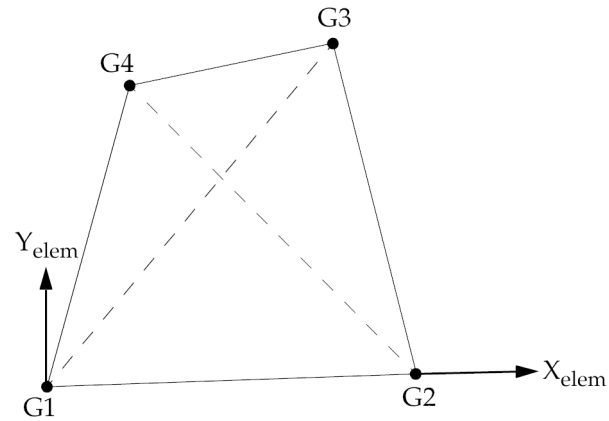


Figure 12-49. CSHEAR Element Connection and Coordinate System

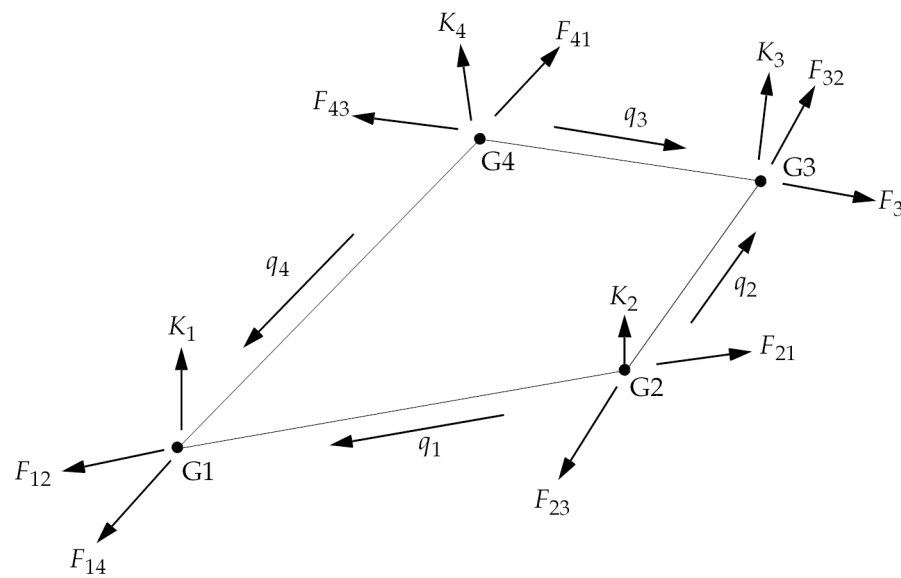


Figure 12-50. CSHEAR Element Corner Forces and Shear Flows

CSSCHD

Aerodynamic Control Surface Schedule Input

Defines a scheduled control surface deflection as a function of Mach number and angle of attack.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSSCHD	SID	AESID	LALPHA	LMACH	LSCHD				

EXAMPLE:

CSSCHD	5	ELEV	12	15	25				

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
AESID	ID of an AESURF Bulk Data entry to which the schedule is being attached.
LALPHA	ID of an AEFACT Bulk Data entry containing a list of angles of attack (in radians) at which schedule information is provided. (Integer > 0, Default = no angle information provided.)
LMACH	ID of an AEFACT Bulk Data entry containing a list of Mach numbers at which schedule information is provided. (Integer > 0, default = no Mach information provided.)
LSCHD	ID of an AEFACT Bulk Data entry which contains the scheduling information. See Remarks 4 and 5 (Integer > 0, no Default).

REMARKS:

1. Control system schedules must be selected with the Case Control command CSSCHD = SID.

2. The AESID cannot appear on an AELINK or TRIM Bulk Data entry for the same subcase.

3. The control surface deflection is computed using a linear interpolation for the Mach number provided on the associated TRIM entry and the angle of attack derived as part of the trim calculation.

4. The LSCHD data are provided as a list of deflections (in radians) as a function of Mach numbers and angles of attack. If there are NMACH Mach numbers and NALPHA angles of attack, the first NALPHA deflections are for the first Mach number, the next NALPHA are for the second Mach number, and so on, until the last NALPHA deflections are for the final Mach number.

5. if LALPHA is blank, LSCHD contains NMACH deflections to define the Mach schedule. If LMACH is blank, LSCHD contains NALPHA deflections to define the angle of attack schedule.

6. LALPHA and LMACH cannot be simultaneously blank. If LALPHA or LMACH are not blank, at least two values of angle of attack or Mach number must be defined in order to perform interpolation.

7. If the Mach number or angle of attack is outside the range specified by the tabulated values, the value at the table end is used. That is, data are not extrapolated.

CSUPER

Secondary Superelement Connection

Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSUPER	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

EXAMPLE:

CSUPER	120003	21	3	6	4	10			

FIELDS:

Field	Contents
SSID	Coded identification number for secondary superelement. See Remark 1 . (Integer > 0)
PSID	Identification number for referenced primary superelement. See Remark 2 . (Integer > 0 or blank)
GPi	Grid or scalar point identification numbers of the exterior points of the secondary superelement. See Remark 3 . (Integer > 0)

REMARKS:

1. The value of SSID is written in the form XXX0000 + n, where n is the referenced secondary superelement identification number and n must be less than 10000 and XXX is a displacement component sign reversal code as follows:

   The sign reversal code specifies the displacement component(s) normal to the plane of the mirror through which the reflection is to be made

Blank or 0	No reversal for identical superelement. If PSID is preceded by a minus sign and there is no xxx code on SSID, then a z-reversal mirror is generated.	
1 2 3	x-reversal y-reversal z-reversal}	Mirror Images
12 23 31	x and y-reversal y and z-reversal z and x-reversal}	Identical Images
123	x, y, and z-reversal	Mirror Images

2. If PSID = 0 or blank, the superelement boundary matrices are obtained from an external source (such as a database or external file). See also PARAM, EXTOUT.

   If PSID ≠ 0, the secondary superelement is identical to, or is a mirror image of, a primary superelement.

3. For identical or mirror image superelements, the grid point IDs, GPi, may appear in any order. However, if they are not in the same order as the external GRIDs of the primary superelement, then the SEQSEP entry is also required. In case of external superelements, the GRID IDs must be in the order that the terms in the associated matrices occur in.

4. Image superelements and their primaries must be congruent. The identical or mirror image superelement must have the same number of exterior grid points as its primary superelement. The exterior grid points of the image superelement must have the same relative location to each other as do the corresponding points of the primary superelement. The global coordinate directions of each exterior grid point of the image superelement must have the same relative alignment as those of the corresponding grid points of the primary superelement. If congruency

is not satisfied because of round-off, then the tolerance may be adjusted with PARAM,CONFAC or DIAG 37.

5. For superelements from an external source, please refer to PARAMS EXTDR,EXTDROUT, EXTDRUNT, EXTOUT,, and EXTUNIT.

CSUPEXT

Superelement Exterior Point Definition

Assigns exterior points to a superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CSUPEXT	SEID	GP1	GP2	GP3	GP4	GP5	GP6	GP7	

EXAMPLE:

CSUPEXT	2	147	562	937					

ALTERNATE FORMAT AND EXAMPLE:

CSUPEXT	SEID	GP1	“THRU”	GP2					

CSUPEXT	5	12006	THRU	12050					

FIELDS:

Field	Contents
SEID	Identification number of a primary superelement. (Integer > 0)
GPi	Grid or scalar point identification number in the downstream superelement or residual structure. (Integer > 0 or “THRU”; for “THRU” option, GP1 < GP2)

REMARKS:

1. Grid or scalar points are connected (that is, are exterior) to a superelement only if they are connected by structural, rigid, or plot elements. MPC entries are not regarded as elements. This entry is a means of providing connectivity for this purpose.

2. Open sets are allowed with the “THRU” option.

3. Scalar points may be interior to the residual structure (SEID = 0) only.

4. This entry may be applied only to the primary superelements. The CSUPER entry is used for secondary superelements (identical image, mirror image, and external superelements).

CTETRA

Four-Sided Solid Element Connection

Defines the connections of the four-sided solid element with four to ten grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

EXAMPLE:

CTETRA	112	2	3	15	14	4	103	115	
	5	16	8	27					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0; Required)
PID	Property identification number of a PSOLID, PLSOLID, or PMIC entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer > 0 or blank)

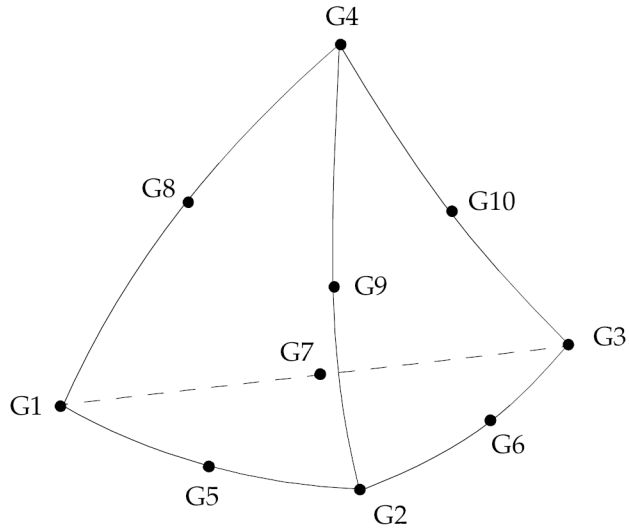


Figure 12-51. CTETRA Element Connection

REMARKS:

1. Element ID numbers must be unique with respect to all other element ID numbers.

2. The topology of the diagram must be preserved, i.e., G1, G2, G3 define a triangular face; G1, G8, and G4 are on the same edge, etc.

3. The edge points, G5 to G10, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.

4. Components of stress are output in the material coordinate system. See remark 6 on the PSOLID bulk entry for hyperelastic and nonlinear exceptions.

5. For nonhyperelastic elements, the element coordinate system is derived from the three vectors R, S, and T, which join the midpoints of opposite edges.

   R vector joins midpoints of edges G1-G2 and G3-G4.

   S vector joins midpoints of edges G1-G3 and G2-G4.

   T vector joins midpoints of edges G1-G4 and G2-G3.

The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that, if the R, S, and T vectors are described in the element coordinate system, a 3x3 positive definite symmetric matrix would be produced.)

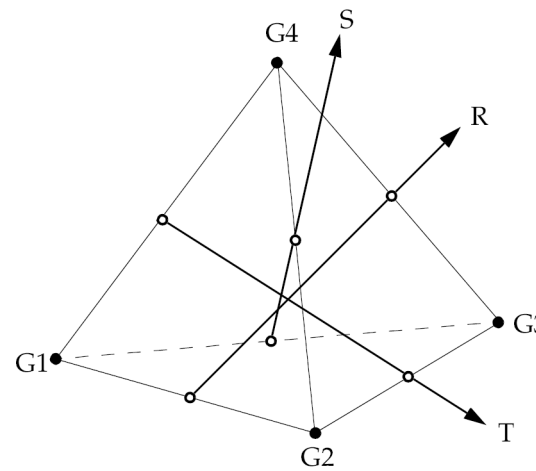


Figure 12-52. CTETRA Element R, S, and T Vectors

6. It is recommended that the edge points be located within the middle third of the edge.

7. For hyperelastic elements, the plot codes are specified under the CTETRAFD element name in “Item Codes”.

8. By default, all of the six edges of the element are considered straight unless any of G5 through G10 are specified.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 701, only elements with 4 grid points are allowed.

2. For SOL 601, 10-node CTETRA elements may be converted to 14-node CTETRA elements (1 additional node on the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. 11-node CTETRA elements are more effective than 10-node CTETRA for analysis of incompressible media and inelastic materials,

e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

CTRAX3

Axisymmetric Triangular Element Connection

Defines an isoparametric and axisymmetric triangular cross-section ring element for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTRAX3	EID	PID	G1	G2	G3	TH			

EXAMPLE:

CTRAX3	203	100	10	20	30	30.0			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID or PLSOLID entry. (Integer > 0; Default=EID)
Gi	Grid point identification number of connected points. (Unique integers > 0)
TH	Material property orientation angle in degrees. See Remark 4. TH is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of the axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), the plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and the plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. When axisymmetric elements are defined on the XZ-plane, X is the radial direction, and Z is the axial direction. The grid points defining these elements must have X ≥ 0. See Figure 54.

   When axisymmetric elements are defined on the XY-plane, Y is the radial direction, and X is the axial direction. The grid points defining these elements must have Y ≥ 0. See Figure 55.

4. For orthotropic materials, the TH angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 54 and 55 show the positive sense of the TH angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. The initial element coordinate system is the basic coordinate system.

6. See the remarks on the PSOLID entry for information on stress and strain output coordinate systems.

7. The CQUADX4, CQUADX8, CTRAX3, and CTRAX6 elements behave linearly in SOL 106 and 129 when not hyperelastic.

8. For hyperelastic element, the plot codes are specified under the CTRAX3FD element name in Item Codes Overview.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement

coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. A concentrated load (e.g. FORCE entry) at Gi is the total of the force around the circumference. Reaction force and applied load output are the same. For example, to apply a distributed load of 135.0 Newton/mm on a single grid where the radius is 0.5 mm:

    The value entered on a FORCE entry = (Distributed force * 2 * π * Radius)

    = 135.0 N/mm * 2 * π * 0.5 mm

    = 424.1 N

    See system cell 587 for information on the pre-version 10 axisymmetric element behavior.

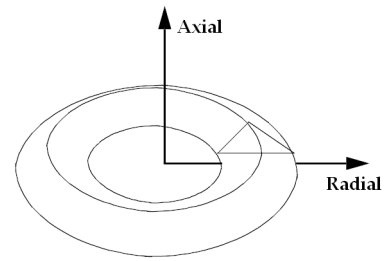


Figure 12-53. CTRAX3 Element Idealization

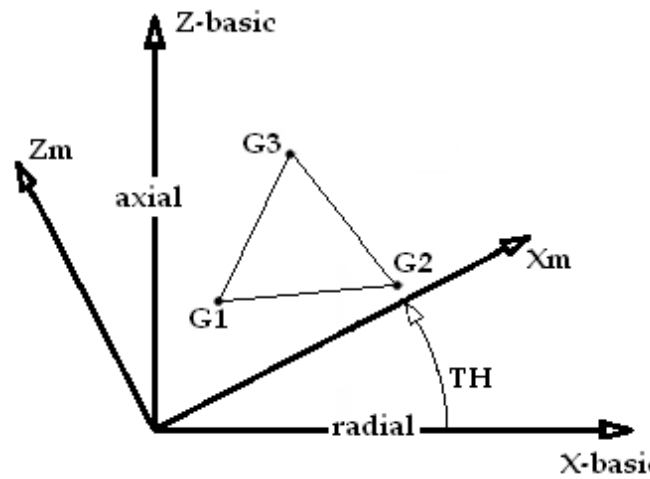


Figure 12-54. CTRAX3 Element Geometry and Coordinate Systems, XZ-Plane

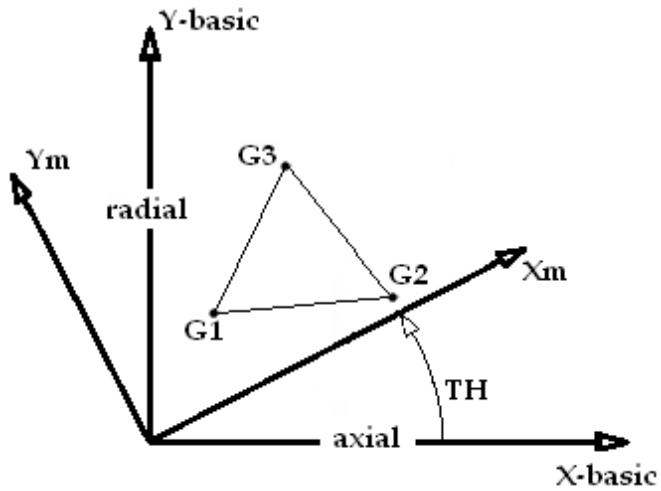


Figure 12-55. CTRAX3 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. When TH is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CTRAX6

Axisymmetric Triangular Element Connection

Defines an isoparametric and axisymmetric triangular cross-section ring element with midside nodes for use in linear and fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTRAX6	EID	PID	G1	G2	G3	G4	G5	G6	
	TH								

EXAMPLE:

CTRAX6	203	100	10	20	30	40	50	60	
	30.0								

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID or PLSOLID entry. (Integer > 0; Default=EID)
Gi	Grid point identification numbers of connected points (midside grids cannot be eliminated). (Unique integers > 0)
TH	Material property orientation angle in degrees. See Remark 4. TH is ignored for hyperelastic elements. (Real; Default = 0.0)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points of the axisymmetric elements (CTRAX3, CQUADX4, CTRAX6, CQUADX8, CCHOCK3, CCHOCK4, CCHOCK6, CCHOCK8), the plane stress elements (CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8), and the plane strain elements (CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8) must all lie in either the XZ-plane, or all in the XY-plane of the basic coordinate system. The software automatically determines the orientation.

3. When axisymmetric elements are defined on the XZ-plane, X is the radial direction, and Z is the axial direction. The grid points defining these elements must have X ≥ 0. See Figure 57.

   When axisymmetric elements are defined on the XY-plane, Y is the radial direction, and X is the axial direction. The grid points defining these elements must have Y ≥ 0. See Figure 58.

4. For orthotropic materials, the TH angle orients the principal material coordinates of the element relative to the basic coordinates. Figures 57 and 58 show the positive sense of the TH angle when the element is defined on the XZ-plane and XY-plane, respectively.

5. The initial element coordinate system is the basic coordinate system.

6. See the remarks on the PSOLID entry for information on stress and strain output coordinate systems.

7. The CQUADX4, CQUADX8, CTRAX3, and CTRAX6 elements behave linearly in SOL 106 and 129 when not hyperelastic.

8. For hyperelastic element, the plot codes are specified under the CTRAX6FD element name in Item Codes Overview.

9. For any grid point (Gi) selected on axisymmetric, plane stress, or plane stain elements, if you select a displacement coordinate system with the CD field on the GRID entry, you must orient the system according to the following rules:

   - If the elements are defined on the XY-plane of the basic coordinate system, the Z-axis is the out-of-plane direction. In this case, you must orient the Z-axis (φ-axis for a spherical system) of the displacement coordinate system to be parallel with the Z-axis of the basic coordinate system.

   - If the elements are defined on the XZ-plane of the basic coordinate system, the Y-axis is the out-of-plane direction. In this case, you must orient the Y-axis (θ-axis for the cylindrical and spherical systems) of the displacement

coordinate system to be parallel with the Y-axis of the basic coordinate system.

10. A concentrated load (e.g. FORCE entry) at Gi is the total of the force around the circumference. Reaction force and applied load output are the same. For example, to apply a distributed load of 135.0 Newton/mm on a single grid where the radius is 0.5 mm:

    The value entered on a FORCE entry = (Distributed force * 2 * π * Radius)

    = 135.0 N/mm * 2 * π * 0.5 mm

    = 424.1 N

    See system cell 587 for information on the pre-version 10 axisymmetric element behavior.

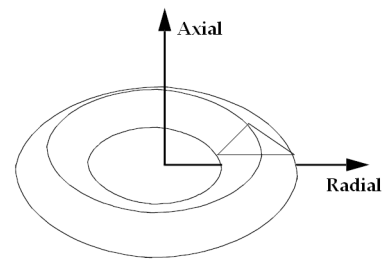


Figure 12-56. CTRAX6 Element Idealization

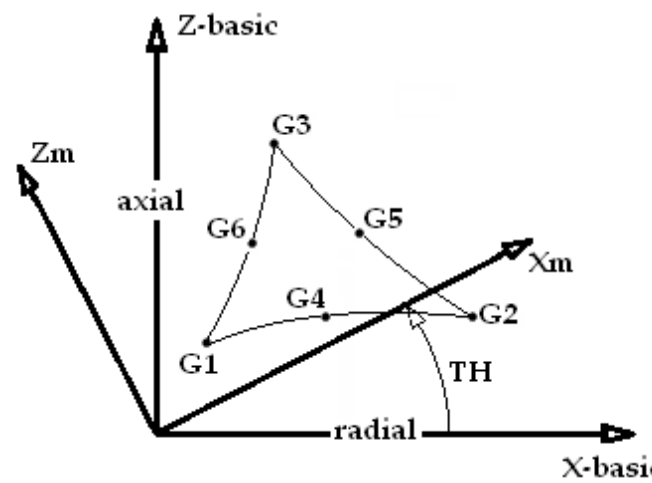


Figure 12-57. CTRAX6 Element Geometry and Coordinate Systems, XZ-Plane

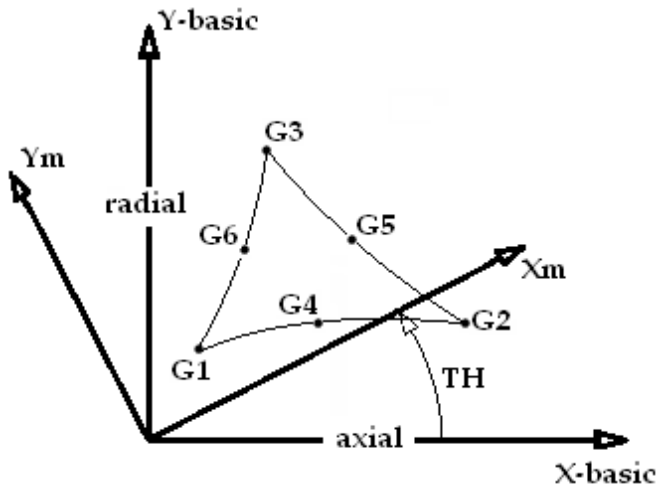


Figure 12-58. CTRAX6 Element Geometry and Coordinate Systems, XY-Plane

REMARKS RELATED TO SOL 601:

1. When TH is defined, the positive element normal direction, which is defined by G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative Y-direction (if in XZ-plane) or the positive Z-direction (if in XY-plane) of the basic system.

CTRIA3

Triangular Plate Element Connection

Defines an isoparametric membrane-bending or plane strain triangular plate element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

EXAMPLE:

CTRIA3	111	203	31	74	75	3.0	0.98		
			1.77	2.04	2.09				

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PCOMP, PCOMPG, PLPLANE, PMIC, or PSHELL entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. MCID is ignored for hyperelastic elements. See Remark 11. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.)

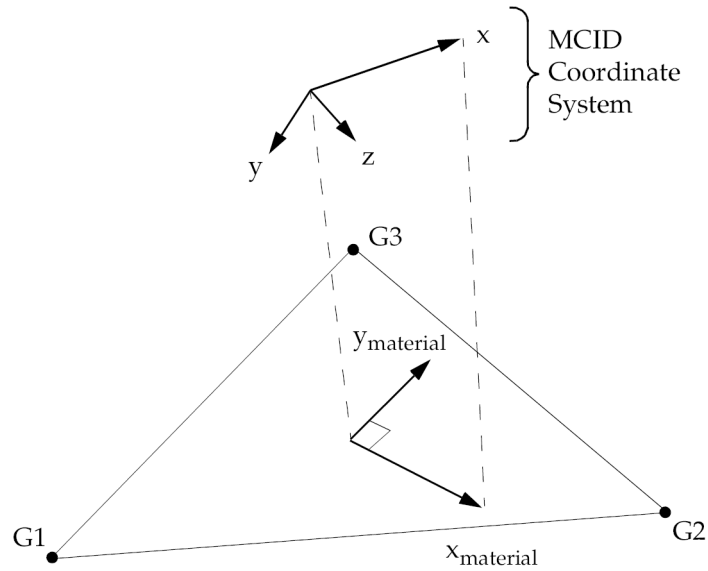


Figure 12-59. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remarks 5 and 6. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)
Ti	Thickness of element at grid points G1 through G3 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. Ti is ignored for hyperelastic elements. See Remark 4. (Real > 0.0 or blank. See Remark 2 for default.)

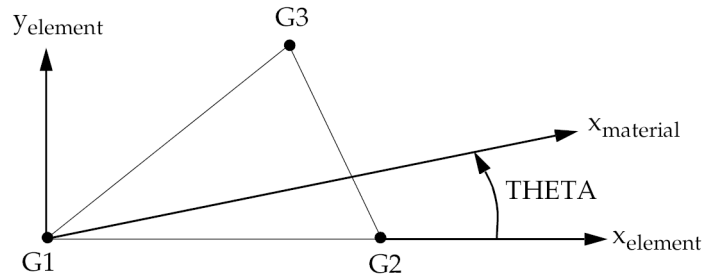


Figure 12-60. CTRIA3 Element Geometry and Coordinate Systems

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.

3. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

4. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

5. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CTRIA3 entry.

   ZOFFS should not be used when differential stiffness is calculated in solutions 105 and 200, and solutions 103, 107 - 112 using STATSUB, since the differential stiffness calculation does not include the offset vectors.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

6. ZOFFS has the following caveats in nonlinear solutions 106 and 129:

- For geometry nonlinear only, the differential stiffness calculation does not include the offset vectors. The exclusion of the offset in the differential stiffness will have an adverse effect on convergence, however, if the solution converges, the results will be correct. In a nonlinear solution, equilibrium (and convergence) is achieved when internal and external forces balance. The offset is correctly accounted for in the internal and external force calculations even though it is not accounted for in the differential stiffness.

- ZOFFS should not be defined on elements which use MATS1 nonlinear material definitions.

- ZOFFS is ignored for hyperelastic elements.

- ZOFFS should not be used in nonlinear solutions 106 and 129 in combination with thermal loads.

7. The reference coordinate system for the output of stress, strain and element force depends on the element type.

   - For CTRIA3 elements which are not hyperelastic, the reference coordinate system for output is the element coordinate system. For SOL 106 geometry nonlinear analysis, the nonlinear stresses are output in deformed element coordinate system.

   - For hyperelastic elements, the stress and strain are output according to CID on the PLPLANE entry.

8. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in “Item Codes”.

9. System Cell 218 allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. The default value is 10.0 degrees.

10. By default, all of the three edges of the element are considered straight.

11. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

12. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

REMARKS RELATED TO SOLS 601 AND 701:

1. ZOFFS is ignored.
2. For geometry nonlinear analysis, the nonlinear stresses are written in undeformed element coordinate system.

CTRIA6

Curved Triangular Shell Element Connection

Defines a curved triangular shell element or plane strain with six grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID	ZOFFS	T1	T2	T3	TFLAG			

EXAMPLE:

CTRIA6	302	3	31	33	71	32	51	52	
	45	0.03	0.02	0.025	0.025				

FIELDS:

Field	Contents
EID	Element Identification number. (Integer > 0)
PID	Property identification number of PSHELL, PCOMP, PCOMPG, or PLPLANE entry. (Integer > 0)
G1, G2, G3	Identification numbers of connected corner grid points. (Unique Integers > 0)
G4, G5, G6	Identification number of connected edge grid points. Optional data for any or all three points. (Integer ≥ 0 or blank)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. MCID is ignored for hyperelastic elements. See Remark 12. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.)

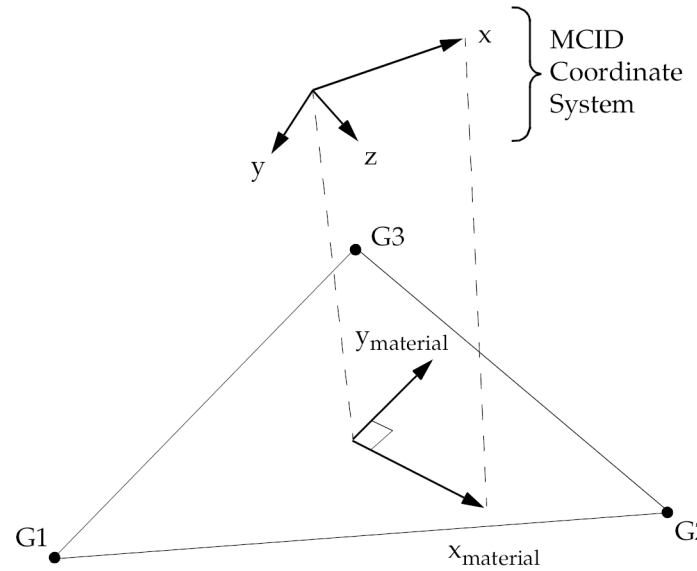


Figure 12-61. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 8. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)
Ti	Thickness of element at grid points G1 through G3 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. Ti is ignored for hyperelastic elements. See Remark 6. (Real > 0.0 or blank. See Remark 4 for default.)

REMARKS:

1. Element identification numbers should be unique with respect to all other element IDs.

2. Grid points G1 through G6 must be numbered as shown in Figure 12-62.

3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between $x_{material}$ and the line of constant η.

4. T1, T2, and T3 are optional. If not supplied, they will be set equal to the value of T on the PSHELL entry.

5. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

6. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

7. It is recommended that the mid-side grid points be located within the middle third of the edge.

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CTRIA6 entry.

   ZOFFS should not be used when differential stiffness is calculated in solutions 105 and 200, and solutions 103, 107 - 112 using STATSUB, since the differential stiffness calculation does not include the offset vectors.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

9. If all mid-side grid points are deleted, then the element will be excessively stiff and the transverse shear forces will be incorrect. A User Warning Message is printed. A CTRIA3 element entry is recommended instead. If the element is hyperelastic, then the element is processed identically to the hyperelastic CTRIA3 element.

10. For a description of the element coordinate system, see “Plate and Shell Elements” in the *NX Nastran Element Library*. Stresses and strains are output in

the local coordinate system identified by x_l and y_l in Figure 12-62. For hyperelastic elements, stresses and strains are output in the coordinate system defined by the CID field on the PLPLANE entry.

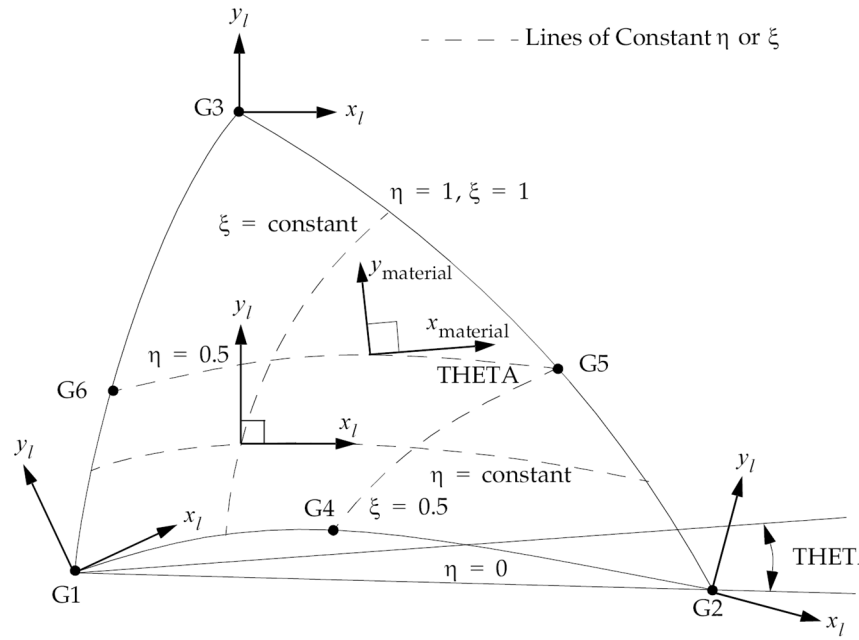


Figure 12-62. CTRIA6 Element Geometry and Coordinate Systems

11. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in "Item Codes".

12. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

13. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

REMARKS RELATED TO SOL 601:

1. Only elements with 3 or 6 grid points are allowed.
2. ZOFFS is ignored.
3. When used as a plane strain element, ELCV=1 in the NXSTRAT entry will convert a 6-node triangular element to a 7-node triangular element (with 1 additional node at the centroid of the element).
4. In a nonlinear analysis, both linear and nonlinear stress/strain data blocks are output for CTRIA6 elements in the op2 file. Note that in a future release, the output of linear stress/strain data blocks for CTRIA6 elements may be discontinued for a nonlinear analysis.
5. For geometry nonlinear analysis, the nonlinear stresses are written in undeformed element coordinate system.

CTRIAR

Triangular Plate Element Connection

Defines an isoparametric membrane-bending triangular plate element. It is a companion to the CQUADR element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

EXAMPLE:

CTRIAR	111	203	31	74	75	3.0			
			1.77	2.04	2.09				

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP, or PCOMPG entry. (Integer > 0; Default = EID)
G1, G2, G3	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. See Remark 10. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.)

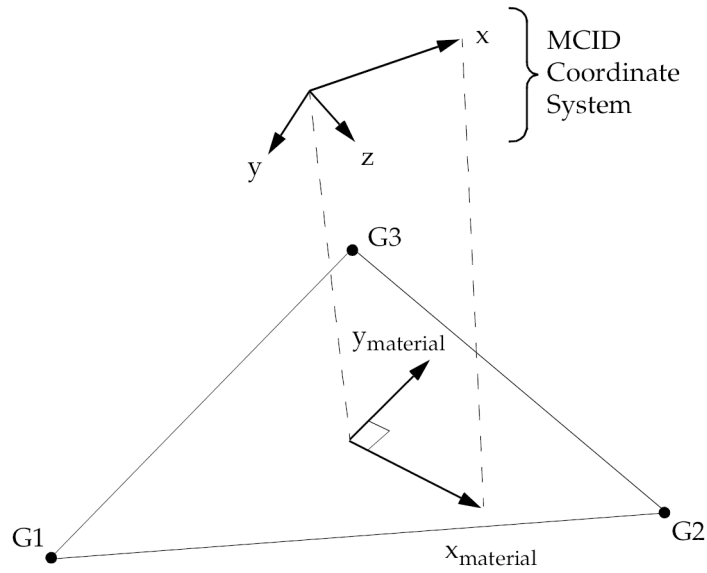


Figure 12-63. MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 9. (Real)
TFLAG	Integer Flag which specifies how Ti is used to define thickness of element. (0, 1, or blank)
Ti	Thickness of element at grid points G1 through G3 if TFLAG=0 or blank. If TFLAG=1, thickness becomes a product of Ti and the thickness on the PSHELL card. See Remark 4. (Real > 0.0 or blank. See Remark 2 for the default.)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.

3. When the PID refers to either a PCOMP or PCOMPG entry, the software converts the PCOMP/PCOMPG entry to equivalent PSHELL/MAT2 entries. If Ti is defined, they override or scale the values on the equivalent PSHELL entry. It should be noted that ply-stress and ply-strain recovery uses the ply thicknesses defined on the PCOMP/PCOMPG entries, and not Ti.

4. By default, a fatal error occurs if a Ti value is zero. To allow for Ti values of zero, use System Cell 495.

5. Stresses and strains are output in the element coordinate system at the centroid unless the CORNER or CUBIC entry is specified on the STRESS case control card. Specifying CORNER or CUBIC will result in stresses at the centroid and corners. See “Shell element item code remarks” at the beginning of the item codes chapter.

6. The rotational degrees-of-freedom normal to the element are active in the element formulation and must not be constrained unless at a boundary. If they are constrained, then inaccurate results will be obtained.

7. As compared to the CTRIA3 element, the CTRIAR element:

   - Accounts for normal rotational DOF to provide improved membrane accuracy.

   - Is less sensitive to high aspect ratios and values of Poisson’s ratio near 0.5.

8. The CTRIAR element is not supported in SOL 401. The CTRIAR element is supported in all other linear structural and heat transfer solutions. If used in SOL 106 or 129, the CTRIAR element behaves linear elastically in a material nonlinear analysis and the stiffness does not reformulate in a geometric nonlinear analysis.

9. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and the MID2 fields must be specified on the PSHELL entry referenced by PID on the CTRIAR entry.

   ZOFFS is ignored in heat transfer solutions 153 and 159 including any solution subcase which includes the case control ANALYSIS=HEAT.

10. If the x-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable. A fatal error will occur when the angle between a shell element face normal and the x-direction of the material coordinate system (MCID) is less than the value specified by System Cell 489.

11. The MCID should be less than 2,000,000 when running with the LP-64 executable. It should be less than 99,999,999 when running with the ILP-64 executable and SYSTEM(525)=0 (default), and less than 99,999,999,999 when running with the ILP-64 executable and SYSTEM(525)=1.

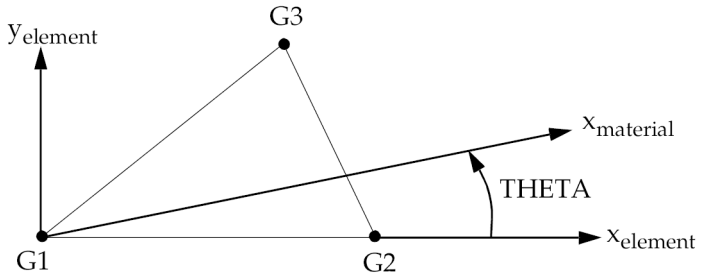


Figure 12-64. CTRIAR Element Geometry and Coordinate Systems

REMARKS RELATED TO SOL 601:

1. ZOFFS is ignored.

2. CTRIAR can only be used in linear analysis. It is recommended that CTRIA3 be used instead of CTRIAR in SOL 601 analysis. If the CTRIAR element is used in SOL 601, the CTRIAR element behaves linear elastically in a material nonlinear analysis and the stiffness does not reformulate in a geometric nonlinear analysis. The CTRIAR element is not supported in SOL 701.

3. For geometric nonlinear analysis, the nonlinear stresses are written in the undeformed element coordinate system.

CTUBE

Tube Element Connection

Defines a tension-compression-torsion tube element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CTUBE	EID	PID	G1	G2					

EXAMPLE:

CTUBE	12	13	21	23					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PTUBE entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Only one tube element may be defined on a single entry.

CVISC

Viscous Damper Connection

Defines a viscous damper element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

EXAMPLE:

CVISC	21	6327	29	31					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PVISC entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

REMARKS:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Only one viscous damper element may be defined on a single entry.

CWELD

Weld or Fastener Element Connection

Defines a weld or fastener connecting two surface patches or points.

ELPAT FORMAT: (SEE REMARK 2)

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	“ELPAT”	GA	GB			
	SHIDA	SHIDB							
	XS	YS	ZS						

ELPAT EXAMPLE:

CWELD	7	29	233	ELPAT					
	15	28							

PARTPAT FORMAT: (SEE REMARK 2)

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	“PARTPAT”	GA	GB			
	PIDA	PIDB							
	XS	YS	ZS						

PARTPAT EXAMPLE:

CWELD	8	30	133	PARTPAT					
	101	201							

GRIDID FORMAT: (SEE REMARK 3)

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	“GRIDID”	GA	GB	SPTYP		
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	
	XS	YS	ZS						

GRIDID EXAMPLE:

CWELD	7	29	233	GRIDID			QT		
	15	28	31	35	46	51	55	60	
	3	5	8						

ELEMID FORMAT: (SEE REMARK 4)

CWELD	EWID	PWID	GS	“ELEMID”					
	SHIDA	SHIDB							
	XS	YS	ZS						

ELEMID EXAMPLE:

CWELD	3	28	354	ELEMID					
	15	16							

ALIGN FORMAT: (SEE REMARK 5)

CWELD	EWID	PWID		“ALIGN”	GA	GB			

ALIGN EXAMPLE:

CWELD	7	29		ALIGN	103	259			

FIELDS:

Field	Contents
EWID	CWELD element identification number. See Remark 6. (Integer > 0)
PWID	Property identification number of a PWELD entry. (Integer > 0)
“ELPAT”	Designates the input format as “ELPAT”. See Remark 2. (Character; No default)
“PARTPAT”	Designates the input format as “PARTPAT”. See Remark 2. (Character; No default)
“GRIDID”	Designates the input format as “GRIDID”. See Remark 3. (Character; No default)
“ELEMID”	Designates the input format as “ELEMID”. See Remark 4. (Character; No default)
“ALIGN”	Designates the input format as “ALIGN”. See Remark 5. (Character; No default)
GA, GB	Grid identification numbers used to define the location of the connector on surface patch A and surface patch B, respectively when format is “ELPAT”, “PARTPAT”, “ELEMID”, or GRIDID. When format is “ALIGN”, GA and GB are required, and they must be vertex nodes of shell elements. See Remark 1. (Integer > 0 or blank)
GS	Grid point identification number used to define the end locations of the connector when GA/GB are undefined. The specific meaning of GS depends on whether it is used to define a location on a shell element patch when format is “ELPAT” , “PARTPAT”, “ELEMID”, or “GRIDID”, or is used to define a point location when the format is “ELEMID”, or “GRIDID”. See Remarks 1–6 for format specific definitions of GS. (Integer > 0)

Field	Contents
XS,YS,ZS	Defines the location of the connector on a shell element patch when format is “ELPAT” , “PARTPAT”, “ELEMID”, or “GRIDID”, and when GA,/GB and GS are all undefined. See Remark 1. (Integer > 0)
PIDA	The physical property identification number of a PSHELL bulk data entry used to define surface patch A when format is “PARTPAT”. (Integer > 0)
PIDB	The physical property identification number of a PSHELL bulk data entry used to define surface patch B when format is “PARTPAT”. (Integer > 0)
SHIDA	The shell element identification number used to define surface patch A when format is “ELPAT” or “ELEMID”. (Integer > 0)
SHIDB	The shell element identification number used to define surface patch B when format is “ELPAT” or “ELEMID”.(Integer > 0)
SPTYP	String indicating the type of surface patches A and B when format is “GRIDID”. SPTYP=”QQ”, “TT”, “QT”, “TQ”, “Q” or “T”. Indicates quadrilateral (“Q”) or triangular (“T”) surface patch. See Remark 3. (Character)
GAi	Grid identification numbers defining surface patch A when format is “GRIDID”. GA1 to GA3 are required. See Remark 3. (Integer > 0)
GBi	Grid identification numbers defining surface patch B when format is “GRIDID”. See Remark 3. (Integer > 0)

REMARKS:

1. GA/GB, or GS, or XS/YS/ZS, in that order of precedence, determine the location(s) of the CWELD connection on surface patches A and/or B, when format is “ELPAT”, “PARTPAT”, “ELEMID”, or “GRIDID”. The connection location on patch A is a specification of grid point GA, or a projection of GS or XS/YS/ZS normal to surface patch A. The location on patch B is a specification of grid point GB, or a projection of GS or XS/YS/ZS normal to surface patch B. When GS or XS/YS/ZS are used, a normal projection must exist in order to define a valid connection element. GS or XS/YS/ZS do not need to lie on either surface patch A or B. If GS or XS/YS/ZS are used to define the connection location, grid points are internally created at the connection location with an ID. The numbering of the

internal IDs is described on the parameter OSWPPT for the formats “ELEMID” and “GRIDID” formats, and in Remark 8 for the formats “ELPAT” and “PARTPAT”.

2. If the “ELPAT” format is used, the shell element selected on SHIDA along with those connected and the shell element specified on the SHIDB along with those connected will create shell element patches A and B, respectively. The “PARTPAT” format works similarly, except that the shell elements for patches A and B are selected with the PSHELL property ID specified on PIDA and PIDB, respectively. See Figure 65.

   For both “ELPAT” and “PARTPAT”, shell element patches A and B are connected together with a weld of diameter D specified on the PWELD bulk entry. The centers of the weld connection GA and GB on patches A and B are determined as described in remark 1. Virtual grid points GHA1-GHA4 on patch A and GHB1-GHB4 on patch B are created at the corners of a square having an area equivalent to a circle using the diameter D. See Figure 66. The points on each patch are connected together using constraint equations represented by the dashed green lines in Figure 67. The diameter of the CWELD should be entered such that all GHAi and GHBi locations are either within the element in which GA and GB are projected on to, or an element directly adjacent. For example, if any GAi location is within the gray area in Figure 67, CWELD creation may fail. Parameters on the SWLDPRM bulk entry can be used to adjust tolerances and defaults when CWELD creation fails.

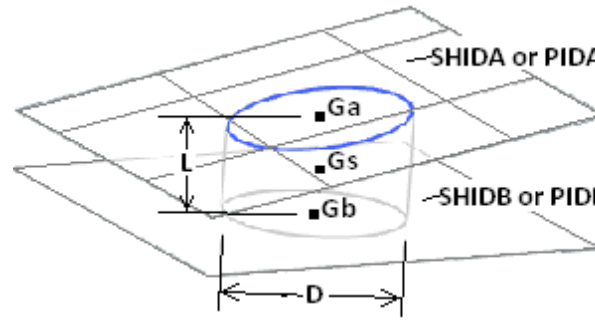


Figure 12-65. Patch-to-Patch Connection Defined with Format ELPAT or PARTPAT

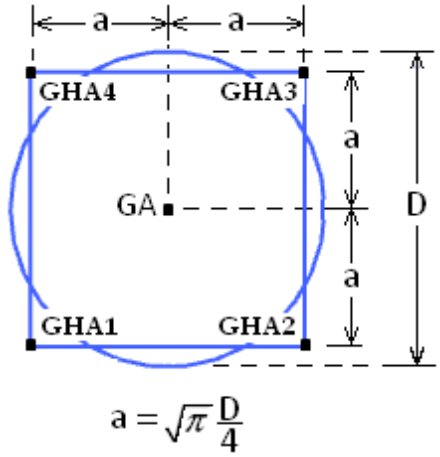


Figure 12-66. GHAi and GHBi Locations with Format ELPAT or PARTPAT

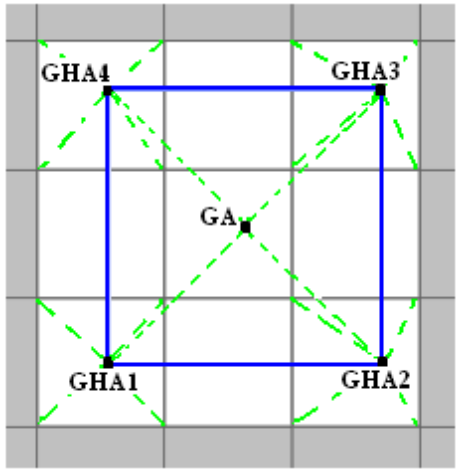


Figure 12-67. Weld Connections with Format ELPAT or PARTPAT

3. The format “GRIDID” defines either a point to element face or an element face to element face connection. For the point to element face connection, you must define GS and GAi. Then it is assumed that GS is a shell vertex grid and GAi are grids describing an element face. For the element face to element face connection, you must define GS, GAi and GBi. Then GAi describes the first element face and GBi the second element face.

   GAi are required for the “GRIDID” format. At least 3 and at most 8 grid IDs may be specified for GAi and GBi, respectively. Triangular and quadrilateral

element definition sequences apply for the order of GAi and GBi, see Figure 12-68. Missing midside nodes are allowed. The envelop that is defined by the grids must represent a valid finite element shape, and not just a string of grids. The envelop may encapsulate more than one finite element.

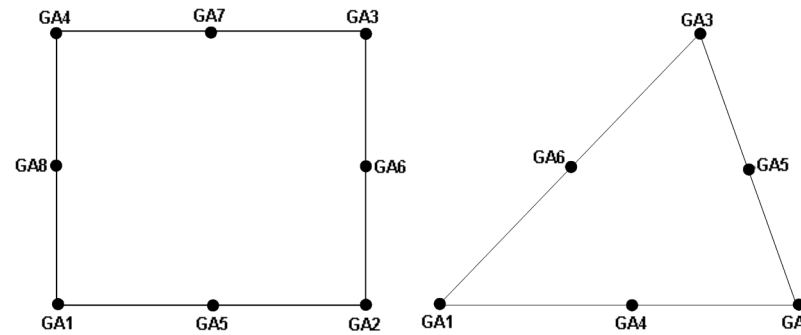


Figure 12-68. Quadrilateral and Triangular Surface Patches Defined with Format GRIDID

SPTYP defines the type of surface patches to be connected. SPTYP is required for the “GRIDID” format to identify quadrilateral or triangular patches. Allowable combinations are:

SPTYP	Description
QQ	Connects a quadrilateral surface patch A (Q4 to Q8) with a quadrilateral surface patch B (Q4 to Q8).
QT	Connects a quadrilateral surface patch A (Q4 to Q8) with a triangular surface patch B (T3 to T6).
TT	Connects a triangular surface patch A (T3 to T6) with a triangular surface patch B (T3 to T6).
TQ	Connects a triangular surface patch A (T3 to T6) with a quadrilateral surface patch B (Q4 to Q8).
Q	Connects the shell vertex grid GS with a quadrilateral surface patch A (Q4 to Q8) if surface patch B is not specified
T	Connects the shell vertex grid GS with a triangular surface patch A (T3 to T6) if surface patch B is not specified

4. The “ELEMID” format defines a point to patch connection, GS to SHIDA or a patch to patch connection, SHIDA to SHIDB.

5. The “ALIGN” format defines a point to point connection. GA and GB are required, and they must be vertex nodes of shell elements. GA and GB are not required for the other formats.

6. CWELD defines a flexible connection between two surface patches, between a point and a surface patch, or between two shell vertex grid points. See Figure 12-69 through Figure 12-71.

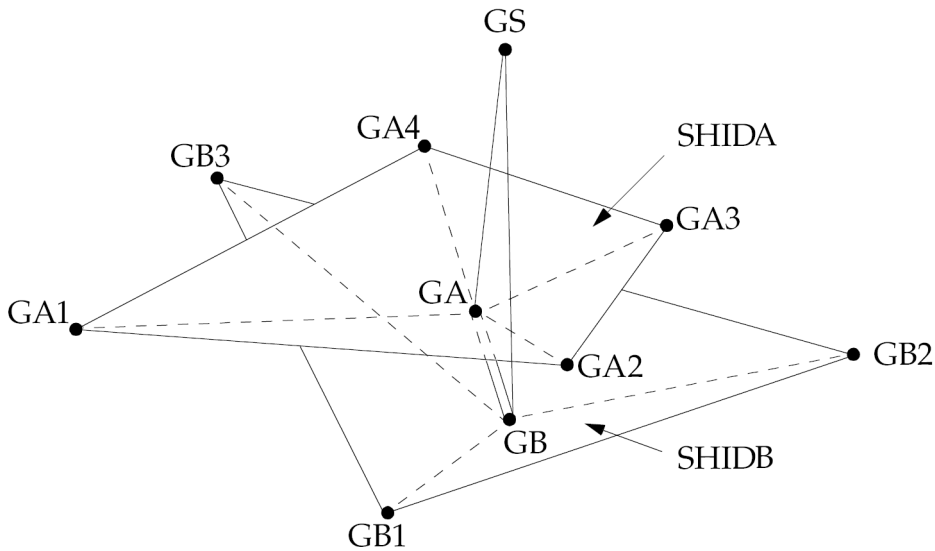


Figure 12-69. Patch-to-Patch Connection Defined with Format GRIDID or ELEMID

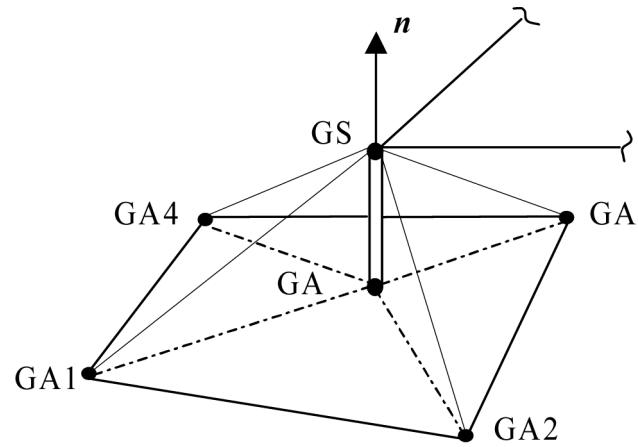


Figure 12-70. Point-to-Patch Connection Defined with Format GRIDID or ELEMID

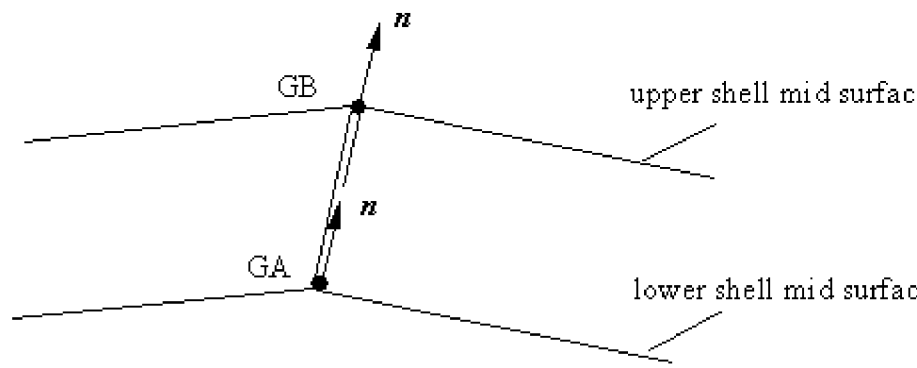


Figure 12-71. Point-to-Point Connection Defined with Format ALIGN

7. Forces and moments are output in the element coordinate system. The element x-axis is in the direction of GA to GB, see Figure 12-72. If zero length (GA and GB are coincident), the x-axis will be defined by the normal from shell A. The element y-axis is perpendicular to the element x-axis and is lined up with the closest axis of the basic coordinate system. The element z-axis is the cross product of the element x- and y-axis. The output of the forces and moments including the sign convention is the same as in the CBAR element.

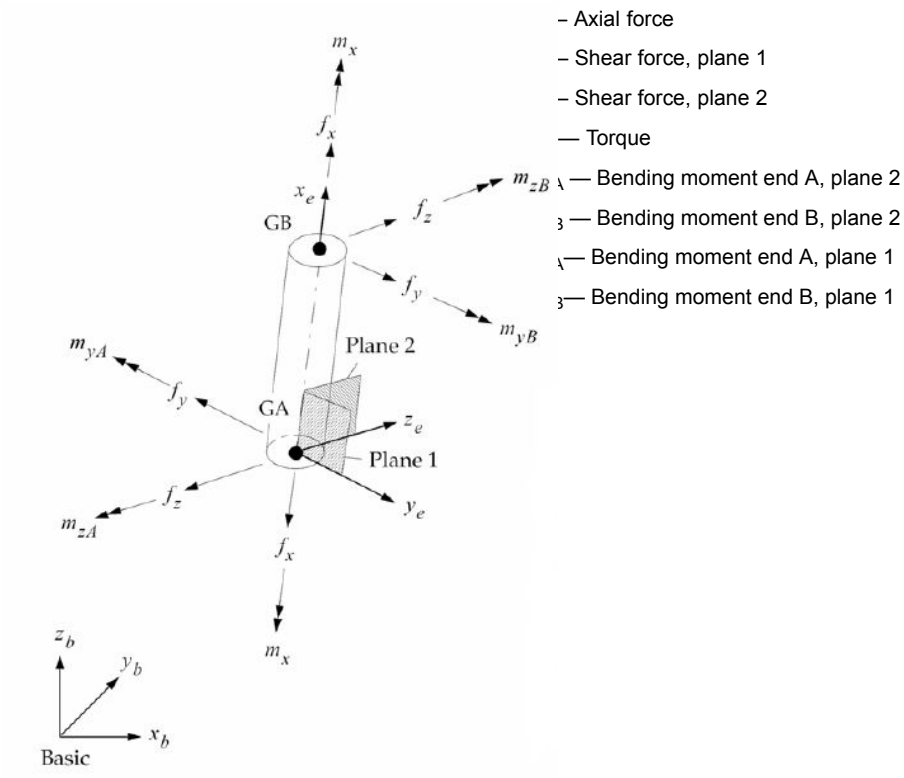


Figure 12-72. Element Coordinate System and Sign Convention of Element Forces

8. When the format is ELPAT or PARTPAT and GA and GB are specified, the displacements for these grids will be printed/stored as requested on the DISPLACEMENT case control command. If these points are not specified, they will be generated by the software, and the displacements for these grids can only be written to the .f06 file with grid IDs calculated as follows:
   Grid ID = Value of SYSTEM(178) + (CWELD ID) + (end ID)
   where the end ID is 1 or 2 for the two ends. SYSTEM(178) defaults to 101,000,000.
   The new grid IDs are only used for labeling the displacement output and do not exist in the analysis or internal tables. This displacement result is not written to the .op2 or punch file.

CYAX

Grid Points on Axis of Symmetry

Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

EXAMPLE:

CYAX	27	152	THRU	160	192	11			

FIELDS:

Field	Contents
Gi	A list of grid points on the axis of symmetry. (Integer > 0 or Character “THRU”)

REMARKS:

1. The displacement coordinate system (see CD field on theGRID entry) for a grid point lying on the axis of symmetry must be a rectangular system with the z-component of motion aligned with the axis of symmetry. The positive axis of symmetry is defined so that the azimuthal angle from positive side 1 to side 2 of a segment is in the same direction as the angle from T1 to T2 for the axis point. This is consistent with the left- or right-hand rule.

2. If the dihedral symmetry option (STYPE = “DIH” on the CYSYM entry) is selected, the y-axis must be perpendicular to side 1.

3. Grid points lying on the axis of symmetry may be constrained by SPCs but not by MPCs. If the number of segments is three or more, SPCs must be applied to both components 1 and 2 or to neither, and SPCs must be applied to both components 4 and 5 or to neither in order to satisfy symmetry. In addition, the degrees-of-freedom (not constrained by SPCs, if any) at these grid points must be in the analysis set (a-set). If all degrees-of-freedom of grid points on the axis of symmetry are constrained by SPCs (including heat transfer, where there is only one degree-of-freedom), the grid point should not be listed on the CYAX entry.

4. Grid points lying on the axis of symmetry must not be defined on side 1 or on side 2 by means of a CYJOIN entry.

5. The word “THRU” must not appear in fields 2 or 9.

CYCADD

Combines cyclic symmetry sets in SOL 401

Combines cyclic symmetry sets in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	S7	-etc-					

EXAMPLE:

CYCADD	12	1	3	6	5	9			

FIELDS:

Field	Contents
SID	Cyclic set identification number. (Integer>0)
Si	Identification numbers of cyclic sets defined with CYCSET entries. (Integer>0)

REMARKS:

1. Multiple cyclic sets (CYCSETs) with unique SIDs can be combined by including their SIDs on a CYCADD entry. The CYCADD entry has its own unique CYCSID which is used on the CYCSET case control command. Multiple cyclic sets with their own CYCSID are a modeling conveience. A single CYCSET Bulk data can be defined with multiple pairs or pairs can be defined in their own CYCSET and combined with the CYCADD.

2. CYCSID's must be unique and may not be the identification of the CYCADD entry.

3. The value defined in the NSEG field on all CYCSET entries referenced by a CYCADD entry must be the same. A fatal error will occur if any are inconsistent.

CYCAXIS

Default cylindrical coordinate system for a SOL 401 cyclic model

Default cylindrical coordinate system for a SOL 401 cyclic model

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYCAXIS	CSID								

EXAMPLE:

CYCAXIS	101								

FIELDS:

Field	Contents
CSID	Identification number of the default cylindrical coordinate system for a SOL 401 cyclic model (Integer≥0).

REMARKS:

1. The CYCAXIS bulk entry is required to define the default cylindrical coordinate system for the automatic coupling selected with the CYCSET case control. The origin of this cylindrical system must be at the center of the revolution, and the z-axis must be consistent with the axial direction.

2. The z-axis of every cylindrical coordinate system referenced by the CYCSET entry must have the same origin and direction as the z-axis of the default coordinate system selected on the CYCAXIS bulk entry.

CYCSET

Defines pairs for cyclic symmetry in SOL 401

Defines pairs for cyclic symmetry in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYCSET	CYSID	NSEG	CSID						
	SID1	TID1	SDIST1						
	SID2	TID2	SDIST2						
	-etc-								

EXAMPLE:

CYCSET	4	11	6						
	1	2							
	4	3							

FIELDS:

Field	Contents
CYSID	CYCSET set identification number. (Integer>0)
NSEG	Number of segments (Integer>1)
CSID	Identification number of a cylindrical coordinate system. (Integer>0 or blank; Default = CSYS selected with the CYCAXIS entry)
NSEG	Number of segments. (Integer>1)
SIDi	Source region identification number for CYCSET pair i. (Integer>0)
TIDi	Target region identification number for CYCSET pair i. (Integer>0)
SDISTi	Search distance for CYCSET pairs (Real); (Default=1.0e-3)

REMARKS:

1. The SIDi and TIDi fields select the BSURFS and BCPROPS bulk entries. These entries define regions on the faces of the CHEXA, CPENTA, CPYRAM, and CTETRA elements.

2. The source region selected in a pair must have a smaller positive theta location than the target region.

3. To do the automatic coupling, NX Nastran internally rotates the target region grids into the source region grids, it does a mesh refinement on both the source and target, and then creates MPC equations using the target as the dependent DOF and the source as the independent DOF. The MPC equations are created between any source and target region grids within the user defined search distance (SDISTi) using a weighted area method. From each source grid, the search occurs in both the positive and negative theta DOF direction. The SDIST field is required for each pair of regions.

4. The software will use the number of segments (NSEG) field to compute the angle between the source and target faces. For example, if a 30 degree segment is modeled, NSEG would be 12 (=360/30).

5. The z-axis of the cylindrical coordinate system selected in the CSYS field must have the same origin and direction as the z-axis of the default coordinate system selected with the CYCAXIS bulk entry.

CYJOIN

Cyclic Symmetry Boundary Points

Defines the boundary points of a segment in cyclic symmetry problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYJOIN	SIDE	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

EXAMPLE:

CYJOIN	1	T2	7	9	16	THRU	33	64	
	72	THRU	89						

FIELDS:

Field	Contents
SIDE	Side identification. (Integer 1 or 2)
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems. See Remark 3 . (Character: “T1”, “T2”, “T3”, “R”, “C”, “S”)
Gi	Grid or scalar point identification numbers. (Integer > 0 or Character “THRU”)

REMARKS:

1. CYJOIN entries are used only for cyclic symmetry problems. The CYSYM entry must be used to specify rotational, dihedral, or axisymmetry.

2. For rotational or axisymmetry problems, there must be one logical entry for SIDE = 1 and one for SIDE = 2. The two lists specify grid points to be connected; therefore, both lists must have the same length.

3. For dihedral problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. For dihedral and/or AXI type of symmetry, the grid point degree-of-freedom that is normal to the boundary must be specified in field 3 as “T1”, “T2”, or “T3”. (“R”, rectangular, and “C”, cylindrical, are the same as “T2”while “S”, spherical, is the same as “T3”). For scalar and extra points with one degree-of-freedom, these should be specified as blank, “T2”, or “T3” if they are to have the same sign, and “T1”, if the two connected points are to be opposite in sign.

4. All components of displacement at boundary points are connected to adjacent segments except those constrained by SPCi, MPC, or OMITi entries.

5. The points on the axis of symmetry of the model, defined in the CYAX entry must not be defined as a side 1 or side 2 point by means of this entry.

6. The word “THRU” may not appear in fields 4 or 9 of the parent entry and fields 2 or 9 of the continuation entries.

7. All grid points that are implicitly or explicitly referenced must be defined.

8. For rotational and axisymmetry problems, the displacement coordinate systems must be consistent between sides 1 and 2. This is best satisfied by the use of a spherical or cylindrical coordinate system.

CYSUP

Fictitious Supports for Cyclic Symmetry

Defines fictitious supports for cyclic symmetry analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

EXAMPLE:

CYSUP	16	1245							

FIELDS:

Field	Contents
GID	Grid point identification number. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

REMARKS:

1. Components of motion defined on this entry may not appear on SPC, SPC1, OMIT, OMIT1 entries, or in rigid elements or multipoint constraints as dependent degrees-of-freedom.

2. Supports are applied at the grid point identified in field 2 to prevent rigid body motions in static analysis, or to define rigid body modes in eigenvalue analysis. All degrees-of-freedom should be at a single grid point. In other words, there can only be one such supported grid point in the model. The supports are applied only to the cyclic components of order k=0 or k=1. In order to satisfy conditions of

symmetry, certain restrictions are placed on the location of the grid point and the orientation of its displacement coordinate system, as shown in the following table:

Symmetry Option (STYPE on the CYSYM entry)	ROT	ROT	DIH	DIH	DIH
Number of Segments, N	2	≥3	1	2	≥3
Location of Grid Point	See Note C.	See Note D.	Side 1	Side 1	Side 1
Special Restrictions on Displacement Coordinate System	See Notes A and E.	See Note B.	None	See Note A.	See Note B.

Note

a. T3 axis must be parallel to axis of symmetry.

b. The displacement coordinate system at the referenced grid point must be cylindrical with the z-axis along the axis of symmetry.

c. Any location except on side 2.

d. Any location except on the axis of symmetry or on side 2.

e. If the grid point is on the axis of symmetry, the displacement coordinate system must be rectangular.

3. If the number of segments, N, is 1 (in the case of DIH symmetry) or 2 (in the case of ROT or AXI symmetry), it is important that the rotational components referenced in field 3 be elastically connected to the structure. If N ≥ 2 (in the case of DIH symmetry) or N ≥ 3 (in the case of ROT or AXI symmetry), it is not important, because in this case the supports for rigid body rotation are actually applied to translational motions.

4. if N ≥ 3, supports will be applied to both the 1 and 2 (inplane-translational) components, if either is referenced, and to both the 4 and 5 (out-of-plane rotational) components, if either is referenced. If component 6 is supported, component 2 should not appear on OMIT or OMIT1 entries.

5. The restrictions noted in Remarks 2 and 4 are related to symmetry requirements. For N ≥ 3, symmetry requires that the supports be symmetrical (or antisymmetrical), with respect to any plane passing through the axis of symmetry. For the DIH options, N = 1 and N = 2, symmetry requires that the supports be symmetrical (or antisymmetrical) with respect to the plane(s) of symmetry. For the ROT option, N = 2, symmetry requires that a support be either parallel or perpendicular to the axis of symmetry.

6. GID must be a grid point and not a scalar point.

CYSYM

Cyclic Symmetry Parameters

Defines parameters for cyclic symmetry analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

EXAMPLE:

CYSYM	6	ROT							

FIELDS:

Field	Contents
NSEG	Number of segments. (Integer > 0)
STYPE	Symmetry type. (Character: “ROT”, “DIH”, or “AXI”)

REMARKS:

1. STYPE = “AXI” is a special case of STYPE = “ROT” used to model axisymmetric structures.
2. If STYPE = “AXI”, then all grid points must lie on side 1, side 2, or the axis. Also, plate elements with midside grid points may not be defined. See “Cyclic Symmetry” in the *NX Nastran User’s Guide*.

Chapter 13: Bulk Data Entries D—E

DAREA

Load Scale Factor

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

EXAMPLE:

DAREA	3	6	2	8.2	15	1	10.1		

FIELDS:

Field	Contents
SID	Identification number. (Integer>0)
Pi	Grid, extra, or scalar point identification number. (Integer>0)
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point) See Remark 3.
Ai	Scale factor. (Real)

REMARKS:

1. One or two scale factors may be defined on a single entry.
2. Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.

3. Component numbers refer to the displacement coordinate system defined with the CD field on the GRID entry.

4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.

5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.

6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE/MOMENT/SLOAD entries (as appropriate) if there are no LSEQ Bulk Data entries. The DAREA entries can only be used in static analysis if there are no LSEQ Bulk Data entries.

7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements if there are no LSEQ Bulk Data entries.

DCONADD

Design Constraint Set Combination

Defines the design constraints for a subcase as a union of DCONSTR entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DCONADD	DCID	DC1	DC2	DC3	DC4	DC5	DC6	DC7	
	DC8	-etc.-							

EXAMPLE:

DCONADD	10	4	12						

FIELDS:

Field	Contents
DCID	Design constraint set identification number. (Integer>0)
DCi	DCONSTR entry identification number. (Integer>0)

REMARKS:

1. The DCONADD entry is selected by a DESSUB or DESGLB Case Control command.
2. All DCi must be unique from other DCi.

DCONSTR

Design Constraints

Defines design constraints.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW or LTID	UALLOW or UTID	LOWFQ	HIGHFQ			

EXAMPLES:

DCONSTR	10	4	1.25						

DCONSTR	300	101	–0.5	5.53					
DCONSTR	300	201	11	12					
DCONSTR	300	207	11	5.53					

FIELDS:

Field	Contents
DCID	Design constraint set identification number. (Integer>0)
RID	DRESPi entry identification number. (Integer>0)
LALLOW or LTID	Lower bound on the response quantity (Real) or TABLEDi ID (Integer) (Default = -1.0E20). See Remarks 4 and 10. See the remarks on the DRESP1 Bulk Data entry.
UALLOW or UTID	Upper bound on the response quantity (Real) or TABLEDi ID (Integer) (Default = 1.0E20). See Remark 4. See the remarks on the DRESP1 Bulk Data entry.

Field	Contents
LOWFQ	Low end of frequency range in Hz. See Remark 9. (Real ≥ 0.0, Default = 0.0)
HIGHFQ	High end of frequency range in Hz (Real ≥ LOWFQ, Default = 1.0E+20). See Remark 9.

REMARKS:

1. The DCONSTR entry may be selected in the Case Control section by the DESSUB or DESGLB command.

2. DCID may be referenced by the DCONADD Bulk Data entry.

3. For a given DCID, the associated RID can be referenced only once.

4. If LALLOW and/or UALLOW are defined as real values, they are interpreted as constant.

   If defined as integers, they are interpreted as the ID of a TABLED1, TABLED2, TABLED3, or TABLED4 bulk entry which defines the bound as a function of frequency. When spawning multiple constraints over a frequency set from a single DCONSTR input, the program loops over the frequencies in the frequency set to determine the corresponding bound for each frequency.

5. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESPi entry. If RID refers to an “EIGN” response, then the imposed bounds must be expressed in the units of the eigenvalues, $(\text{radian/time})^2$. If RID refers to a “FREQ” response, then the imposed bounds must be expressed in cycles/time.

6. LALLOW and UALLOW are unrelated to the stress limits specified on the MATi entry.

7. For selection of DOT as the optimizer (the default), the constraints are computed as follows:

   g = (LALLOW – r) / GNORM for lower bound constraints

   g = (r – UALLOW) / GNORM for upper bound constraints

   where

$$\text{GNORM} = \begin{cases} |\text{LALLOW}| \text{ for lower bounds if } |\text{LALLOW}| > \text{GSCAL} \\ |\text{UALLOW}| \text{ for upper bounds if } |\text{UALLOW}| > \text{GSCAL} \\ \text{GSCAL otherwise} \end{cases}$$

GSCAL is specified on the DOPTPRM entry. (Default = 0.001)

If the alternate optimizer Siemens Design Optimization (SDO) is selected by setting system(425)=1, then a true normalization is performed, and the normalized constraint values always vary between -1.0 and +1.0 in a symmetric manner. Due to the method of normalization employed with the use of this optimizer, there is no need to account for a small denominator. Note also that the “Maximum Value of Constraint” column in the “Summary of Design Cycle History” table in the f06 file will also contain different values based on which optimizer is selected. The normalization for the Analytic Design System follows the below expressions:

If a bound is sandwiched between zero and r:

g=(LALLOW–r) / |r| or g=(r–UALLOW) / |r|

Otherwise:

g=(LALLOW–r) / |2*LALLOW–r| or g=(r–UALLOW) / |2*UALLOW–r|

The only special case gives a constraint value of 0.0, thus no difficulties arise with a zero denominator.

8. As Remark 7 indicates, when DOT is selected as the optimizer, small values of UALLOW and LALLOW require special processing and should be avoided. Bounds of exactly zero are particularly troublesome. This can be avoided by using a DRESP2 entry that offsets the constrained response from zero.

9. LOWFQ and HIGHFQ fields are functional only for DRESP1 response types with the “FR” or “PSD” prefixes, for example FRDISP or PSDVELO. The bounds provided in LALLOW and UALLOW are applicable to a response only when the value of the response of the forcing frequency falls between the LOWFQ and HIGHFQ. If the ATTB field of the DRESP1 entry is not blank, LOWFQ and HIGHFQ are ignored.

10. For responses where 0.0 is a natural lower bound that can never be violated, such as in the case of the von Mises equivalent stress, it is best to leave the lower bound to the default. The reason for this is that a lower bound of 0.0, when satisfied, does not add to solution value and can mislead the algorithm. For example, consider a feasible design where the maximum von Mises equivalent stress value is at 80% of the upper bound, but some elements are stressed very little. If a lower bound of 0.0 is prescribed, the stress values will then be very close to the constraint surfaces for these latter elements, and the constraint values will come out as far more critical (closer to zero) than for the actually stressed

elements for which the constraint values are negative because the stresses are in the feasible region.

DDVAL

Discrete Design Variable Values

Define real, discrete design variable values for discrete variable optimization.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DVAL1	DVAL2	DVAL3	DVAL4	DVAL5	DVAL6	DVAL7	

ALTERNATE FORMAT:

DDVAL	ID	DVAL1	“THRU”	DVAL	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

CONTINUATION ENTRY FORMAT 1:

	DVAL8	DVAL9	DVAL10	DVAL11	-etc.-				

CONTINUATION ENTRY FORMAT 2::

	DVAL8	“THRU”	DVAL9	“BY”	INC				

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
DDVAL	110	0.1	0.2	0.3	0.5	0.6	0.4		
	07	thru	1.0	by	0.05				
	1.5	2.0							

DDVAL

FIELDS:

Field	Contents
ID	Unique discrete value set identification number. (Integer>0)
DVALi	Discrete values. (Real, or “THRU” or “BY”)
INC	Discrete value increment. (Real)

REMARKS:

1. DDVAL entries must be referenced by a DESVAR entry in the DDVAL field (field 8).

2. Trailing fields on a DDVAL record can be left blank if the next record is of type DVALi “THRU” DVALj “BY” INC. Also fields 7 - 9 must be blank when the type DVALi “THRU” DVALj “BY” INC is used in fields 2 - 6 and fields 8 - 9 must be blank when the type DVALi “THRU” DVALj “BY” INC is used in fields 3 - 7 for the first record. Embedded blanks are not permitted in other cases.

3. The DVALi sequence can be random.

4. The format DVALi “THRU” DVALj “BY” INC defines a list of discrete values, e.g., DVALi, DVALi+INC, DVALIi+2.0*INC, ..., DVALj. The last discrete DVALj is always included, even if the range is not evenly divisible by INC.

DEFORM

Static Element Deformation

Defines enforced axial deformation for one-dimensional elements for use in statics problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

EXAMPLE:

DEFORM	1	535	.05	536	-.10				

FIELDS:

Field	Contents
SID	Deformation set identification number. (Integer>0)
EIDi	Element number. (Integer>0)
Di	Deformation. (Real; positive value represents elongation.)

REMARKS:

1. The referenced element must be one-dimensional (CROD, CONROD, CTUBE, CBAR, CBEAM).
2. Deformation sets must be selected in the Case Control Section with DEFORM = SID.
3. One to three enforced element deformations may be defined on a single entry.

4. The DEFORM entry, when called by the DEFORM Case Control command, is applicable to linear static, inertia relief, differential stiffness, and buckling (Solutions 101, 105, 114, and 200) and will produce fatal messages in other solution sequences. Use SPCD to apply enforced displacements in solution sequences for which DEFORM does not apply.

DEFUSET

Degree-of-Freedom Set Name Definition

Defines new names for degree-of-freedom sets.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DEFUSET	OLD1	NEW1	OLD2	NEW2	OLD3	NEW3	OLD4	NEW4	

EXAMPLE:

DEFUSET	U2	X	U4	Y	U3	Z			

FIELDS:

Field	Contents
OLDi	Default set name. (One to four characters)
NEWi	New set name. (One to four characters)

REMARKS:

1. From one to four set names may be specified on a single entry.
2. OLDi must refer to any of the set names given in “Degree-of-Freedom Sets”. It is recommended that OLDi refer only to the set names U1 through U6. If sets PA or PS are referenced, a user fatal message is issued.
3. All NEWi names must be unique with respect to all other set names.
4. The DEFUSET entry is optional since default set names exist for all displacement sets.

5. The DEFUSET entry must be present in the Bulk Data Section in all restarts.

6. The user defined degree-of-freedom sets U1 – U9 are created with the USET and USET1 bulk entries. Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases. See “User Defined Degree-of-Freedom Sets” for the list of these special cases.

DELAY

Dynamic Load Time Delay

Defines the time delay term τ in the equations of the dynamic loading function.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	T1	P2	C2	T2		

EXAMPLE:

DELAY	5	21	6	4.25	7	6	8.1		

FIELDS:

Field	Contents
SID	Identification number of the DELAY entry. (Integer>0)
Pi	Grid, extra, or scalar point identification number. (Integer>0)
Ci	Component number. (Integer 1 through 6 for grid point, blank or 0 for extra point or scalar point.)
Ti	Time delay τ for designated point Pi and component Ci. (Real)

REMARKS:

1. One or two dynamic load time delays may be defined on a single entry.
2. SID must also be specified on a RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entry. See those entry descriptions for the formulas that define the manner in which the time delay τ is used.

3. A DAREA and/or LSEQ entry should be used to define a load at Pi and Ci.

4. In superelement analysis, DELAY entries may only be applied to loads on points in the residual structure.

DEQATN

Design Equation Definition

Defines one or more equations for use in design sensitivity analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DEQATN	EQID	EQUATION							
	EQUATION (Cont.)								

EXAMPLE:

DEQATN	14	F1 (A, B, C, D, R) = A + B · C − (D ** 3 + 10.0) + sin (PI (1 · R))	
	+ A**2/ (B − C) ; F = A + B − F1 · D		

FIELDS:

Field	Contents
EQID	Unique equation identification number. (Integer>0)
EQUATION	Equation(s). See Remarks. (Character)

REMARKS:

1. EQUATION is a single equation or a set of nested equations and is specified in fields 3 through 9 on the first entry and may be continued on fields 2 through 9 on the continuation entries. On the continuation entries, no commas can appear in columns 1 through 8. All data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

   A single equation has the following format:

   variable-1 (x1, x2, ..., xn)=expression-1

A set of nested equations is separated by semicolons and has the format:

variable-1 (x1, x2, ..., xn)=expression-1;
variable-2=expression-2;
variable-3=expression-3;
etc.
variable-m=expression-m

Expression-i is a collection of constants, real variables, and real functions, separated by operators, and must produce a single real value. (x1, x2, ..., xn) is the list of all the variable names (except variable-i) that appear in all expressions. Variable-i may be used in subsequent expressions. The last equation, variable-m=expression-m, provides the value that is returned to the Bulk Data entry that references EQID; e.g., DRESP2. The example above represents the following mathematical equations:

$$F1 = A + B \cdot C - (D^3 + 10) + \sin(\mathrm{PI}(1 \cdot R)) + \frac{A^2}{B - C}$$

$$F = A + B + F1 \cdot D$$

where SIN and PI are intrinsic functions. See Remark 4 .

2. EQUATION may contain embedded blanks. EQUATION must contain less than 32,000 nonblank characters. If more characters are required for use with a DRESP2 entry, the DRESP2 can be divided into two or more DRESP2 entries with a master DRESP2 referencing subsequent DRESP2s.

3. The syntax of the expressions follows FORTRAN language standards. The allowable arithmetic operations are shown in Table 13-1 in the order of execution precedence. Parenthesis are used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses.

Table 13-1. DEQATN Entry Operators

Operator	Operation	Sample Expressions	Interpreted As
-, +	Negative or Positive immediately preceded by exponentiation	X * * -Y	X * * (-Y)
* *	Exponentiation	-X * * Y	(-X * * Y)
-, +	Negative or Positive	-X-Y	(-X)-Y

Table 13-1. DEQATN Entry Operators

Operator	Operation	Sample Expressions	Interpreted As
*, /	Multiplication or Division	X * Y-Z	(X * Y)-Z
+, -	Addition or Subtraction	X+Y	X+Y

4. The expressions may contain intrinsic functions. Table 13-2 contains the format and descriptions of functions that may appear in the expressions. The use of functions that may be discontinuous must be used with caution because they can cause discontinuous derivatives. These are ABS, DIM, MAX, MIN, and MOD. For examples and further details see the *NX Nastran DMAP Programmer's Guide*.

Table 13-2. DEQATN Entry Functions

Format	Description	Mathematical Expressions
ABS(x)	absolute value	|x|
ACOS(x)	arccosine	$\cos^{-1} x$
ACOSH(x)	hyperbolic arccosine	$\cosh^{-1} x$
ASIN(x)	arcsine	$\sin^{-1} x$
ASINH(x)	hyperbolic arcsine	$\sinh^{-1} x$
ATAN(x)	arctangent	$\tan^{-1} x$
ATAN2(x,y)	arctangent of quotient	$\tan^{-1} (x/y)$
ATANH(x)	hyperbolic arctangent	$\tanh^{-1} x$
ATANH2(x,y)	hyperbolic arctangent of quotient	$\tanh^{-1} (x/y)$
AVG(X_1, X_2, .., Xn)	average	$\frac{1}{n}\sum_{i=1}^{n} X_i$

Table 13-2. DEQATN Entry Functions

Format	Description	Mathematical Expressions
COS(x)	cosine	cos x
COSH(x)	hyperbolic cosine	cosh x
DB(P, PREF)	sound pressure in decibel	$20.0 \cdot \log\left(\frac{P}{PREF}\right)$
DBA(P, PREF, F)	sound pressure in decibel (perceived)	$20.0 \cdot \log\left(\frac{P}{PREF}\right) + 10.0 \cdot \log(Ta1) + 10.0 \cdot \log(Ta2)$
DIM(x,y)	positive difference	x-MIN(x,y)
EXP(x)	exponential	e^x
INVDB(DB, PREF)	inverse Db	$10^{\left(\frac{DB}{20.0} + \log PREF\right)}$
INVDBA(DBA, PREF, F)	inverse Dba	$10^{\left(\frac{DBA - 10.0 \cdot \log(Ta1) - 10.0 \cdot \log(Ta2)}{20.0}\right)}$
LOG(x)	natural logarithm	$\log_e x$
LOG10(x)	common logarithm	$\log_{10} x$
LOGX(x,y)	base x logarithm	$\log_x y$
MAX(x_1, x_2, ..)	maximum	maximum of x_1, etc.
MIN(x_1, x_2, ...)	minimum	minimum of x_1, etc.
MOD(x,y)	remainder (modulo)	x − y · (INT (x / y))
PI(x)	multiples of pi (π)	x · π

Table 13-2. DEQATN Entry Functions

Format	Description	Mathematical Expressions
RSS(X_1, ..., Xn)	square root of sum of squares	$\sqrt{\sum_{i=1}^{n} X_i^2}$
SIN(x)	sine	sin x
SINH(x)	hyperbolic sine	sinh x
SQRT(x)	square root	$\sqrt{x}$
SSQ(X_1, ..., Xn)	sum of squares	$\sum_{i=1}^{n} X_i^2$
SUM(X_1, ..., Xn)	summation	$\sum_{i=1}^{n} X_i$
TAN(x)	tangent	tan x
TANH(x)	hyperbolic tangent	tanh x

$X_1, X_2, .., Xn, P$ = structure responses or acoustic pressure

PREF = reference pressure

F = forcing frequency

DB = acoustic pressure in Decibel

DBA = perceived acoustic pressure in Decibel

$$T_a1 = \frac{K3 \cdot F^4}{(F^2 + P2^2)(F^2 + P3^2)}$$

$$T_a2 = \frac{K1 \cdot F^4}{(F^2 + P1^2)^2 (F^2 + P4^2)^2}$$

$K1$	=	2.242882e+16
$K3$	=	1.562339
$P1$	=	20.598997
$P2$	=	107.65265
$P3$	=	737.86223
$P4$	=	12194.22

5. If the DEQATN entry is referenced by the:
   - DVPREL2 entry, then X_i represents the DVIDj and LABLk fields.
   - DRESP2 entry, then X_i represents the DVIDj, LABLk, NRm, Gp, DPIPq, DCICr, DMIMs, DPI2Pt, DCI2Cu, DMI2Mv, and NRRw fields in that order.
6. The DMAP logical operators NOT, AND, OR, XOR, and XQV cannot be used as X_i names.
7. Input errors on the DEQATN entry often result in poor messages. Substituting a “[” for a parenthesis or violating the restriction against large field format are examples. Known messages are UFM 215, SFM 233 and UFM 5199. If any of these messages are encountered then, review the DEQATN entry input.
8. Intrinsic functions MAX and MIN are limited to <100 arguments. If more arguments are desired, the functions may be concatenated.
9. Arithmetic is carried out using the type of the input data. For example, in the expression:

   X = *A***(1/2)

   both values in the exponent are integers so that the value returned for the exponent is calculated using integer arithmetic or 1/2 = 0. In this case 1/2 should be replaced by (.5).

DESVAR

Design Variable

Defines a design variable for design optimization.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	DELXV	DDVAL		

EXAMPLE:

DESVAR	2	BARA1	35.0	10.	100.	0.2			

FIELDS:

Field	Contents
ID	Unique design variable identification number. (Integer>0)
LABEL	User-supplied name for printing purposes. (Character. See Bulk Data Syntax Rules.)
XINIT	Initial value. (Real, XLB ≤ XINIT ≤ XUB)
XLB	Lower bound. (Real, Default = -1.0E+20)
XUB	Upper bound. (Real, Default = +1.0E+20)
DELXV	Fractional change allowed for the design variable during approximate optimization. (Real>0.0; for Default see Remark 2 .)
DDVAL	ID of a DDVAL entry that provides a set of allowable discrete values. (Blank or Integer>0; Default=blank for continuous design variables. See Remark 3 .)

REMARKS:

1. DELXV can be used to control the change in the design variable during one optimization cycle.

2. If DELXV is blank, the default is taken from the specification of the DELX parameter on the DOPTPRM entry. If DELX is not specified, then the default is 1.0.

3. If the design variable is to be discrete (Integer>0 in DDVAL field), and if either of the XLB and/or XUB bounds are wider than those given by the discrete list of values on the corresponding DDVAL entry, XLB and/or XUB will be replaced by the minimum and maximum discrete values.

4. See also bulk data input for DVCREL1, DVCREL2, DVGEOM, DVGRID, DVMREL1, DVMREL2, DVPREL1, DVPREL2 in regard to associating the design variables with structural quantities.

5. When associating design variables with structural quantities by way of the bulk data listed in Remark 4, it is useful to scale the design variables so as to make all design variables the same order of magnitude as far as possible, for purposes of healthier optimization. For example, Young's modulus and material density values are usually orders of magnitude apart, and if design variables were associated with each of these, the derivative for the first may end up being numerically negligible as compared to the second. By scaling the design variables properly, you can ensure that the effect of a given design variable does not get disregarded during optimization.

DIVERG

Divergence Analysis Data

Defines Mach numbers (m) for a divergence analysis in SOLs 144 and 200.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DIVERG	SID	NROOT	M1	M2	M3	M4	M5	M6	
	M7	M8	-etc.-						

EXAMPLE:

DIVERG	70	2	.5	.8	.9				

FIELDS:

Field	Contents
SID	Unique set identifier. (Integer>0)
NROOT	Number of divergence roots that are to be output and their eigenvectors printed. (Integer; Default = 1)
Mi	Mach number. (Real ≥ 0.0)

REMARKS:

1. The DIVERG entry is referenced in Case Control by “DIVERG = SID”.
2. The NROOT lowest divergence dynamic pressures are printed. If there are fewer than NROOT pressures, all available dynamic pressures are printed.
3. Mi values must be distinct.

4. A blank Mach number field terminates the input.

DLINK

Multiple Design Variable Linking

Relates one design variable to one or more other design variables.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DLINK	ID	DDVID	C0	CMULT	IDV1	C1	IDV2	C2	
	IDV3	C3	-etc.-						

EXAMPLE:

DLINK	10	2	0.1	0.33	2	2.0	6	-1.0	
	8	7.0							

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
DDVID	Dependent design variable identification number. (Integer>0)
C0	Constant term. (Real; Default = 0.0)
CMULT	Constant multiplier. (Real; Default = 1.0)
IDVi	Independent design variable identification number. (Integer>0)
Ci	Coefficient i corresponding to IDVi. (Real)

REMARKS:

1. DLINK defines the relationship

$$\mathrm{DDVID} = C0 + \mathrm{CMULT}\sum_{i} \mathrm{Ci} \cdot \mathrm{IDVi}$$

2. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.

3. CMULT provides a simple means of scaling the Ci. For example if Ci = 1/7, 2/7, 4/7, etc. is desired, then CMULT = 1/7 and Ci = 1, 2, 4, etc., may be input.

4. An independent IDVi must not occur on the same DLINK entry more than once.

5. ID is for user reference only.

6. If a design variable is specified as dependent on one DLINK entry, then it cannot be specified as independent on another DLINK entry.

DLOAD

Dynamic Load Combination or Superposition

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-	*					

EXAMPLE:

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	
	-2.0	9							

FIELDS:

Field	Contents
SID	Load set identification number. (Integer>0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries. (Integer>0)

REMARKS:

1. Dynamic load sets must be selected in the Case Control section with DLOAD = SID.

2. The load vector being defined by this entry is given by

$$\{P\} = S\sum_{i} Si\{P_i\}$$

3. Each Li must be unique from any other Li on the same entry.

4. SID must be unique from all TLOADi and RLOADi entries.

5. Nonlinear transient load sets (NOLINi entries) may not be specified on DLOAD entries. NOLINi entries are selected separately in the Case Control section by the NONLINEAR command.

6. A DLOAD entry may not reference a set identification number defined by another DLOAD entry.

7. TLOAD1 and TLOAD2 loads may be combined only through the use of the DLOAD entry.

8. RLOAD1 and RLOAD2 loads may be combined only through the use of the DLOAD entry.

9. In SOL 401, when RFORCE or RFORCE1 entries are referenced by the EXCITEID field on a TLOAD1 entry, the data on the associated TABLEDi, along with the scale factors S and Si on a DLOAD entry (if defined), scale the angular velocity (ω) and acceleration (α), which are used to compute an inertia force in the equation F = [m] [ω x (ω x r)) + α x r]. Since ω is squared in the force computation, the resulting scaling is not linearly related to the computed force (F). All other solutions scale the computed force (F).

REMARKS RELATED TO SOLS 601 AND 701:

1. DLOAD may be used in a nonlinear static or transient analysis to combine time-dependent loads.

2. Li can only be the load set identification numbers of TLOAD1 entries.

3. Both LOAD and DLOAD can be used in the same analysis to define some loads with constant magnitude and some time-dependent loads.

DMI

Direct Matrix Input

Defines matrix data blocks. Generates a matrix of the following form:

$$[\text{NAME}] = \begin{bmatrix} X_{11} & X_{12} & \cdots & X_{1n} \\ X_{21} & X_{22} & \cdots & X_{2n} \\ \cdot & \cdot & \cdot & \cdot \\ X_{m1} & \cdots & \cdots & X_{mn} \end{bmatrix}$$

where the elements X_{ij} may be real ($X_{ij} = A_{ij}$) or complex ($X_{ij} = A_{ij} + iB_{ij}$). The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements. Any unspecified elements of the matrix are taken to be zero.

HEADER ENTRY FORMAT:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	“0”	FORM	TIN	TOUT		m	n	

GENERAL COLUMN ENTRY FORMAT FOR REAL MATRICES:

In this data entry format, you selectively specify elements in a column of a real matrix. Because elements having a value of zero can be excluded, it is useful for defining sparsely populated real matrices.

DMI	NAME	J	I1	A(I1,J)	A(I1+1,J)	A(I1+2,J)	-etc.-	I2	
	A(I2,J)	A(I2+1,J)	A(I2+2,J)	-etc.-					

UNABRIDGED COLUMN ENTRY FORMAT FOR REAL MATRICES:

In this data entry format, you specify all the elements in a column of a real matrix. It is useful for defining highly populated real matrices.

DMI	NAME	J	“1”	A(1,J)	A(2,J)	A(3,J)	A(4,J)	A(5,J)	
	A(6,J)	A(7,J)	-etc.-						

GENERAL COLUMN ENTRY FORMAT FOR COMPLEX MATRICES:

In this data entry format, you selectively specify elements in a column of a complex matrix. Because elements having a value of zero can be excluded, it is useful for defining sparsely populated complex matrices.

DMI	NAME	J	I1	A(I1,J)	B(I1,J)	A(I1+1,J)	B(I1+1,J)	A(I1+2,J)	
	B(I1+2,J)	-etc.-	I2	A(I2,J)	B(I2,J)	A(I2+1,J)	B(I2+1,J)	A(I2+2,J)	
	B(I2+2,J)	-etc.-							

UNABRIDGED COLUMN ENTRY FORMAT FOR COMPLEX MATRICES:

In this data entry format, you specify all the elements in a column of a complex matrix. It is useful for defining highly populated complex matrices.

DMI	NAME	J	“1”	A(1,J)	B(1,J)	A(2,J)	B(2,J)	A(3,J)	
	B(3,J)	A(4,J)	B(4,J)	-etc.-					

GENERAL COLUMN ENTRY FORMAT EXAMPLES:

$$[\mathrm{BBB}] = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

DMI	BBB	0	2	1	1		4	2	
DMI	BBB	1	1	1.0	3.0	5.0			
DMI	BBB	2	2	6.0	4	8.0			

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & 0.0 + 0.0i \\ 3.0 + 0.0i & 6.0 + 7.0i \\ 5.0 + 6.0i & 0.0 + 0.0i \\ 0.0 + 0.0i & 8.0 + 9.0i \end{bmatrix}$$

DMI	QQQ	0	2	3	3		4	2	
DMI	QQQ	1	1	1.0	2.0	3.0	0.0	3	
	5.0	6.0							
DMI	QQQ	2	2	6.0	7.0	4	8.0	9.0	

UNABRIDGED COLUMN ENTRY FORMAT EXAMPLES:

$$[BBB] = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

DMI	BBB	0	2	1	1		4	2	
DMI	BBB	1	1	1.0	3.0	5.0	0.0		
DMI	BBB	2	1	0.0	6.0	0.0	8.0		

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & 0.0 + 0.0i \\ 3.0 + 0.0i & 6.0 + 7.0i \\ 5.0 + 6.0i & 0.0 + 0.0i \\ 0.0 + 0.0i & 8.0 + 9.0i \end{bmatrix}$$

DMI	QQQ	0	2	3	3		4	2	
DMI	QQQ	1	1	1.0	2.0	3.0	0.0	5.0	
	6.0	0.0	0.0						
DMI	QQQ	2	1	0.0	0.0	6.0	7.0	0.0	
	0.0	8.0	9.0						

FIELDS:

Field	Contents
NAME	Name of the matrix. See Remark 1. NAME is used to reference the data block in the DMAP sequence. (One to eight alphanumeric characters, the first of which must be alphabetic.)
FORM	Form of matrix. (Integer) 1 = Square matrix (not symmetric) 2 = General rectangular matrix 3 = Diagonal matrix (elements on the diagonal are stored in a column vector having m rows) 4 = Lower triangular factor 5 = Upper triangular factor 6 = Symmetric matrix 8 = Identity matrix (m = number of rows, n = m)
TIN	Type of matrix being input. (Integer) 1 = Real, single precision (one field used per element) 2 = Real, double precision (one field used per element) 3 = Complex, single precision (two fields used per element) 4 = Complex, double precision (two fields used per element)
TOUT	Type of matrix being output. (Integer) 0 = Set by precision cell 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
m	Total number of rows in the matrix. (Integer>0)

Field	Contents
n	Total number of columns in the matrix. Not required if FORM 3 or 8. (Integer>0)
“0”	Indicates a header entry. (Integer)
J	Specified column number of the matrix. (Integer>0)
“1”	Designates element entry begins with the first row of column J. (Integer)
I1, I2, etc.	Row number in column J where a single element is to be defined or the beginning of a string of elements is to be defined. See Remark 12. (Integer>0)
A(Ix,J)	Real part of the element (see TIN). (Real)
B(Ix,J)	Imaginary part of the element (see TIN). (Real)

REMARKS:

1. In order to use the DMI feature, the user must write a DMAP, or make alterations to a solution sequence that includes the DMIIN module. See the *NX Nastran DMAP Programmer’s Guide*. All rules governing the use of data blocks in DMAP sequences apply.

2. The total number of DMIs and DTIs may not exceed 1000.

3. Field 3 of the header entry must contain an integer of zero (0).

4. For symmetric matrices, the entire matrix must be input.

5. Leading and trailing zeros in a column do not have to be entered. However, a blank field between nonzero fields on this entry is not equivalent to a zero. If a zero input is required, the appropriate type zero must be entered (i.e. 0.0 or 0.0D0).

6. Complex input must have both the real and imaginary parts entered if either part is nonzero (i.e. the zero component must be input explicitly).

7. If A(Ix,J) is followed by “THRU” in the next field and an integer row number “IX” after THRU, then A(Ix,J) will be repeated in each row through IX. The “THRU” must follow an element value. For example, the entries for a real matrix RRR would appear as follows:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	J	I1	A(I1,J)			I1	A(I2,J)	
DMI	RRR	1	2	1.0	THRU	10	12	2.0	

These entries will cause the first column of the matrix RRR to have a zero in row 1, the values 1.0 in rows 2 through 10, a zero in row 11, and 2.0 in row 12.

8. Each column must be a single logical entry. The terms in each column must be specified in increasing row number order.

9. The “FORM” options 4, 5, and 8 are nonstandard forms and may be used only in conjunction with the modules indicated in Table 13-3.

Table 13-3. DMI FORM Options

FORM	Matrix Description	Modules			
		ADD	FBS	MATPRN	MPYAD
4	Lower Triangular Factor		X	X	
5	Upper Triangular Factor		X	X	
8	Identity	X	X	X	X

10. Form 3 matrices are converted to Form 6 matrices, which may be used by any module.

11. Form 7 matrices may not be defined on this entry.

12. I1 must be specified. I2, etc. are not required if their matrix elements follow the preceding element in the next row of the matrix. For example, in the column entry for column 1 of QQQ, neither I2 nor I3 is specified.

13. The DMIG entry is more convenient for matrices with rows and columns that are referenced by grid or scalar point degrees-of-freedom.

DMIG

Direct Matrix Input at Points

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

HEADER ENTRY FORMAT:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	“0"	IFO	TIN	TOUT	POLAR		NCOL	

COLUMN ENTRY FORMAT:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

EXAMPLE:

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
	2	4	2.5+10	0.	50		1.0	0.	

FIELDS:

Field	Contents
NAME	Name of the matrix. See Remark 1 . (One to eight alphanumeric characters, the first of which is alphabetic.)

Field	Contents
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, B2GG, and K42GG Case Control commands. (Integer) 1 = Square 9 or 2 = Rectangular 6 = Symmetric
TIN	Type of matrix being input. (Integer) 1 = Real, single precision (One field is used per element.) 2 = Real, double precision (One field is used per element.) 3 = Complex, single precision (Two fields are used per element.) 4 = Complex, double precision (Two fields are used per element.)
TOUT	Type of matrix that will be created: (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer>0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5. and 6. (Integer>0)
GJ	Grid, scalar or extra point identification number for a column index. (Integer>0)
CJ	Component number for a grid point GJ. (0<Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for a row index. (Integer>0)

Field	Contents
Ci	Component number for Gi for a grid point. (0<CJ ≤ 6 ; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

REMARKS:

1. Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, or B2PP = NAME, or M2PP = NAME for [K_{pp}], [B_{pp}], or [M_{pp}], respectively. Matrices may also be selected for all solution sequences with K2GG = NAME, or B2GG = NAME, or M2GG = NAME, or K42GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.

2. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but input of an element of the matrix more than once will produce a fatal message.

3. Field 3 of the header entry must contain an integer 0.

4. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.

5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term is used for the column index. The CJ term is ignored.

6. If NCOL is not used for rectangular matrices, two different conventions are available:

   - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted

being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

- If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.

7. The matrix names must be unique among all DMIGs.

8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double-field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.

9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.

10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

REMARKS RELATED TO SOL 601:

1. Only IFO=6 is allowed. Hence, NCOL is ignored.

2. Only TIN=1 or 2 is allowed and TOUT is not supported.

3. POLAR is ignored. Only real format specified by Ai is supported. Bi is ignored.

4. Matrices defined on this entry may be selected in the Case Control section by K2GG, M2GG or B2GG.

DMIG,UACCEL

Direct Matrix Input of Enforced Static Acceleration

Defines rigid body accelerations in the basic coordinate system.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	“0"	“9"	TIN				NCOL	
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

EXAMPLE:

DMIG	UACCEL	0	9	1				4	
DMIG	UACCEL	2			2	3	386.4		
DMIG	UACCEL	3			2	4	3.0		
DMIG	UACCEL	4			2	6	1.0		

FIELDS:

Field	Contents
TIN	Type of matrix being input. (Integer 1 or 2) 1 = Real, single precision. (one field is used per element)2 = Real, double precision. (one field is used per element)
NCOL	Number of columns. See Remark 2 . (Integer>0; Default is the number of columns specified)
L	Load sequence number. (Integer>0)
Gi	Grid point identification number of a single reference point. (Integer>0)

Field	Contents
Ci	Component number for Gi in the basic coordinate system. See Remark 4 . (0<Integer ≤ 6)
Xi	Value of enforced acceleration term in the basic coordinate system. (Real)

REMARKS:

1. DMIG,UACCEL is an optional entry when PARAM,INREL,-1 is specified in SOLs 101 or 200. If DMIG,UACCEL is present, the loads applied to the structure are the sum of the conventional applied loads plus the inertia loads resulting from the rigid body accelerations defined on this entry. If it is not present, conventional inertia relief calculations are performed.

2. The load sequence number interpretation depends on the value of the NCOL field. The recommended method is to set it equal to the number of loading conditions. The load sequence number L is then the sequence number of the subcase to which the applied acceleration will be applied.

3. The grid point identification number listed on Gi defines a single grid point on the model where loads will be applied to cause the enforced acceleration state. Gi must also appear on a SUPORT Bulk Data entry and a PARAM,GRDPNT entry. In superelement analysis, it must be a residual structure point exterior to all superelements.

4. The Xi value is the enforced acceleration at grid point Gi. The translation and rotation components are in consistent units and will be applied in the basic coordinate system regardless of the displacement coordinate system specified for Gi (CD field on GRID entry).

5. Only nonzero terms need be entered.

6. See the *NX Nastran User's Guide* for the theoretical basis of inertia relief with superelements.

7. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

DMIJ

Direct Matrix Input at js-Set of the Aerodynamic Mesh

Defines direct input matrices related to collation degrees of freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the interference elements of a CAERO2, use DMIJI or DMI.

HEADER ENTRY FORMAT:

1	2	3	4	5	6	7	8	9	10
DMIJ	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

COLUMN ENTRY FORMAT:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

EXAMPLE:

DMIJ	ALPH1	0	9	2	0			1	
DMIJ	ALPH1	1	1		1	1	.1		
	2	1	.1						

FIELDS:

Field	Contents
NAME	Name of the matrix. See Remark 1 . (One to eight alphanumeric characters, the first of which is alphabetic.)

Field	Contents
IFO	Form of matrix being input. (Integer) 1 = Square 9 or 2 = Rectangular 6 = Symmetric
TIN	Type of matrix being input. (Integer) 1 = Real, single precision (one field s used per element) 2 = Real, double precision (one field used per element) 3 = Complex, single precision (two fields used per element) 4 = Complex, double precision (two fields used per element)
TOUT	Type of matrix being created: (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer>0 indicates magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer>0)
GJ	Grid, scalar or extra point identification number for a column index. (Integer>0)
CJ	Component number for grid point GJ. (0<Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for a row index. (Integer>0)
Ci	Component number for Gi for a grid point. (0<CJ ≤6; blank or zero if Gi is a scalar or extra point.)

Field	Contents
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

REMARKS:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. Also, DMIJ may also be used for the W2GJ and FA2J entries. Again, a single column is required. If both DMI and DMIJ are specified for W2GJ or FA2J, the DMI entry will be used. DMI may NOT be used for AEDW and AEPRESS.

2. The js-set DOFs for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOFs are permanently SPC'd based on the theory's ability to support those degrees of freedom. Unlike the DMIG entry, DMIJ data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data.The j-set DOFs for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X		
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.

5. For symmetric matrices (very rare in the j-set) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower

triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.

6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIJ matrices is always the js-set size—the union of the j-set and the permanently SPC'd partition.) The GJ term is used for the column index. the CJ term is ignored.

7. If NCOL is not used for rectangular matrices, two different conventions are available:

   - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

   - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Training null columns of the js-size matrix will be truncated.

8. The matrix names must be unique among all DMIJ.

9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.

10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJ input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.

11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIJI

Direct Matrix Input at js-Set of the Interference Body

Defines direct input matrices related to collation degrees of freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the slender elements of a CAERO2 or a CAERO1, 3, 4 or 5 use DMIJ or DMI.

HEADER ENTRY FORMAT:

1	2	3	4	5	6	7	8	9	10
DMIJI	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

COLUMN ENTRY FORMAT:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

EXAMPLE:

DMIJI	ALPH1	0	9	2	0			1	
DMIJI	ALPH1	1	1		1	1	.1		
	2	1	.1						

FIELDS:

Field	Contents
NAME	Name of the matrix. See Remark 1 . (One to eight alphanumeric characters, the first of which is alphabetic.)

Field	Contents
IFO	Form of matrix being input. (Integer) 1 = Square 9 or 2 = Rectangular 6 = Symmetric
TIN	Type of matrix being input. (Integer) 1 = Real, single precision (one field is used per element) 2 = Real, double precision (one field is used per element) 3 = Complex, single precision (two fields used per element) 4 = Complex, double precision (two fields used per element)
TOUT	Type of matrix being created. (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer>0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer>0)
GJ	Grid, scalar or extra point identification number for a column index. (Integer>0)
CJ	Component number for grid point GJ. (0<Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for a row index. (Integer>0)
Ci	Component number for Gi for a grid point. (0<CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)

Field	Contents
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

REMARKS:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. DMI may NOT be used for AEDW and AEPRESS.

2. The js-set DOFs for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOFs are permanently SPC'd based on the theory's ability to support those degrees of freedom. Unlike the DMIG entry, DMIJI data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOFs for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.

5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.

6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows

in all **DMIJI** matrices is always the js-set size—the union of the j-set and the permanently SPC'd partition.) The GJ term is used for the column index. The CJ term is ignored.

7. If NCOL is not used for rectangular matrices, two different conventions are available:

   - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

   - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.

8. The matrix names must be unique among all DMIJI.

9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.

10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJI input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.

11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIK

Direct Matrix Input at ks-Set of the Aerodynamic Mesh

Defines direct input matrices related to physical (displacement) degrees of freedom (ks-set) of aerodynamic grid points. These include WKK, WTFACT and input forces associated with AEFORCE entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

HEADER ENTRY FORMAT:

1	2	3	4	5	6	7	8	9	10
DMIK	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

COLUMN ENTRY FORMAT:

DMIK	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

EXAMPLE:

DMIK	ALPH1	0	9	2	0			1	
DMIK	ALPH1	1	1		1	1	1.0		
	2	1	1.0						

FIELDS:

Field	Contents
NAME	Name of the matrix. See Remark 1 . (One to eight alphanumeric characters, the first of which is alphabetic.)

Field	Contents
IFO	Form of matrix being input. (Integer)
	1 = Square
	9 or 2 = Rectangular
	6 = Symmetric
TIN	Type of matrix being input. (Integer)
	1 = Real, single precision (One field is used per element)
	2 = Real, double precision (One field is used per element)
	3 = Complex, single precision (Two fields are used per element)
	4 = Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)
	0 = Set by precision system cell (Default)
	1 = Real, single precision
	2 = Real, double precision
	3 = Complex, single precision
	4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer>0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer>0)
GJ	Grid, scalar or extra point identification number for column index. (Integer>0)
CJ	Component number for grid point GJ. (0<Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer>0)
Ci	Component number for Gi for a grid point. (0<CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)

Field	Contents
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

REMARKS:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. If both DMI and DMIK are specified for WKK or WTFACT, the DMI entry will be used. DMI may NOT be used for AEFORCE.

2. The ks-set DOFs for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOFs are permanently SPC'd based on the theory's ability to support those degrees of freedom. Unlike the DMIG entry, DMIK data will be partitioned to the k-set, not reduced. No warnings are issued about truncated data. The k-set DOFs for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2-Y		X		X	
CAERO2-Z			X		X
CAERO2-ZY		X	X	X	X

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.

5. For symmetric matrices (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may

be mixed, a fatal message will be issued if an element is input both above and below the diagonal.

6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIK matrices is always the ks-set size—the union of the k-set and the permanently SPC'd partition). The GJ term is used for the column index. The CJ term is ignored.

7. If NCOL is not used for rectangular matrices, two different conventions are available:

   - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

   - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Training null columns of the js-size matrix will be truncated.

8. The matrix names must be unique among all DMIK

9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.

10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIK matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIK input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.

11. If any DMIK entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIK entry changes or additions are not recommended on restart.

DOPTPRM

Design Optimization Parameters

Overrides default values of parameters used in design optimization.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DOPTPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

EXAMPLE:

DOPTPRM	IPRINT	5	DESMAX	10					

FIELDS:

Field	Contents
PARAMi	Name of the design optimization parameter. Allowable names are given in Table 13-4. (Character)
VALi	Value of the parameter. (Real or Integer) See Table 13-4.

REMARKS:

1. Only one DOPTPRM entry is allowed in the Bulk Data Section.

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
APRCOD	Approximation method to be used. 1 = Direct Linearization; 2=Mixed Method based on response type; 3 = Convex Linearization. APRCOD = 1 is recommended for shape optimization problems. (Integer 1, 2, or 3; Default = 2)
CONV1	Relative criterion to detect convergence. If the relative change in objective between two optimization cycles is less than CONV1, then optimization is terminated. (Real>0.0; Default = 0.0001)
CONV2	Absolute criterion to detect convergence. If the absolute change in objective between two optimization cycles is less than CONV2, then optimization is terminated. (Real>0.0; Default = 1.0E-20)
CONVDV	Relative convergence criterion on design variables. (Real>0.0; Default = 0.001)
CONVPR	Relative convergence criterion on properties. (Real>0.0; Default = 0.001)
CT	Constraint tolerance. Constraint is considered active if current value is greater than CT. (Real<0.0; Default = -0.03)
CTMIN	Constraint is considered violated if current value is greater than CTMIN.(Real>0.0; Default = 0.003)
DABOBJ*	Maximum absolute change in objective between ITRMOP consecutive iterations (see ITRMOP) to indicate convergence at optimizer level. F0 is the initial objective function value. See Remark 2. (Real>0.0; Default = MAX[0.001*ABS(F0), 0.0001])
DELB	Relative finite difference move parameter. (Real>0.0; Default = 0.0001)
DELOBJ*	Maximum relative change in objective between ITRMOP consecutive iterations to indicate convergence at optimizer level. See Remark 2. (Real>0.0; Default = 0.001)

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
DELP	Fractional change allowed in each property during any optimization design cycle. This provides constraints on property moves. (Real>0.0; Default = 0.2)
DELX	Fractional change allowed in each design variable during any optimization cycle. (Real>0.0; Default = 1.0) See DXMIN for absolute minimum change allowed per cycle for design variables.
DESMAX	Maximum number of design cycles (not including FSD cycle) to be performed. (Integer ≥ 0; Default = 5) While the default for DESMAX is 5 cycles, large problems often require a significantly higher number of cycles. For example, this could be somewhere between 20 and 50, depending on many factors. The most significant factors are the time required for a full analysis pass, the number of design variables, and the nonlinearity of the constraints.
DISBEG	Design cycle ID for discrete variable processing initiation. Discrete variable processing analysis is carried out for every design cycle after DISBEG. (Integer>=0, default = 0=the last design cycle) If fully stressed design (FSD), then discrete variable processing can start only after the last FSD cycle.
DISCOD	Discrete Processing Method: (Integer 1, 2, 3 or 4; Default = 1) 1: Design of Experiments 2: Conservative Discrete Design 3: Rounding up to the nearest design variable 4: Rounded off to the nearest design variable
DOBJ1*	Relative change in objective attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses. See Remark 2. (Real>0.0; Default = 0.1)
DOBJ2*	Absolute change in objective attempted on the first optimization iteration. See Remark 2. (Real>0.0; Default = 0.2*(F0))

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
DPMIN	Minimum move limit imposed. (Real>0.0; Default = 0.01)
DX1*	Maximum relative change in a design variable attempted on the first optimization iteration. Used to estimate the initial move in the one dimensional search. Updated as the optimization progresses. See Remark 2. (Real>0.0; Default = 0.01)
DX2*	Absolute change in a design variable attempted on the first optimization iteration. See Remark 2. (Real>0.0; Default = 0.2*MAX[X(I)])
DXMIN	Minimum absolute limit on design variable move (Real>0.0; Default = 0.05). See DELX for allowable relative change per cycle in design variables.
EDVOUT	Defines what fraction of DVEREL1 generated design variables with increased and decreased values are to be printed into the Design Variable History at the end of the f06 file, as well as at requested cycles. This fraction is applied separately to the increased and to the decreased DVEREL1 generated design variables. For example, when the default of 0.001 is used, both 0.1% of the most increased and 0.1% of the most decreased DVEREL1 generated design variables will be printed. A value of 1.0 requests to print all DVEREL1 generated design variables which have either increased or decreased. (0.0<Real≤1.0; Default = 0.001) Note: The increase or decrease in the value of a DVEREL1 generated design variable is based on the relative change between the “initial” and the “best design” cycles. Note: The design variables which are not generated by the DVEREL1 entry are all printed in full.
FSDALP	Relaxation parameter applied in Fully Stressed Design (Real, 0.0<FSDALP ≤ 1.0, Default = 0.9)
FSDMAX	Specifies the number of Fully Stressed Design Cycles that are to be performed (Integer, Default = 0)

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
GMAX	Maximum constraint violation allowed at the converged optimum. (Real>0.0; Default = 0.005)
GSCAL	Constraint normalization factor. See Remarks under the DSCREEN and DCONSTR entries. (Real>0.0; Default = 0.001)
IGMAX	If IGMAX = 0, only gradients of active and violated constraints are calculated. If IGMAX>0, up to NCOLA gradients are calculated including active, violated, and near active constraints. (Integer≥0; Default = 0)
IPRINT*	Print control during approximate optimization phase. Increasing values represent increasing levels of optimizer information. See Remark 2. (0 ≤ Integer ≤ 7; Default = 0) 0 no output (Default) 1 internal optimization parameters, initial information, and results 2 same as 1, plus objective function and design variables at each iterations 3 same as 2, plus constraint values and identification of critical constraints 4 same as 3, plus gradients 5 same as 4, plus search direction 6 same as 5, plus scaling factors and miscellaneous search information 7 same as 6, plus one dimensional search information
IPRNT1*	If IPRNT1 = 1, print scaling factors for design variable vector. See Remark 2. (Integer 0 or 1; Default = 0)
IPRNT2*	If IPRNT2 = 1, print miscellaneous search information. If IPRNT2 = 2, turn on print during one-dimensional search process. (Warning: This may lead to excessive output.) See Remark 2. (Integer 0, 1, or 2; Default = 0)

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
ISCAL	Design variables are rescaled every ISCAL iterations. Set ISCAL = -1 to turn off scaling. (Integer; Default=NDV (number of design variables))
ITMAX*	Maximum number of iterations allowed at optimizer level during each design cycle. See Remark 2. (Integer; Default = 40)
ITRMOP*	Number of consecutive iterations for which convergence criteria must be satisfied to indicate convergence at the optimizer level. See Remark 2. (Integer; Default = 2)
ITRMST*	Number of consecutive iterations for which convergence criteria must be met at the optimizer level to indicate convergence in the Sequential Linear Programming Method. See Remark 2. (Integer>0; Default = 2)
IWRITE*	FORTRAN unit for print during approximate optimization phase. Default value for IWRITE is set to the FORTRAN unit for standard output. See Remark 2. (Integer>0, Default=6 or value of SYSTEM(2).)
JPRINT*	Sequential Linear Programming subproblem print. If JPRINT>0, IPRINT is turned on during the approximate linear subproblem. See Remark 2. (Default = 0)
JTMAX*	Maximum number of iterations allowed at the optimizer level for the Sequential Linear Programming Method. This is the number of linearized subproblems solved. See Remark 2. (Integer≥0; Default = 20)
JWRITE*	If JWRITE>0, file number on which iteration history will be written. See Remark 2. (Integer>0; Default = 0)

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
METHOD	DOT Optimization Method: (Integer 1, 2, or 3; Default = 1) 1: Modified Method of Feasible Directions. (Default) 2: Sequential Linear Programming 3: Sequential Quadratic Programming For Siemens Analytic Design System (SADS), all selections will automatically revert to a proprietary, linear programming based algorithm. To request the SADS optimizer, system cell 425 in Table 1-1 of the NASTRAN Statement section should be set to 1.
P1	Determines the design cycles in which output is printed. (Integer ≥ 0; Default = 0) Initial results are always printed prior to the first approximate optimization. If an optimization task is performed, final results are always printed for the final analysis unless PARAM,SOFTEXIT,YES is specified. These two sets of print are not controllable. n: Print at every n-th design cycle.
P2	Items to be printed at the design cycles defined by P1 (Integer; Default = 1) 0: No print. 1: Print objective and design variables. (Default) 2: Print properties. 4: Print constraints. 8: Print responses. 16: Print weight as a function of a material ID (note that this is not a design quantity so that only inputs to the approximate design are available). n: Sum of desired items. For example, P2 = 10 means print properties and responses.

Table 13-4. PARAMi Names and Descriptions

Name	Description , Type, and Default Value
P2RSET	ID of a SET1 bulk entry listing constrained responses to be printed if retained (the SET1 lists DRESPi ID numbers). The default is to print all retained constrained responses. Constrained responses are those responses referenced directly or indirectly by a DCONSTR bulk entry. Retained constrained responses are the constrained responses which the software determines to be relevant in a design cycle in terms of being critical in driving the optimization process.
PLVIOL	Flag for handling of property limit violation. By default, the job will terminate with a user fatal message if the property derived from design model (DVPRELi, DVMRELi, DVCRELi) exceeds the property limits. Setting PLVIOL to a non-zero number will cause the program to issue a user warning message by ignoring the property limits violation and proceed with the analysis. (Integer; Default=0)
PTOL	Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry (through DVPRELi relations). PTOL is provided to trap ill-posed design models. (The minimum tolerance may be specified on user parameter DPEPS. See “Parameters and Parameter Applicability Tables”) (Real>0.0; Default = 1.0E+35)
STPSCL	Scaling factor for shape finite difference step sizes, to be applied to all shape design variables. (Real>0.0; Default = 1.0)

2. The parameter names with “*” are supported only by the DOT Optimization Method.

DPHASE

Dynamic Load Phase Lead

Defines the phase lead term θ in the equation of the dynamic loading function.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	TH1	P2	C2	TH2		

EXAMPLE:

DPHASE	4	21	6	2.1	8	6	7.2		

FIELDS:

Field	Contents
SID	Identification number of DPHASE entry. (Integer>0)
Pi	Grid, extra, or scalar point identification number. (Integer>0)
Ci	Component number. (Integers 1 through 6 for grid points; zero or blank for extra or scalar points)
THi	Phase lead θ in degrees. (Real)

REMARKS:

1. One or two dynamic load phase lead terms may be defined on a single entry.
2. SID must be referenced on a RLOADi or ACSRCE entry. Refer to the RLOAD1, RLOAD2, or ACSRCE entry for the formulas that define how the phase lead θ is used.

3. A DAREA and/or LSEQ entry should be used to define a load at Pi and Ci.

4. In superelement analysis, DPHASE entries may only be applied to loads on points in the residual structure.

DRESP1

Design Response Quantities

Defines a set of structural responses that are used for the objective and/or design constraints, or for sensitivity analysis purposes.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	ATT3	ATT4	-etc.-					

DRESP1	1	DX1	STRESS	PROD	2	3		102	
	103								

FIELDS:

Field	Contents
ID	Unique entry identifier. (Integer>0)
LABEL	User-defined label. (Character. See Bulk Data Syntax Rules.)
RTYPE	Response type. See Table 13-5. (Character)
PTYPE	Element flag (PTYPE = “ELEM”), or property entry name, or panel flag for ERP responses (PTYPE = “PANEL” – See Remark 34), or RANDPS ID. Blank for grid point responses. “ELEM” or property name used only with element type responses (stress, strain, force, etc.) to identify the relevant element IDs, or the property type and relevant property IDs. (Character: “ELEM”, “PBAR”, “PSHELL”, “PCOMP”, “PANEL”, etc.). PTYPE = RANDPS ID when RTYPE = “PSDDISP” or “PSDVELO” or “PSDACCL”. See Remarks 6 and 32.
REGION	Region identifier for constraint screening. See Remark 10 for defaults. (Integer>0)

Field	Contents
ATTA	Response attribute. See Table 13-5. See Remark 7. (Integer>0, or Blank if specified)
ATTB	Response attribute. See Table 13-5. (Integer>0 or Real>0.0 or Character or Blank)
ATTi (i=1,2,3…)	List of response attributes. See Table 13-5. (Integer>0, or Blank if specified)

Table 13-5. Design Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer>0, or Blank if specified)	ATTB (Integer>0 or Real>0.0 or Character or Blank)	ATTi (i=1,2,3...) (Integer>0, or Blank if specified)
CEIG	Complex Eigenvalue Mode Number	ALPHA or OMEGA (Character) (Default=ALPHA) See Remark 23.	Blank
CFAILURE See Remark 2.	Failure Criterion Item Code	LAMINA Number (Integer>0; Default = 1). See Remark 3.	PCOMP Property entry PID(s) See Remark 6.
CSTRAIN See Remark 2.	Strain Item Code See Remark 1.	LAMINA Number (Integer>0; Default = 1). See Remark 3.	PCOMP Property entry PID(s) See Remark 6.
CSTRESS See Remark 2.	Stress Item Code See Remark 1.	LAMINA Number (Integer>0; Default = 1). See Remark 3.	PCOMP Property entry PID(s) See Remark 6.
DISP	Displacement Component	Blank	Grid ID(s) See Remark 28.
EIGN See Remark 19.	Normal Modes Mode Number	Approximation Code (Integer 1 or 2) See Remark 20.	Blank
ERP	For ERP magnitude: Blank or 0 For ERP density: 1 See Remark 33.	Frequency Value (Blank or Real>0.0 or Character) See Remarks 16 and 21.	Panel related SET3 IDs

Table 13-5. Design Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer>0, or Blank if specified)	ATTB (Integer>0 or Real>0.0 or Character or Blank)	ATTi (i=1,2,3...) (Integer>0, or Blank if specified)
ESE	Strain Energy Item Code See Remark 22.	Blank	Property entry PID(s) See Remark 6.
FLUTTER	Blank	Blank	See Remark 15.
FORCE	Force Item Code See Remark 1.	Blank	Property entry PID(s) See Remark 6.
FRACCL	Acceleration Component See Remarks 8 and 9.	Frequency Value (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
FRDISP	Displacement Component See Remarks 8 and 9.	Frequency Value (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
FREQ See Remark 19.	Normal Modes Mode Number	Approximation Code (Integer 1 or 2) See Remark 20.	Blank
FRFORC	Force Item Code See Remark 1.	Frequency Value (Blank or Real>0.0 or Character) See Remarks 16 and 21 .	Property entry PID(s) See Remark 6.
FRSPCF	SPC Force Component See Remarks 8 and 9.	Frequency Value (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
FRSTRE	Stress Item Code See Remark 1.	Frequency Value. (Blank or Real>0.0 or Character) See Remarks 16 and 21 .	Property entry PID(s) See Remark 6.

Table 13-5. Design Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer>0, or Blank if specified)	ATTB (Integer>0 or Real>0.0 or Character or Blank)	ATTi (i=1,2,3...) (Integer>0, or Blank if specified)
FRVELO	Velocity Component See Remarks 8 and 9.	Frequency Value. (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
LAMA	Buckling Mode Number	Approximation Code (Integer 1 or 2) See Remark 20.	Blank
PRES	Acoustic Pressure Component	Frequency Value (Real>0.0) See Remark 16.	Grid ID(s) See Remark 28.
PSDACCL See Remarks 27, 29, 30, 31.	Acceleration Component See Remark 8.	Frequency Value. (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
PSDDISP See Remarks 27, 29, 30, 31.	Displacement Component See Remark 8.	Frequency Value. (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
PSDVELO See Remarks 27, 29, 30, 31.	Velocity Component See Remark 8.	Frequency Value. (Blank, Real>0.0 or Character) See Remarks 16 and 21.	Grid ID(s) See Remark 28.
RMSACCL See Remarks 27, 31.	Acceleration Component See Remark 8.	RANDPS ID (Integer>0)	Grid ID(s) See Remark 28.
RMSDISP See Remarks 27, 31.	Displacement Component See Remark 8.	RANDPS ID (Integer>0)	Grid ID(s) See Remark 28.
RMSVELO See Remarks 27, 31.	Velocity Component See Remark 8.	RANDPS ID (Integer>0)	Grid ID(s) See Remark 28.
SPCFORCE	SPC Force Component	Blank	Grid ID(s) See Remark 28.

Table 13-5. Design Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer>0, or Blank if specified)	ATTB (Integer>0 or Real>0.0 or Character or Blank)	ATTi (i=1,2,3...) (Integer>0, or Blank if specified)
STABDER See Remark 14.	AESTAT or AESURF Entry ID	Restraint Flag. (Integer 0 or 1) See Remark 13.	Component
STRAIN	Strain Item Code See Remark 1.	Blank	Property entry PID(s) See Remark 6.
STRESS	Stress Item Code See Remark 1.	Blank	Property entry PID(s) See Remark 6.
TACCL	Acceleration Component	Time Value (Blank, Real>0.0, or Character) See Remarks 17 and 21.	Grid ID(s) See Remark 28.
TDISP	Displacement Component	Time Value (Blank, Real>0.0, or Character) See Remarks 17 and 21.	Grid ID(s) See Remark 28.
TFORC	Force Item Code See Remark 1.	Time Value. (Blank or Real>0.0 or Character) See Remarks 17 and 21 .	Property entry PID(s) See Remark 6.
TRIM	AESTAT or AESURF Entry ID	Blank	Blank
TSPCF	SPC Force Component	Time Value. (Blank, Real>0.0, or Character) See Remarks 17 and 21.	Grid ID(s) See Remark 28.
TSTRE	Stress Item Code See Remark 1.	Time Value. (Blank or Real>0.0 or Character) See Remarks 17 and 21 .	Property entry PID(s) See Remark 6.

Table 13-5. Design Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer>0, or Blank if specified)	ATTB (Integer>0 or Real>0.0 or Character or Blank)	ATTi (i=1,2,3...) (Integer>0, or Blank if specified)
TVELO	Velocity Component	Time Value (Blank, Real>0.0, or Character) See Remarks 17 and 21.	Grid ID(s) See Remark 28.
VOLUME	Blank	Blank	SEIDi or ALL
WEIGHT	Row Number (1 ≤ ROW ≤ 6) See Remark 26.	Column Number (Integer) (1 ≤ COL ≤ 6)	SEIDi or All or blank

REMARKS:

1. Stress, strain, and force item codes can be found in “Item Codes”. For stress or strain item codes that have dual meanings, such as von Mises or maximum shear, the option specified in the Case Control Section will be used; i.e., STRESS(VONM) or STRESS(MAXS). Note that item codes may vary depending on whether CENTER or CORNER is chosen with the relevant output request in case control.

2. RTYPE = “CSTRESS”, “CSTRAIN”, and “CFAILURE” are used only with the PCOMP and PCOMPG entries. “CSTRESS” and “CSTRAIN” item codes are described under Table 1 (Element Stress/Strain Item Codes) in “Item Codes”. “CFAILURE” item codes are described under Table 2 (Element Force Item Codes) in “Item Codes”. Only force item codes that refer to failure indices of direct stress and interlaminar shear stress are valid.

   The CFAILURE response type requires the following specifications on the applicable entries:

   - Failure theory in the FT field on PCOMP entry.
   - Allowable bonding shear stress in the SB field on PCOMP entry.
   - Stress limits in the ST, SC, and SS fields on all MATi entries.

3. ATTB is used only for responses of composite laminae, dynamics, complex eigenvalue, and stability derivatives. For other responses, this field must be

blank. When including the ID of a PCOMPG entry in a DRESP1 response, the lamina number entered in the ATTB field is relative to the order of the ply definitions (1,2,3, ..) and is not the global ply id.

4. All grids associated with a DRESP1 entry are considered to be in the same region for screening purposes. Only up to NSTR displacement constraints (see DSCREEN entry) per group per load case will be retained in the design optimization phase.

5. DRESP1 identification numbers must be unique with respect to DRESP2 and DRESP3 identification numbers.

6. If PTYPE = “ELEM”, the ATTi correspond to element identification numbers. If RTYPE=”ERP” and PTYPE = blank or “PANEL”, the ATTi correspond to one or more panel related SET3 IDs (as in the PANEL bulk data).

7. If RTYPE = “DISP”, “TDISP”, “TVELO”, “TACCL”, “TSPCF”, “TFORC” or “TSTRE”, multiple component numbers (any unique combination of the digits 1 through 6 with no embedded blanks) may be specified on a single entry. Multiple response components may not be used on any other response types.

8. For grid frequency response type DRESP1, i.e. those starting with “FR”, “RMS”, or “PSD”, only one component number may be specified in the ATTA field. Numbers 1 through 6 correspond to real (or magnitude) components and 7 through 12 imaginary (or phase) components. If more than one component for the same grid is desired, then a separate entry is required.

   Note that, in case the response is used for a constraint, the choice of response computation format (real/imaginary vs. magnitude/phase) will affect constraint values. See also Remark 9.

9. Real/imaginary representation is the default for complex response types. Magnitude/phase representation must be requested by the corresponding Case Control command; e.g., DISP(PHASE) = ALL.

10. REGION is used for constraint screening. The NSTR field on DSCREEN entries gives the maximum number of constraints retained for each region per load case.

    IF RTYPE = “WEIGHT”, “VOLUME”, “LAMA”, “EIGN” or “FREQ”, no REGION identification number should be specified. For all other responses, if the REGION field is left blank, the default specified in Table 13-6 is used. Usually, the default value is appropriate.

    If the REGION field is not blank, all the responses on this entry as well as all responses on other DRESP1 entries that have the same RTYPE and REGION identification number will be grouped into the same region.

Table 13-6. Default Regions for Design Sensitivity Response Types

Response Type	Default Region
EIGN	No region
FREQ	No region
LAMA	No region
VOLUME	No region
WEIGHT	No region
DISP	One region per DRESP1 entry
ERP	One region per DRESP1 entry
FLUTTER	One region per DRESP1 entry
FRACCL	One region per DRESP1 entry
FRDISP	One region per DRESP1 entry
FRSPCF	One region per DRESP1 entry
FRVELO	One region per DRESP1 entry
PRES	One region per DRESP1 entry
SPCFORCE	One region per DRESP1 entry
TACCL	One region per DRESP1 entry
TDISP	One region per DRESP1 entry
TSPCF	One region per DRESP1 entry
TVELO	One region per DRESP1 entry
OTHER	One region per PROPERTY entry. If PTYPE = “ELEM”, then one region per DRESP1 entry

11. REGION is valid only among the same type of responses. Responses of different types will never be grouped into the same region, even if they are assigned the same REGION identification number by the user.

12. If RTYPE = “WEIGHT” or “VOLUME”, field ATTi = “ALL” implies total weight/volume of all superelements except external superelements.

13. RTYPE = “STABDER” identifies a stability derivative response. ATTB is the restraint flag for the stability derivative. ATTB = 0 means unrestrained, and ATTB = 1 means restrained. For example, ATTA = 4000, ATTB = 0, and ATT1 = 3

reference the unrestrained C_z derivative for the AESTAT (or AESURF) entry ID = 4000.

14. Aeroelastic static subcases that reference only constrained STABDER responses must come after any aeroelastic static subcases that reference other types of constrained responses or reference a combination of other constrained responses including STABDER responses.

15. RTYPE = “FLUTTER” identifies a set of damping responses. The set is specified by ATTi:

    ATT1 = Identification number of a SET1 entry that specifies a set of modes.

    ATT2 = Identification number of an FLFACT entry that specifies a list of densities.

    ATT3 = Identification number of an FLFACT entry that specifies a list of Mach numbers.

    ATT4 = Identification number of an FLFACT entry that specifies a list of velocities.

    If the flutter analysis is type PKNL, it is necessary to put PKNL in the PTYPE field of this entry.

16. For RTYPE = ”ERP”, FRDISP”, “FRVELO”, “FRACCL”, “FRSPCF”, “FRFORC”, “FRSTRE”, “PSDDISP”, “PSDVELO”, and “PSDACCL”, a real value for ATTB specifies a frequency value in cycles per unit time. If ATTB is specified, then the responses are evaluated at the closest frequency selected by the OFREQ command. The default for ATTB is all frequencies selected by the OFREQ command. See Remark 21 for additional ATTB options.

17. For RTYPE = “TDISP”, “TVELO”, “TACCL”, “TSPCF”, “TFORC”, and “TSTRE”, ATTB specifies a time value. If ATTB is specified, then the responses are evaluated at the closest time selected by the OTIME command. The default for ATTB is all time steps selected by the OTIME command.

18. Intermediate station responses on CBAR elements due to PLOAD1 and/or CBARAO entries may not be defined on the DRESP1 entry.

19. RTYPE = “EIGN” refers to normal modes response in terms of eigenvalue (radian/time)**2 while RTYPE = “FREQ” refers to normal modes response in terms of natural frequency or units of cycles per unit time.

20. For RTYPE = LAMA, EIGN or FREQ, the response approximation used for optimization can be individually selected using the ATTB field. (Approximation Code = 1 = direct linearization, = 2 = Inverse Linearization).

21. Character input for ATTB is available for RTYPE of ERP, FRDISP, FRVELO, FRACCL, FRSPCF, PSDDISP, PSDVELO, PSDACCL, TDISP, TVELO, TACCL, TSPCF, FRFORC, FRSTRE, TFORC, and TSTRE. The character input represents

a mathematical function and the options for character input are SUM, AVG, SSQ, RSS, MAX and MIN. The expression of mathematical function is shown as follows:

$$\mathrm{SUM}(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i$$

$$\mathrm{AVG}(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i / n$$

$$\mathrm{SSQ}(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i^2$$

$$\mathrm{RSS}(X_1, X_2, \ldots, Xn) = \sqrt{\sum_{i=1}^{n} X_i^2}$$

MAX (X_1, X_2, ..., X_n) = Largest value among X_i (i=1 to n)

MIN (X_1, X_2, ..., X_n) = Smallest value among X_i (i=1 to n)

where X_i is the response for a forcing frequency. For example

DRESP1,10,DX1,FRSTRE,ELEM,,3,AVG,10

yields a response which is equal to the average stress for element 10. The average is done by first adding up the component 3 stress of element 10 for all forcing frequencies (all time steps for transient response), and then dividing by the number of forcing frequencies. Note that the response computed is considered as type 2 response. **Therefore, if referenced on a DRESP2, the ID of such DRESP1 (ATTB with character input) must be listed following the DRESP2 keyword**.

Such DRESP1 may only be referenced from a DRESP2 (not from a DRESP3), and may only be nested one level. That is, the referencing DRESP2 may not be referenced further from another DRESP2 or a DRESP3.

(Important note: If any such integrated response DRESP1 is either erroneously or intentionally referred to as DRESP1 from within a DRESP2 or DRESP3, the program will spawn multiple responses over the frequency set, or, for the objective function, take only the value corresponding to the first frequency in the set, not the integrated response, whether or not the DRESP1 is DRSPAN related.)

22. Element strain energy item codes can be found in "Element Strain Energy Item Codes". Only element strain energy and element strain energy density can be referenced on a DRESP1 entry.

23. For RTYPE=CEIG, the allowable character inputs are ALPHA (for the real component) and OMEGA (for the imaginary component), with ALPHA being the default.

24. For RTYPE=RMSDISP, RMSVELO, or RMSACCL the ATTB specifies the appropriate RANDPS ID.

25. Input other than 1 or 7 of ATTA field, acoustic pressure component, for PRES response type will be reset to 1 (if less than 7) or 7 (if greater than 7 and less than 13).

26. Design response weight is obtained from the Grid Point Weight Generator for a reference point (see the parameter GRDPNT). For SOL 200, if PARAM,GRDPNT is either not defined, equal to zero, or not a defined grid point, the reference point is taken as the origin of the basic coordinate system. Fields ATTA and ATTB refer to the row and column numbers of the rigid body weight matrix, which is partitioned as follows:

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6 \times 6}$$

The default value of ATTA and ATTB is 3, which specifies the Wz value. Typically the value of Wx, Wy, and Wz each equal the weight of the structure. Although, if directional mass is defined, which is possible using the CONM1, CMASS, or DMI bulk entries, the values can be different. If you are uncertain about the weight matrix value to use in your response, include the parameter GRDPNT to print the weight matrix in the .f06 file.

The first field of ATTi: ATT1 = "ALL" implies total weight of all superelements except external superelements. When the ATTi fields contain SEIDi, each SEIDi refers to a superelement identification number. SEIDi= "0" refers to the residual

superelement. The default for this first field of ATTi, ATT1, is blank, which is equivalent to “ALL”.

27. PSD and RMS type DRESP1 responses are meaningful only for those random loading frequency response analysis subcases listed in the RANDPS bulk data referred to by the relevant RANDOM ID.

28. Multiple grid point IDs in the ATTi fields are supported.

29. Since PSD type DRESP1 responses are a summation over the sets of relevant subcases correlated in the RANDPS bulk data, a constraint on a PSD type DRESP1 response should be specified only in the lead subcase of such a batch of subcases. Any PSD response specified otherwise will be ignored. For example, if the subcases referred to by RANDPS ID=75 are (3,3); (4,4); (5,5); and (3,5), where the last two numbers indicate cross-correlation among two subcases, any PSD response for this particular RANDPS ID should be placed in subcase 3, as that is the lead subcase.

30. If a PSD type response is to appear in a DRESP2 or DRESP3 response together with responses from other subcases, it should then be assigned to the lead subcase defined in item 28 above by an appropriate DRSPAN case control command and a related set.

31. Models with superelements will either ignore any RMS and PSD type DRESP1 design responses, or will fail in the presence of such DRESP1.

32. For PBUSHT, PDAMPT, and PELAST property types, currently use PBUSH, PDAMP, or PELAS, as appropriate, in the PTYPE field, since the latter have the same IDs as the corresponding frequency dependent properties.

33. ATTA can be 0 for ERP magnitude responses. This is equivalent to leaving it blank.

34. Currently, for ERP design responses, any entry or blank in the PTYPE field defaults to PANEL.

DRESP2

Design Equation Response Quantities

Defines equation responses that are used for the objective and/or design constraints, or for sensitivity analysis purposes.

FORMAT:

1	2	3	4	5	6	7	8	9
DRESP2	ID	LABEL	EQID or FUNC	REGION				
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7
		DVID8	-etc.-					
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7
		LABL8	-etc.-					
	“DFRFNC”	DFRFID1	DFRFID2	DFRFID3	DFRFID4	DFRFID5	DFRFID6	DFRFID7
		DFRFID8	-etc.-					
	“DRESP1”	NR1	NR2	NR3	NR4	NR5	NR6	NR7
		NR8	-etc.-					
	“DNODE”	G1	C1	G2	C2	G3	C3	
		G4	C4	etc.				
	“DVPREL1”	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7
		DPIP8	DPIP9	-etc.-				
	“DVCREL1”	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7
		DCIC8	DCIC9	-etc.-				
	“DVMREL1”	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7
		DMIM8	DMIM9	-etc.-				
	“DVPREL2”	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7
		DPI2P8	DPI2P9	-etc.-				
	“DVCREL2”	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7
		DCI2C8	DCI2C9	-etc.-				
	“DVMREL2”	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7
		DMI2M8	DMI2M9	-etc.-				
	“DRESP2”	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7
		NRR8	-etc.-					

EXAMPLE:

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
LABEL	User-defined label. (Character. See Bulk Data Syntax Rules.)
EQID	DEQATN entry identification number. (Integer>0)
FUNC	Function to be applied to the arguments. See Remark 8. (Character)
REGION	Region identifier for constraint screening. See Remark 5. (Integer>0)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
“DTABLE”	Flag indicating that the labels for the constants in a DTABLE entry follow. (Character)

Field	Contents
LABLj	Label for a constant in the DTABLE entry. (Character. See Bulk Data Syntax Rules.)
“DFRFNC”	Flag indicating DFRFNC ID numbers to be found in DTI, DFRFNC table records.
DFRFIDi	ID number for a DFRFNC record in the DTI, DFRFNC table. Usually there will be a single DFRFNC ID entry here. However, as long as there is a match in the number of frequencies for which the DRESP2 is to be evaluated, there may be multiple DFRFNC ID numbers.
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer>0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
Gm	Identification number for any grid point in the model. (Integer>0)
Cm	Component number of grid point Gm. (1 ≤ Integer ≤ 3)
DVPREL1	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer>0)
DVCREL1	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer>0)
DVMREL1	Flag indicating DVPREL2 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer>0)
DVPREL2	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer>0)
DVCREL2	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer>0)
DVMREL2	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number. (Integer>0)

Field	Contents
DRESP2	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer>0)

REMARKS:

1. DRESP2 entries may only reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, and DVMREL2 entries. They may also reference other DRESP2 entries. However, a DRESP2 entry cannot reference itself directly or recursively.

2. Referenced DRESP1 entries cannot span analysis types or superelements, except when such DRESP1 are specified in DRSPAN referenced sets, in which case they may span subcases and analysis types (see also Remark 11 below, and the remarks on the DRSPAN case control command).

3. DRESP2 entries must have unique identification numbers with respect to DRESP1 and DRESP3 entries.

4. The “DESVAR”, “DTABLE”, “DFRFNC”, “DRESP1”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, DVCREL2”, “DVMREL2”, and “DRESP2” flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP2 relationship. However, at least one of these twelve types of arguments must exist.

5. The REGION field follows the same rules as for the DRESP1 entries. DRESP1, DRESP2, and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP2 entry in the same region.

6. The variables identified by DVIDi, LABLj, NRk, the Gm, Cm pairs, DPIPi, DCICm, DMIMn, DPI2Po, DCI2Cp, DMI2Mq, and NRRu are assigned (in that order) to the variable names (x1, x2, x3, etc.) specified in the left-hand side of the first equation on the DEQATN entry referenced by EQID. In the example below,

   DESVARs 101 and 3 are assigned to arguments A and B.DTABLEs PI and YM are assigned to arguments C and D.Grid 14, Component 1 is assigned to argument R.

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3						

	DTABLE	PI	YM						
	DNODE	14	1						
DEQATN	5	F1(A, B, C, D, R)=A+B*C-(D**3+10.0)+sin(C*R)							

7. (Gm, Cm) can refer to any grid component and is no longer limited to a designed grid component.

8. The FUNC attributes can be used in place of the EQID and supports the functions shown in the following table:

Function	Description
AVG	Average of the arguments
SSQ	Sum of the squares of the arguments
RSS	Square root of the sum of the squares of the arguments
MAX	The largest value of the argument list
MIN	The smallest value of the argument list

When EQID has character input, the DEQATN entry is no longer needed. The functions are applied to all arguments on the DRESP2 regardless of the type.

9. The number of arguments of a DEQATN can be more than the number of values defined on the DRESP2 if the DRESP1s referenced have RTYPE with ‘FR’ prefix. Arguments are still positional. The extra arguments in the DEQATN must appear at the end of the argument list. The discrepancy is resolved internally with the forcing frequency(ies) associated with DRESP1s. An example is shown as follows:

DRESP1	10	FDISP1	FRDISP			1	10.	1001	
DRESP1	20	FDISP2	FRDISP			1	20.	1001	
DRESP2	30	AVGFD	100						
	DRESP1	10	20						
DEQATN	100	AVG(D1,D2,F1,F2) = (D1/F1+D2/F2)*0.5							

In the above example, the DEQATN has two more additional terms than have been defined on the DRESP2. The first additional term is the forcing frequency (in hertz) of the first DRESP1 ID on the DRESP2. The second additional term is the forcing frequency of second DRESP1 ID in the list. When all DRESP1s involved have the same frequency, the user is not required to name all the additional terms in the argument list of DEQATN.

10. The DRESP1 IDs listed under a DRESP2 response may only point to DRESP1s with the same number of responses, in case the DRESP1s are spawned for multiple frequencies or time steps.

11. When referenced DRESP2 entries are in fact DRESP1 that are mathematical functions (integrated responses) described in Remark 20 for DRESP1, they may again span subcases and analysis types, if specified in DRSPAN referenced sets.

DRESP3

Externally Computed Design Responses

Defines SOL 200 design responses to be evaluated in an external user-supplied program. The differences from the DRESP2 data are indicated in bold. Such responses are then used for the objective and/or design constraints, or for sensitivity analysis purposes.

FORMAT:

1	2	3	4	5	6	7	8	9
DRESP3	ID	LABEL	**GROUP**	**TYPE**	REGION			
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7
		DVID8	-etc.-					
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7
		LABL8	-etc.-					
	“DFRFNC”	DFRFID1	DFRFID2	DFRFID3	DFRFID4	DFRFID5	DFRFID6	DFRFID7
		DFRFID8	-etc.-					
	“DRESP1”	NR1	NR2	NR3	NR4	NR5	NR6	NR7
		NR8	-etc.-					
	“DNODE”	G1	C1	G2	C2	G3	C3	
		G4	C4	etc.				
	“DVPREL1”	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7
		DPIP8	DPIP9	-etc.-				
	“DVCREL1”	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7
		DCIC8	DCIC9	-etc.-				
	“DVMREL1”	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7
		DMIM8	DMIM9	-etc.-				
	“DVPREL2”	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7
		DPI2P8	DPI2P9	-etc.-				
	“DVCREL2”	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7
		DCI2C8	DCI2C9	-etc.-				
	“DVMREL2”	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7
		DMI2M8	DMI2M9	-etc.-				
	“DRESP2”	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7

DRESP3

1	2	3	4	5	6	7	8	9
		NRR8	-etc.-					
	“USRDATA”	**String**	**(character strings in user-defined format for any use in the evaluator)**					
		-etc.-						

EXAMPLE:

DRESP3	32	OBJ1	OBJ1GRP	REG1					
	DESVAR	1	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						
	USRDATA	102,ABCD	EF,(X-Y)						

FIELDS:

Field	**Contents**
ID	Unique identification number. (Integer>0)
LABEL	User-defined label which is referenced by a constraint bulk data entry or by the objective function. (Character. See Bulk Data Syntax Rules.)

Field	Contents
GROUP	Selects a specific external response routine. (Character) See Remark 2.
TYPE	Refers to a specific user-created response calculation type in the external function evaluator. See Remark 3. (Character)
REGION	Region identifier for constraint screening. See Remark 5. (Integer>0)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
“DTABLE”	Flag indicating that the labels for the constants in a DTABLE entry follow. (Character)
LABLj	Label for a constant in the DTABLE entry. (Character. See Bulk Data Syntax Rules.)
“DFRFNC”	Flag indicating DFRFNC ID numbers to be found in DTI, DFRFNC table records.
DFRFIDi	ID number for a DFRFNC record in the DTI, DFRFNC table. Usually there will be a single DFRFNC ID entry here. However, as long there is a match in the number of frequencies for which the DRESP3 is to be evaluated, there may be multiple DFRFNC ID numbers.
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer>0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
Gm	Identification number for any grid point in the model. (Integer>0)
Cm	Component number of grid point Gm. (1 ≤ Integer ≤ 3)
DVPREL1	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer>0)
DVCREL1	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer>0)
DVMREL1	Flag indicating DVPREL2 entry identification number. (Character)

Field	Contents
DMIMi	DVMREL1 entry identification number. (Integer>0)
DVPREL2	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer>0)
DVCREL2	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer>0)
DVMREL2	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number. (Integer>0)
DRESP2	Flag indicating DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer>0)
“USRDATA”	Flag indicating user input data (Character).

REMARKS:

1. DRESP3 entries must have unique identification numbers with respect to DRESP1 and DRESP2 entries.

2. The “GROUP” field must reference the group defined on a FMS CONNECT statement.

3. The “TYPE” field is used to select one type of response evaluation, since more than one option can exist in the external user-created routine.

4. The “DESVAR”, “DTABLE”, “DFRFNC”, “DRESP1”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, DVCREL2”, “DVMREL2”, “DRESP2”, and USRDATA flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.

5. The REGION field is used with the DSCREEN bulk entry in the constraint screening process as discussed in the section titled ”Constraint Regionalization and Deletion” in chapter one of the book *NX Nastran Design and Optimization User's Guide*. If REGION is defined, all responses which share the REGION id

and have the same response type will be evaluated together according to the DSCREEN bulk entry which also has the same response type.

The REGION field follows the same rules as for the DRESP1 and DRESP2 entries. DRESP1, DRESP2 or DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.

6. The string input for USRDATA is limited to 32000 characters.

7. The DRESP1 IDs listed under a DRESP3 response may only point to DRESP1s with the same number of responses, in case the DRESP1s are spawned for multiple frequencies or time steps.

8. Template Fortran subroutines are provided for DRESP3, together with NX Nastran, wherein the user can modify these subroutines and add, if necessary, new called subroutines to complete a customized external program. More information is supplied with the template subroutines as to the data they receive by way of the DRESP3 bulk data. The external program can then be built using the files provided for this purpose. While the default name for this external program will be dr3serv, the user can modify the build file to change this to a desired name.

9. The external program described in Remark 8 is referenced by way of three entities: a) the CONNECT statement in the data file, which provides an alias for the path name, b) a separate file which resolves this alias by providing the corresponding path name, and c) the “nastran” statement which refers to this separate file by the gmconn option (i.e. gmconn = filename).

10. Referenced DRESP1 entries cannot span analysis types or superelements, except when such DRESP1 are specified in DRSPAN referenced sets, in which case they may span subcases and analysis types (see the remarks on the DRSPAN case control command).

11. DRESP1 that are mathematical functions described in Remark 20 for DRESP1 may not be referenced by DRESP3.

12. See Defining Design Responses with External Programs in the Design Sensitivity and Optimization User's Guide, and the CONNECT File Management statement.

DSCREEN

Design Constraint Screening Data

Defines screening data for constraint deletion.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DSCREEN	RTYPE	TRS	NSTR						

EXAMPLE:

DSCREEN	STRESS	-0.7	2						

FIELDS:

Field	Contents
RTYPE	Response type for which the screening criteria apply. See Table 13-7 and Remark 3 . (Character)
TRS	Truncation threshold. (Real; Default = -0.5)
NSTR	Maximum number of constraints to be retained per region per load case. See Remark 3 . (Integer>0; Default = 20)

REMARKS:

1. Displacement and displacement derivative constraints associated with one particular load case are grouped by the specification of DRESP1 entries. From each group, a maximum of NSTR constraints are retained per load case.

Table 13-7. Default Region Specifications for RTYPE

RTYPE	Default Region Specification		RTYPE	Default Region Specification
CFAILURE	Property ID		FRSPC	DRESP1
CSTRAIN	Property ID		FRSTRE	Property ID
CSTRESS	Property ID		FRVELO	DRESP1
DISP	DRESP1		SPCFORCE	DRESP1
DRESP3	DRESP3		STRAIN	Property ID
EQUA	DRESP2		STRESS	Property ID
ESE	Property ID		TACCL	DRESP1
FLUTTER	DRESP1		TDISP	DRESP1
FORCE	Property ID		TFORC	Property ID
FRACCL	DRESP1		TSPC	DRESP1
FRDISP	DRESP1		TSTRE	Property ID
FRFORC	Property ID		TVELO	DRESP1

2. Stress-strain constraints are grouped by the property; i.e., all elements belonging to the set of PIDs specified under ATTi on a DRESPi entry are regarded as belonging to the same region. In superelement sensitivity analysis, if the property (PID) is defined in more than one superelement, then separate regions are defined. A particular stress constraint specification may be applied to many elements in a region generating many stress constraints, but only up to NSTR constraints per load case will be retained.

3. If RTYPE = “WEIGHT”, “VOLUME”, “EIGN”, “FREQ”, “LAMA”, “TRIM”, or “STABDER” then NSTR is unlimited and must be left blank. TRS is available for all response types.

4. If a certain type of constraint exists but no corresponding DSCREEN entry is specified, all the screening criteria used for this type of constraint will be furnished by the default values.

5. Constraints can be retained only if they are greater than TRS. See the Remarks under the “DCONSTR” entry for a definition of constraint value.

6. Constraint screening is applied to each superelement.

DTABLE

Table Constants

Defines a table of real constants that are used in equations (see DEQATN entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTABLE	LABL1	VALU1	LABL2	VALU2	LABL3	VALU3	LABL4	VALU4	
	LABL5	VALU5	LABL6	VALU6	-etc.-				

EXAMPLE:

DTABLE	PI	3.142	H	10.1	E	1.0E6	T	0.1	
	G	5.5E5	B	100.					

FIELDS:

Field	Contents
LABLi	Label for the constant. (Character. See Bulk Data Syntax Rules.)
VALUi	Value of the constant. (Real)

REMARKS:

1. Only one DTABLE entry may be specified in the Bulk Data Section.
2. LABLi are referenced by the LABi on the DVPREL2, DRESP2, or DRESP3 entries.

DTEMP

Time dependent temperature set definition for SOLs 401, 601 and 701.

Defines a time dependent temperature set.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTEMP	SID								
	T1	TID1	T2	TID2	T3	TID3	T4	TID4	
	T5	TID5	-etc.-						

EXAMPLE:

DTEMP	200								
	0.	1	45.	2	50.	3	60.	4	
	105.	5	160.	6	170.	7	180..	8	

FIELDS:

Field	Contents
SID	Unique identification number specified in the DTEMP case control command. (Integer>0)
Ti	Times in increasing order of magnitude. (Real ≥ 0.0) T1 < T2 < T3 <…< Tn
TIDi	SID of TEMP or TEMPD bulk entries (Integer>0).

REMARKS:

1. SID must be unique to all other DTEMP entries.

2. The SID of the DTEMP bulk entry can only be selected with the DTEMP case control command. The TEMPERATURE case control command cannot be used to select the DTEMP bulk entry.

3. The same TIDi can be used multiple times on the same DTEMP entry in order to define the same temperature conditions for multiple times.

4. All TID on a DTEMP entry must have temperatures assigned to the same set of grid points.

DTEMPEX

Time dependent temperature set defined in a .bun file for SOL 401.

Defines a time dependent temperature set using a .bun file.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTEMPEX	SID	UNID							

EXAMPLE:

DTEMPEX	300	22							

FIELDS:

Field	Contents
SID	Unique identification number specified in the DTEMP case control command. (Integer>0)
UNID	Fortran unit number of an external .bun file included on an ASSIGN statement. For example, ASSIGN BUN='temperature0.bun' UNIT=22. (Integer>0)

REMARKS:

1. SID must be unique to all other DTEMPEX entries.
2. The SID of the DTEMPEX bulk entry can only be selected with the DTEMP case control command. The TEMPERATURE case control command cannot be used to select the DTEMPEX bulk entry.

DTI

Direct Table Input

Defines table data blocks.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	NAME	“0"	T1	T2	T3	T4	T5	T6	
	V01	V02	-etc.-						
DTI	NAME	IREC	V1	V2	V3	V4	V5	V6	
	V7	V8	V9	V10	-etc.-	“ENDREC”			

EXAMPLE: (THE FIRST LOGICAL ENTRY IS THE HEADER ENTRY.)

DTI	XXX	0	3	4	4096		1	0	
	1.2	2.3							
DTI	XXX	1	2.0	-6	ABC	6.000	-1	2	
	4	-6.2	2.9	1	DEF	-1	ENDREC		

FIELDS:

Field	Contents
NAME	Any character string that will be used in the DMAP sequence to reference the data block. See Remark 1 . (Character; the first character must be alphabetic.)
Ti	Trailer values. (Integer ≥ 0; Default = 32767)
IREC	Record number. (Integer>0)
V0i, Vi	Value. (Integer, Real, Character or blank)
“ENDREC”	Flags the end of the string of values (V0i or Vi) that constitute record IREC. (Character)

REMARKS:

1. The user defines the data block and therefore must write a DMAP (or ALTER a solution sequence), which includes the DTIIN modules, in order to use the DTI feature. See the *NX Nastran DMAP Programmer's Guide*. All of the rules governing the use of data blocks in DMAP sequences apply.

2. All fields following ENDREC must be blank.

3. The entry using IREC = 0 is called the header entry and is an optional entry. The values T1 through T6 go to a special record called the trailer. Other values on the optional continuation go to the header record. If the header entry or the trailer is not specified, T1 through T6 = 32767. On this entry, "ENDREC" may be used only if there is at least one continuation.

4. In addition to the optional header entry, there must be one logical entry for each record in the table. Null records require no entries.

5. "ENDREC" is used to input blank values at the end of a record. If "ENDREC" is not specified, the string for a record ends with the last nonblank field.

6. The maximum number of DMI and DTI data blocks is 1000.

7. If Ti is not an integer, a machine-dependent error will be issued that may be difficult to interpret.

DTI,DFRFNC

Frequency function input for use in DRESP2 & DRESP3

Provides a way to input functions of frequency so that they can be automatically looped over when spawning multiple responses over the frequency set when using DRESP2 or DRESP3. Such functions can be, for example, frequency dependent DRESP1 response bounds. More than one such function can be entered in sequence.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	DFRFNC	“0”							
DTI	DFRFNC	IREC	FID	FTYPE	V1	V2	V3	V4	
	V5	V6	V7	V8	-etc.-				

EXAMPLE:

DTI	DFRFNC	0								
DTI	DFRFNC	1	325	LISTING	0.036	0.038	0.0395	0.043		
		0.049	0.054							
DTI	DFRFNC	2	513	EQUATION	750					
DTI	DFRFNC	3	517	EQUATION	755					
DTI	DFRFNC	4	647	CONSTANT	12.34					
DTI	DFRFNC	5	116	INCRMENT	3.0	0.2				

FIELDS:

Field	Contents
IREC	Logical DFRFNC record number. The first DTI,DFRFNC record should be 0, and unreferenced. Each later DTI,DFRFNC record should point to a separate DFRFNC ID (FID).
FID	The ID of the DFRFNC record as referenced from a DRESP2 or DRESP3 type response.

Field	Contents
FTYPE	DFRFNC function input format type. There are currently four different ways of inputting DFRFNC function values: CONStant; EQUAtion; INCRment; LISTing. (It is sufficient to enter the first four letters as shown by capitals here). The format of the rest of the data for each DFRFNC record depends on the FTYPE for that record.
V1	For FTYPE = CONS, V1 is the constant real value of the function. For FTYPE = EQUA, V1 is the ID (EQID) of the equation by which the function values are to be computed, with the frequency values in the relevant frequency set being the only variable in the equation. Thus, the equation loops over the frequencies in the relevant frequency set, and computes the corresponding DFRFNC function value for each frequency in the set. For FTYPE = INCR, V1 is the starting value of the incrementation, i.e. the value of the frequency function corresponding to the first frequency in the relevant frequency set. For FTYPE = LIST, V1 is the first value of the frequency function, i.e. the value of the frequency function corresponding to the first frequency in the relevant frequency set.
V2	For FTYPE = INCR, V2 is the value of the increment by which the function of the frequency will change. The program automatically increments the function value for the actual number of frequencies in the relevant frequency set. For FTYPE = LIST, V2 is the value of the frequency function corresponding to the second frequency in the relevant frequency set.
Vi	These are currently valid only for FTYPE = LIST. Vi corresponds to the value of the frequency function corresponding to the ith frequency in the frequency set.

REMARKS:

1. As DTI type table data makes no check on the validity of the data, the user should exercise care in input. It is anticipated that this input will be replaced or paralleled by bulk data input DFRFNC in the future.

2. When a DFRFNC ID is referenced by a DRESP2 or DRESP3 type bulk data, this is normally to enable the spawning of multiple DRESP2 or DRESP3 data over the relevant frequency set. Such spawning of multiple responses is triggered by

nested DRESP1 data within the DRESP2 or DRESP3, where such DRESP1 responses are spawned over the frequency set by leaving the ATTB field blank in the DRESP1 input.

3. DFRFNC normally should not be used for cases where multiple DRESP2 or DRESP3 responses are not going to be spawned over the relevant frequency set. Simple table input of a constant value is sufficient in these cases.

4. Normally, the relevant frequency set is determined by the subcase which indirectly references the DRESP2 or DRESP3 response in question. However, for DRESP2 or DRESP3 responses which contain DRSPAN related DRESP1 responses spawned over one or more frequency sets, it is required that the referenced frequency sets for the subcases to which these DRESP1 belong have the same number of frequencies. If the FTYPE = EQUA option is used, the frequency values will be picked up from the frequency set belonging to the last subcase to be analyzed in the relevant DRSPAN domain.

DTI,ESTDATA

Superelement Estimation Data Overrides

Provides override data for time and space estimation for superelement processing operations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	ESTDATA	“0"							
	kd1	vd1	kd2	vd2	-etc.-				

The next entries are repeated for any superelement for which estimate data overrides are desired. IREC must be incremented by 1.

DTI	ESTDATA	IREC	SEFLAG	SEID	k1	v1	k2	v2	
	k3	v3	-etc.-						

EXAMPLE:

DTI	ESTDATA	0							
	NOMASS	-1							
DTI	ESTDATA	1	SE	10	C1	5.5	C3	4.5	
	C7	7.3							

FIELDS:

Field	Contents
kdi	Keyword for estimation parameter. (Character from Table 13-8.)
vdi	Value assigned to the estimation parameter kdi. (The type given in Table 13-8.)
IREC	Record number beginning with 1. (Integer>0)

Field	Contents
SEFLAG	SEFLAG = “SE” or “SEID” indicates the next field containing a superelement identification number. (Character)
SEID	Superelement identification number. (Integer>0)
ki	Keyword for override of estimation parameter for indicated superelement. (Character from Table 13-8.)
vi	Value for keyword ki. (Type depends on ki as shown in the Table 13-8 .)

Table 13-8. DTI,ESTDATA Input Parameter Descriptions

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
CRMS*	Real	-1.0	C	Number of active columns in $[K_{oo}]$.
FCRMS*	Real	0.10		If FCRMS ≤ 0.0, FCRMS is used (c/o).
C1	Real	6.0	c_1	Average number of degrees-of-freedom per grid point in o-set.
C3	Real	8.0	c_3	Average number of connections per grid point.
C4	Real	0.15	c_4	I/O time (seconds) per block transferred.
C5	Real	6.0	c_5	Average number of effective degrees-of-freedom per grid point in a-set.
C6	Real	1.2	c_6	Total CPU factor.
C7	Real	6.0	c_7	Number of equivalent KGG data blocks for space estimation.
WF	Real	-1.0	W	If WF ≤ 0.0 then use available working storage in units of single-precision words.
NOMASS	Integer	1		If NOMASS ≠ 1 then exclude mass terms from estimates.
TSEX	Real	0.5 (min)		Threshold limit for CPU.
SSEX	Real	50.0(blocks)		Threshold limit for space.
TWALLX	Real	5.0 (min)		Threshold limit for wall time.
BUFSIZ	Integer	Machine Buffsize	B	Buffsize.

Table 13-8. DTI,ESTDATA Input Parameter Descriptions

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
ML	Real	MachineLoop Time	M	Arithmetic time for the multiply/add loop. See the *NX Nastran Installation and Operations Guide*.
CONIO	Integer	MachineI/O ratio		I/O count/CPU equivalence
PREC	Integer	1 or 2		Machine Word Length (1 = long, 2 = short).
NLOADS	Integer	1	N_L	Number of loading conditions
SETYPE	Character	“T”		Superelement type (T = Tip)
CMAX	Real	-1.0	C_{max}	Maximum bandwidth

PARAMETERS OBTAINED FROM SEMAP

NGI	Number of interior grid points.
NPE	Number of exterior grid points.
NS	Number of scalar points
NE	Number of elements.

DERIVED PARAMETERS

$O=C1 \cdot NG1$	Size of o-set.
$A=C5(NPE - NS) + NS$	Size of a-set.
T = BUFFSIZE/PREC	Number of matrix terms in a buffer.

ESTIMATION EQUATIONS

For each superelement, estimates of CPU time and disk space are made using the following equations.

Table 13-9. Equations Used for CPU Time and Disk Space Estimate

Printout Symbol	Math Symbol	Equations
TD	T_1	$T_1 = 1/2 \cdot M \cdot O \cdot C^2$
TFBS	T_2	$T_2 = 2 \cdot M \cdot C \cdot O \cdot a$
TMAA	T_3	$T_3 = M \cdot O \cdot a^2$ (set to 0.0 if NOMASS ≠ +1)
TSE	T_{SE}	$T_{SE} = C_6(T_1 + T_2 + T_3)$
SLOO	S_1	$S_1 = O \cdot C \cdot \frac{\text{PREC}}{b}$
SGO	S_2	$S_2 = O \cdot a \cdot \frac{\text{PREC}}{B}$
SKGG	S_3	$S_3 = 36(NG_i + NG_e - NS)(c_3 + 1.0)\left(\frac{\text{PREC}}{B}\right)$
SSE	S_{SE}	$S_{SE} = S_1 + S_2 + c_7 \cdot S_3$
PASSES	p	$\text{FBS passes} = p = a \cdot O \cdot \frac{\text{PREC}}{WF}$
BKSTRN	BT	Blocks Transferred = $BT = 2 \cdot p \cdot S_1 + S_2 + p \cdot S_2$ (Last term omitted if NOMASS ≠ +1)
TWALL	T_W	Wall Time = $T_W = T_{SE} + c_4 \cdot BT$

REMARKS:

1. In the superelement solution sequences, this data is stored automatically.

2. The header record continuation entries are optional if no global override data is to be specified. In this case, the complete header entry is optional.

- Active column data can come from one of several places. The value for CRMS is determined as follows:
- RMS from the entry when IREC>0 and field 4 is “SE”.
- RMS from entries with IREC = 0.
- Computed bandwidth when PARAM,OLDSEQ is specified.
- If FCRMS is specified when IREC>0 and field 4 is “SE”, then CRMS = FCRMS · O.
- If FCRMS is specified when IREC = 0, then CRMS = FCRMS · O.
- CRMS = 0.1 · O.

3. If CMAX is not specified, then it is defaulted to CRMS.
4. In the example above, mass terms are excluded for all superelements and new values are given for parameters C1, C3, and C7 for Superelement 10 only.
5. The estimates for TSEX, SSEX, and TWALLX are not printed unless at least one estimate exceeds the threshold.

DTI,INDTA

Stress, Strain and/or Force Sort/Filter Item Code Override

Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	INDTA	“0"							

To specify/override items for a sort of stress quantities:

DTI	INDTA	“1"	B1	C1	B2	C2	“ENDREC”		

To specify/override items for a sort of force quantities:

DTI	INDTA	“2"	B1	C1	B2	C2	“ENDREC”		

EXAMPLE:

DTI	INDTA	0							

To specify/override items for a sort of stress quantities:

DTI	INDTA	1	64	18	75	18	ENDREC		

To specify/override items for a sort of force quantities:

DTI	INDTA	2	34	2	2	4	ENDREC		

FIELDS:

Field	Contents
Bi	Element type identification number. See the table in “Item Codes” for allowable values. (Integer>0)

Field	Contents
Ci	Item code identification number for the stress, strain, or force quantity on which the sort or filter is to be performed. See the table in the "Item Codes" for allowable values. (Integer)

REMARKS:

1. This table is recognized only in SOLs 101, 103, 105, 106, 108, 109, 111, 112, 114, 115, 144, 153, and for stress quantities only. One or more of the user parameters S1, S1G, or S1M must be specified with a value greater then or equal to zero in order to request sorting and/or filtering. See user parameter S1 in "Parameters and Parameter Applicability Tables" . In order to sort force or strain quantities, a DMAP Alter is required.

2. If the Ci value is -1, the element type will be suppressed on the output file. An example of this feature could be as follows: If an element type is to be sorted on two different values and output twice, this can be accomplished by two calls to the STRSORT module with two unique DTI tables. However, other element types will be printed twice. This additional print can be suppressed by setting their sort codes to -1.

3. Table 13-10 lists the elements currently that are sortable. In addition, the element type identification number, the default stress output quantity, and the associated stress code identification numbers are provided. If this entry is not specified, then the stresses are sorted based on the default quantity given in Table 13-10.

   The following should be noted:

   - The element type identification number is used internally by the program to differentiate element types.

   - The stress code identification number is merely the word number in the standard printed output for the stress quantity of interest. For example, the thirteenth word of stress output for the CHEXA element is the octahedral shear stress. For this element type, the element identification number and the grid point ID each count as a separate word. Stress codes for the elements are tabulated in "Item Codes" .

   - By default, stress sorting for the membrane and plate elements will be performed on the Hencky-von Mises stress. For maximum shear stress, the STRESS (MAXS) Case Control command should be specified.

Table 13-10. Sortable Elements

Element	Element Type ID Number	Default Stress Output Quantity and Identification Number	
		Quantity	Stress Code ID Number
CBAR	34	Maximum stress at end B	14
CBEAM	2	Maximum stress at end B	108
CBEND	69	Maximum stress at end B	20
CONROD	10	Axial stress	2
CELAS1	11	Stress	2
CELAS2	12	Stress	2
CELAS3	13	Stress	2
CHEXA	67	Hencky-von Mises or Octahedral stress	13
CQUAD4	33	Maximum shear or Hencky-von Mises stress at Z_2	17
CQUAD4*	144	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUAD8	64	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUADR	82	Maximum shear or Hencky-von Mises stress at Z_2	19
CPENTA	68	Octahedral stress	13
CROD	1	Axial stress	2
CSHEAR	4	No default	—
CTETRA	39	No default	—
CTRIA3	74	Maximum shear or Hencky-von Mises stress at Z_2	17
CTRIA6	75	Maximum shear or Hencky-von Mises stress at Z_2	19
CTRIAR	70	Maximum shear or Hencky-von Mises stress at Z_2	19
CTUBE	3	Axial stress	2

* CORNER output

DTI,SELOAD

External Superelement Load Set IDs

Usage overrides automatic generation of DTI,SELOAD definitions created through the use of the EXTSEOUT case control command.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	SELOAD	LIDSE	EXCSE	“0”	DESCRIPT	DESCRIPT (cont.)	DESCRIPT (cont.)	DESCRIPT (cont.)	
	-etc.-	“ENDREC”							

EXAMPLE:

DTI	SELOAD	1	100	0	FORCE_AT	_GRID_2	ENDREC	ELEM_100	

FIELDS:

Field	Contents
LIDSE	Load set identification number in the external superelement. See Remark 1. (Integer > 0)
EXCSE	Excitation identification number in the external superelement. See Remark 1. (Integer > 0)
DESCRIPT	Alphanumeric description of the loading. The description continues until ENDREC is encountered in a field. The description must be defined such that the first non-blank character in each field is not numeric. If the first non-blank character in a field is numeric, the field may be interpreted as numeric. If the field then contains any alphabetic character, an error will be reported. If the data in the field is all numeric, it will be shifted to the right. Imbedded blanks within a field may produce an error. To avoid this problem, use underscores to represent blank characters. (Character)

REMARKS:

1. The values of LIDSE and EXCSE depend on the describers specified on the EXTSEOUT case control command in the external superelement creation run.

   - For a static solution, two options are available. If the loading is referenced in the case control using the LOAD case control command, LIDSE and EXCSE are both equal to the set identification number of the LOAD case control command. If the loading is referenced in the case control using the LOADSET case control command, LIDSE and EXCSE are the values corresponding to the values of LID and EXCITEID, respectively, on each LSEQ bulk entry referenced by the LOADSET case control command. If both LOAD and LOADSET case control commands exist, LOADSET takes precedence.

   - For a dynamic solution, LIDSE and EXCSE are both equal to the EXCITEID value for each load (not enforced motion) defined on RLOADi and TLOADi bulk entries, regardless of whether or not they are referenced in the case control. The use of LOADSET and LSEQ is identical to that for a static solution.

   - Thermal loads and enforced motion loads defined using the SPCD bulk data method are not supported.

2. The use of DTI,SELOAD is optional. If a DTI,SELOAD bulk entry does not exist for an external superelement load set (LIDSE), one will be automatically generated. The description (DESCRIPT) of the automatically generated entry will have the form “SE_LOAD_xx”, where “xx” is the character representation of the external superelement load set.

3. DTI,SELOAD can be considered for use when the EXTSEOUT case control command specifies one of the following data storage options: MATDB (or MATRIXDB), DMIGDB, DMIGOP2 = unit, or MATOP4 (or MATRIXOP4) = unit. The data storage option DMIGPCH is only applicable to static analysis and requires the use of the P2G case control command. See the EXTSEOUT case control command for additional information.

DTI,SETREE

Superelement Tree Definition

Defines a superelement tree that determines the superelement processing order.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	SETREE	“1"	SEUP1	SEDOWN1	SEUP2	SEDOWN2	SEUP3	SEDOWN3	
	SEUP4	SEDOWN4	SEUP5	SEDOWN5	-etc.-				

EXAMPLE:

DTI	SETREE	1	1	14	2	14	3	14	
	4	14	14	0					

FIELDS:

Field	Contents
SEUPi	Identification number of the superelement upstream from SEDOWNi. (Integer>0)
SEDOWNi	Identification number of the superelement into which SEUPi is assembled. (Integer ≥ 0)

REMARKS:

1. SETREE entries or the DTI,SETREE entry are required for multi-level superelement configurations.
2. If an DTI,SETREE entry is provided, then SETREE entries are not required.

3. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.

4. Superelements not referenced on the SETREE or DTI,SETREE entry will have the residual structure as their downstream superelement.

5. If this entry is not present, the superelement tree and the processing order are determined automatically.

6. A superelement identification may appear only once in a SEUPi field.

7. On restart, if a superelement identification does not appear in a SEUPi field, its matrices will not be assembled, even though they may be present in the database.

8. See the *NX Nastran Superelement User's Guide* for a description of user-designated trees.

9. This entry is stored in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

10. In the example above, the following superelement tree is defined:

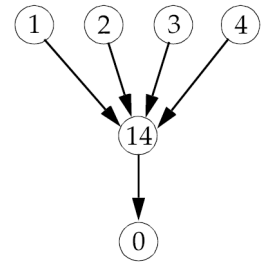


Figure 13-1. Sample Superelement Tree

DTI,SPECSEL

Response Spectra Input Correlation Table

Correlates spectra lines specified on TABLED1 entries with damping values.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	SPECSEL	RECNO		TYPE	TID1	DAMP1	TID2	DAMP2	
	TID3	DAMP3	TID4	DAMP4	TID5	DAMP5	-etc.-		

EXAMPLE:

DTI	SPECSEL	1		A	1	.02	2	.04	
	3	.06							
DTI	SPECSEL	3		V	4	.01			

FIELDS:

Field	Contents
RECNO	Spectrum number. (Integer>0)
TYPE	Type of spectrum. (Character: “A” for acceleration, “V” for velocity, or “D” for displacement.)
TIDi	TABLED1 entry identification number. (Integer>0)
DAMPi	Damping value assigned to TIDi. (Real)

REMARKS:

1. The RECNO is the number of the spectrum defined by this entry. It is referenced on DLOAD Bulk Data entries.

2. The TIDi, DAMPi pairs list the TABLEDI entry, which defines a line of the spectrum and the damping value assigned to it. The damping value is in the units of fraction of critical damping.

3. This entry is placed in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

DTI,SPSEL

Response Spectra Generation Correlation Table

Correlates output requests with frequency and damping ranges.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	SPSEL	RECNO	DAMPL	FREQL	G1	G2	G3	G4	
	G5	G6	G7	-etc.-					

EXAMPLE:

DTI	SPSEL	1	2	1	11	12			
DTI	SPSEL	2	4	3	1	7	11	12	
	13	14							

FIELDS:

Field	Contents
DAMPL	Identification number of the FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values. (Integer>0)
FREQL	Identification number of the FREQi Bulk Data entry that specifies the list of frequencies. (Integer>0)
Gi	Grid point number where response spectra will be calculated. (Integer>0)
RECNO	Record number of spectra to be generated. (Sequential integer beginning with 1.)

REMARKS:

1. This table is used in SOLs 109, and 112.

2. Damping values are in the units of fraction of critical damping.

3. Output of response spectra requires the use of the XYPLOT...SPECTRA(RECNO)/Gi... command, where Gi is restricted to the grid points listed on the (RECNO) record of this entry.

4. See the *NX Nastran User's Guide* for example problems using this feature.

5. The SPSEL table is stored in the database automatically in SOLs 109 and 112. Once stored, the Bulk Data entry may be removed from the input file.

DTI,UNITS

Specifies the system of units.

Specifies the system of units.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DTI	UNITS	“1”	MASS	FORCE	LENGTH	TIME	TEMPERATURE		

EXAMPLE:

DTI	UNITS	1	KG	N	M	S	F		

FIELDS:

Field	Contents
MASS	Defines the mass units Acceptable mass inputs are: KG - kilogram LBM – pound-mass (0.45359237 kg) SLUG – slug (14.5939029372 kg) GRAM – gram (1E-3 kg) OZM – ounce-mass (0.02834952 kg) KLBM – kilo pound-mass (1000 lbm) (453.59237 kg) MGG – megagram (1E3 kg) MG – milligram (1E-6 kg) MCG – microgram (1E-9 kg) NG – nanogram (1E-12 kg)

Field	Contents
	UTON – U.S. ton (907.18474 kg)
	SLI – slinch (175.1268352 kg)
FORCE	Defines the force units
	Acceptable force inputs are:
	N – Newton
	LBF – pound-force (4.44822161526 N)
	KGF – kilograms-force (9.80665 N)
	OZF – ounce-force (0.2780139 N)
	DYNE – dyne (1E-5 N)
	KN – kilonewton (1E3 N)
	KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)
	MN – millinewton (1E-3 N)
	MCN – micronewton (1E-6 N)
	NN – nanonewton (1E-9 N)
	CN – centinewton[1] (1E–2 N)
	P – poundal[1] (0.138254954 N)
LENGTH	Defines the length units
	Acceptable length inputs are:
	M – meter
	KM – kilometer (1E3 m)
	CM – centimeter (1E-2 m)
	MM – millimeter (1E-3 m)
	MI – mile (1609.344 m)
	FT – foot (0.3048 m)
	IN – inch (25.4E-3 m)
	MCM – micrometer (1E-6 m)
	NM – nanometer (1E-9 m)

Field	Contents
	A – Angstrom (1E-10 m)
	YD – yard (0.9144 m)
	ML – mil (25.4E-6 m)
	MCI – microinch (25.4E-9 m)
TIME	Defines the time units
	Acceptable time inputs are:
	S – second
	H – hour (3600.0 sec)
	MIN-minute (60.0 sec)
	MS – millisecond (1E-3 sec)
	MCS – microsecond (1E-6 sec)
	NS – nanosecond (1E-9 sec)
	D – day (86.4E3 sec)
TEMPERATURE	Defines the temperature units[2]
	Acceptable temperature inputs are:
	F – Fahrenheit
	R – Rankin
	C – Celsius
	K – Kelvin

[1]: not supported by MNF

[2]: not supported by MNF or RFI

REMARKS:

1. Since DTI,UNITS determines all units for the MNF and RFI, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to these files. For example, if the model mass is kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1

ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the MNF or RFI would be: “DTI,UNITS,1,KG,N,M,S”.

DVBSHAP

Design Variable to Boundary Shapes

Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVBSHAP	DVID	AUXMID	COL1	SF1	COL2	SF2	COL3	SF3	

EXAMPLE:

DVBSHAP	4	1	1	1.6					

FIELDS:

Field	Contents
DVID	Design variable identification number of a DESVAR entry. (Integer>0)
AUXMID	Auxiliary model identification number. (Integer>0)
COLi	Load sequence identification number from AUXMODEL Case Control command. (Integer>0)
SFi	Scaling factor for load sequence identification number. (Real; Default = 1.0)

REMARKS:

1. Design variable DVID must be defined on a DESVAR entry.
2. Multiple references to the same DVID and/or COLi will result in the vector addition of the referenced boundary shape vectors.

3. Multiple DVBSHAP entries may be specified.

DVCREL1

Design Variable to Connectivity Property Relation

Defines the relation between a connectivity property and design variables.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVCREL1	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

EXAMPLE:

DVCREL1	5	CQUAD4	1	ZOFFS		1.0			
	1	1.0							

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of an element connectivity entry, such as “CBAR”, CQUAD4”, etc. (Character; case sensitive)
EID	Element Identification number. (Integer>0)
CPNAME	Name of connectivity property, such as “X1”, X2”, “X3”, “ZOFFS”, etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
CPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default =1.0E+20)

Field	Contents
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer>0)
COEFi	Coefficient of linear relation. (Real)

REMARKS:

1. The relationship between the connectivity property and design variables is given by:

$$CP_j = C_0 + \sum_i COEF_i \times DVID_i$$

2. The continuation entry is required.

3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc), ZOFFS (case insensitive) must be specified on the CPNAME field.

4. The CPMIN and CPMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVCREL2

Design Variable to Connectivity Property Relation

Defines the relation between a connectivity property and design variables with a user-supplied equation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVCREL2	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	EQID		
	“DESVAR”	DVID1	DVID2	DVID3	-etc.-				
	“DTABLE”	LABL1	LABL2	LABL3	-etc.-				

EXAMPLE:

DVCREL2	1	CBAR	100	X1	0.05	1.0	100		
	DESVAR	1001							
	DTABLE	X10							

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of an element connectivity entry, such as “CBAR”, CQUAD4”, etc. (Character)
EID	Element Identification number. (Integer>0)
CPNAME	Name of connectivity property, such “X1”, X2”, “X3”, “ZOFFS”, etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-20. Otherwise, it is -1.0E35. See Remark 4. (Real)

Field	Contents
CPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default =1.0E+20)
EQID	DEQATN entry identification number. (Integer>0)
“DESVAR”	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character. See Bulk Data Syntax Rules.)

REMARKS:

1. The variable identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where $N = m + n$) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.

2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.

3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc.), ZOFFS (case insensitive) must be specified on the CPNAME field.

4. The CPMIN and CPMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVEREL1

Automatic shell element thickness design variable

Automatic shell element thickness design variable.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVEREL1	ID	TYPE	GRPID	PNAME	PMIN	PMAX			
	DVID1	COEF1							

EXAMPLE:

DVEREL1	701	PSHELL	1001	T	0.1	10.0			
	4053	0.1							

The referenced GROUP bulk entry may look like the following.

```
...
GROUP, 1001, shell_thickness,+
    +, ELEM, 1001, THRU, 1871
    +, ELEM, 2001, THRU, 2589
    ...
```

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	The property type used by the elements selected on the associated GROUP entry. The software will auto-generate design variables for these elements. Currently only PSHELL is supported. This field is case sensitive. (Character; Default=PSHELL)

Field	Contents
GRPID	ID of a GROUP bulk entry which selects the shell elements CQUADR, CQUAD4, CQUAD8, CTRIAR, CTRIA3, CTRIA6 for which the software will auto-generate a design variable associated with the thickness. See the remarks and the example. (Integer>0)
PNAME	Property name or field position of the property entry. Currently, only "T" is supported. (Character or Integer; Default=T)
PMIN	Minimum value allowed for the associated properties. (Real; Default=1.0E-15)
PMAX	Maximum value allowed for the associated properties. (Real; Default =1.0E+20)
DVID	Identification number for the first auto-generated PSHELL, DESVAR, and DVPREL1 entry. When the default is used, the design variable IDs will be the same as the IDs of the associated shell elements. (Integer>0; Default=0)
COEF	The coefficient in the expression P = COEF*DV. (Real; Default =1)

REMARKS:

1. The GROUP entry referenced by the DVEREL1 entry must currently use TYPEi=ELEM. The software ignores any unsupported elements in the ELEM lists, and it ignores elements selected on a GROUP entry with TYPEi=PROP.

2. For each element selected by the DVEREL1 and associated GROUP entries, the software automatically associates a new design variable with the element thickness. This results in a separate independent thickness design variable for each element selected. The starting thickness value is determined as follows:

   - If the shell element definition in the input file includes corner thicknesses, the software will compute an average thickness for the element.

   - If corner thicknesses are not defined, the thickness on the original PSHELL in the input file is used.

3. The relationship between the connectivity property and design variables is given by:

$$P = COEF \times DVID$$

4. A GROUP ID may only be referenced by a single DVEREL1.

5. The ID of each auto-generated DESVAR entry must be unique with respect to any existing DESVAR or other DVEREL1 generated DESVAR. These IDs cannot duplicate any existing DVPREL1 or PSHELL IDs either. Therefore, care should be taken when defining the DVID1 field on the DVEREL1 entry.

6. If you request an .op2 file with the PARAM,POST,-1 or PARAM,POST-2 definitions, the final shell corner thicknesses are written to the OSHT1 datablock in the .op2 file at the end of the analysis. The thickness values written to the .op2 file represent the best design obtained during optimization. You can optionally display the thickness values in a post processor which supports the OSHT1 datablock.

7. The EDVOUT parameter is available on the DOPTPRM bulk entry to control what fraction of DVEREL1 generated design variables are printed. The printing of variables occurs in the Design Variable History at the end of the f06 file, as well as at requested cycles. The fraction you assign the EDVOUT parameter is applied separately to the increased and to the decreased DVEREL1 generated design variables. For example, when the default of 0.001 is used, both 0.1% of the most increased and 0.1% of the most decreased DVEREL1 generated design variables will be printed. A value of 1.0 requests to print all DVEREL1 generated design variables which have either increased or decreased.

   To determine the most increased and the most decreased thickness variables, the software evaluates the variables using their percentage change between the best and initial design cycles. For example, if the thickness of variable one decreases from 1.0 to 0.8 or 20%, and the thickness of variable two decreases from 3.0 to 2.55 or 15%, variable one will have priority for printing.

   Note that the design variables which are not generated by the DVEREL1 entry are all printed in full.

DVGRID

Design Variable to Grid Point Relation

Defines the relationship between design variables and grid point locations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVGRID	DVID	GID	CID	COEFF	N1	N2	N3		

EXAMPLE:

DVGRID	3	108	5	0.2	0.5	0.3	1.0		

FIELDS:

Field	Contents
DVID	DESVAR entry identification number. (Integer>0)
GID	Grid point (GRID) or geometric point (POINT) identification number. (Integer>0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
COEFF	Multiplier of the vector defined by Ni. (Real; Default = 0.0)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni ≠ 0.0)

REMARKS:

1. A CID of zero or blank (the default) references the basic coordinate system.

2. Multiple references to the same grid ID and design variable result in vectorial addition of the participation vectors defined by CID, COEFF, and Ni. There is no restriction on the number of DVGRID entries that may reference a given grid (GID) or design variable (DVID).

3. The coordinate update equation is given as

$$\{g\}_i - \{g\}_i^0 = \sum_i \mathrm{COEFF}_j \left(X_{\mathrm{DVIDj}} - X_{\mathrm{DVIDj}}^0 \right) \{N\}_j$$

where $\{g\}_i$ is the location of the *i-th* grid, $[g_x g_y g_z]^T$.

The vector $\{N\} = [N_x N_y N_z]^T$ is determined from CID and Ni. Note that it is a change in a design variable from its initial value X^0, and not the absolute value of the design variable itself, that represents a change in a grid point location, $\{g\}_i - \{g\}^0_i$.

4. The DVGRID entry defines the participation coefficients (basis vectors) of each design variable for each of the coordinates affected by the design process in the relationship

$$\{\Delta g\}_i = \sum_i \{T\}_{ij} \cdot \Delta X_j$$

5. DVGRID entries that reference grid points on MPCs or RSSCON entries produce incorrect sensitivities. Often the sensitivities are 0.0 which may result in a warning message indicating zero gradients which may be followed by UFM 6499. Other rigid elements produce correct results.

DVMREL1

Design Variable to Material Relation

Defines the relation between a material property and design variables.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVMREL1	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

EXAMPLE:

DVMREL1	5	MAT1	1	RHO	0.05	1.0			
	1	1.0							

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a material property entry, such as “MAT1”, MAT2”, etc. (Character)
MID	Material Identification number. (Integer>0)
MPNAME	Name of material property, such “E”, “RHO”, etc. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
MPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real, Default = 0.0)

Field	Contents
DVIDi	DESVAR entry identification number. (Integer>0)
COEFi	Coefficient of linear relation. (Real)

REMARKS:

1. The relationship between the material property and design variables is given by:

$$MP_i = C_0 + \sum_i COEF_i \cdot DVID_i$$

2. The continuation entry is required.

3. The fifth field of the entry, MPNAME, only accepts string characters. It must refer to the same name as appears in the "Bulk Data Entries" for various material properties. For example, if the isotropic material density is to be design, RHO (case insensitive) must be specified on the MPNAME field.

4. The MPMIN and MPMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVMREL2

Design Variable to Material Relation

Defines the relation between a material property and design variables with a user-supplied equation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVMREL2	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	EQID		
	“DESVAR”	DVID1	DVID2	DVID3	-etc.-				
	“DTABLE”	LABL1	LABL2	LABL3	-etc.-				

EXAMPLE:

DVMREL2	5	MAT1	1	E	0.05	1.0	100		
	DESVAR	1	2						
	DTABLE	E0							

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a material property entry, such as “MAT1”, MAT2”, etc. (Character)
MID	Material Identification number. (Integer>0)
MPNAME	Name of material property, such “E”, “RHO”, etc. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-20. Otherwise, it is -1.0E35. See Remark 4. (Real)

Field	Contents
MPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default = 1.0E+20)
EQID	DEQATN entry identification number. (Integer>0)
“DESVAR”	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character. See Bulk Data Syntax Rules.)

REMARKS:

1. The variables identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where $N = m + n$) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.

2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.

3. The fifth field of the entry, MPNAME, only accepts string characters. It must refer to the same name as appears in the “Bulk Data Entries” for various material properties. For example, if Young’s modulus is to be defined as a function of design variables, E (case insensitive) must be specified in the MPNAME field.

4. The MPMIN and MPMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVPREL1

Design Variable to Property Relation

Defines the relation between an analysis model property and design variables.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	C0	FREQ	
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

EXAMPLES:

DVPREL1	12	PBAR	612	6	0.2	3.0			
	4	0.25	20	20.0	5	0.3			

DVPREL1	73	PELAST	319	TKID			0.725	3.5	
	18	0.375	28	1.52					

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a property entry, such as “PBAR”, “PBEAM”, etc. (Character)
PID	Property entry identification number. (Integer>0)
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. (Character or Integer ≠ 0)

Field	Contents
PMIN	Minimum value allowed for this property. If FID references a stress recovery location, then the default value for PMIN is -1.0 + 35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). See Remark 9. (Real; Default = 1.0E-15)
PMAX	Maximum value allowed for this property. See Remark 9. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
FREQ	Frequency at which a frequency dependent property given by PNAME/FID is defined. See Remark 8. (Real)
DVIDi	DESVAR entry identification number. (Integer>0)
COEFi	Coefficient of linear relation. (Real)

REMARKS:

1. The relationship between the analysis model property and design variables is given by:

$$P_i = C0 + \sum_i \mathrm{COEFi} \cdot \mathrm{DVIDi}$$

2. The continuation entry is required.

3. PTYPE = “PBEND” is not supported. PTYPE= “PBEAML” is not supported with the FID option.

4. FID may be either a positive or a negative number. However, for PTYPE=“PBEAM” or “PBUSH”, FID must be negative. If FID>0, it identifies the field position on a property entry. If FID<0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either FID = +4 or FID = -3 can be used. In general, use of PNAME is recommended.

5. Designing PBEAML or PBEAM requires specification of both property name and station. Table 13-11 shows several examples.

Table 13-11.

PTYPE	Property Name	END A	END B	i-th Station
PBEAML	DMI1	DIM1 or DIM1(A)	DIM1(B)	DIM1(i)
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, using an END B property name on a DVPREL1 entry when the referenced PBEAM is a constant section is not allowed.

6. The PWELD and PFAST property types are not supported.

7. The PPLANE property type is not supported.

8. FREQ (Field 9) is applicable only for TYPE=PBUSHT/PDAMPT/PELAST type properties. A FREQ value must currently match an analysis frequency, though there is no need to match a TABLED1 entry. Properties should be designed at a minimum of two frequencies for a TABLED1 referenced by a designed frequency dependent property PNAME/FID, because these tables are currently re-generated, not added to.

9. The PMIN and PMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVPREL2

Design Variable to Property Relation

Defines the relation between an analysis model property and design variables with a user-supplied equation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	EQID	FREQ	
	“DESVAR”	DVID1	DVID2	DVID3	-etc.-				
	“DTABLE”	LABL1	LABL2	LABL3	-etc.-				

EXAMPLES:

DVPREL2	13	PBAR	712	5	0.2	4			
	DESVAR	4	11	13	5				
	DTABLE	PI	YM						

DVPREL2	17	PBUSHT	122	TBID2	0.01	0.03	200	6.0	
	DESVAR	111	112	113	114				
	DTABLE	XYZ11	XYZ13	XYZ15					

FIELDS:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a property entry, such as PBAR, PBEAM, etc. (Character)
PID	Property entry identification number. (Integer>0)

Field	Contents
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). See Remark 10. (Real; Default = 1.E-20)
PMAX	Maximum value allowed for this property. See Remark 10. (Real; Default = 1.0E20)
EQID	DEQATN entry identification number. (Integer>0)
FREQ	Frequency at which a frequency dependent property given by PNAME/FID is defined. See Remark 9. (Real)
“DESVAR”	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character. See “Bulk Data Syntax Rules”.)

REMARKS:

1. The variables identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where N = m+n) are assigned in the order DVID1, DVID2, ..., DVIDn, LABL1, LABL2, ..., LABLm.

2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.

3. FID may be either a positive or a negative number. However, for PTYPE=“PBEAM” or “PBUSH”, FID must be negative. If FID>0, it identifies the field position on a

property entry. If FID<0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended.

4. PTYPE = “PBEND” is not supported.

5. Designing PBEAM requires specification of both property name and station. Table 13-12 shows one example.

Table 13-12.

PTYPE	Property Name	END A	END B	i-th Station
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, using an END B property name on a DVPREL2 entry when the referenced PBEAM is a constant section is not allowed.

6. PWELD and PFAST property types are not supported.

7. PBARL and PBEAML property types are not supported.

8. The PPLANE property type is not supported.

9. FREQ (Field 9) is applicable only for TYPE=PBUSHT/PDAMPT/PELAST type properties. A FREQ value must currently match an analysis frequency, though there is no need to match a TABLED1 entry. Properties should be designed at a minimum of two frequencies for a TABLED1 referenced by a designed frequency dependent property PNAME/FID, because these tables are currently re-generated, not added to.

10. The PMIN and PMAX values may be violated at a given design cycle if there is other, more critical violation by response constraints. Since design variables may never violate their lower and upper bounds, it may be best, if possible and reasonable, to configure the property bounds accordingly, so that they are not violated even for infeasible designs.

DVSHAP

Design Variable to Basis Vector(s)

Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.

FORMAT:

1	2	3	4	5	6	7	8	9	10
DVSHAP	DVID	COL1	SF1	COL2	SF2	COL3	SF3		

EXAMPLE:

DVSHAP	2	1	2.0	4	1.0				

FIELDS:

Field	Contents
DVID	Design variable identification number on the DESVAR entry. (Integer>0)
COLi	Column number of the displacement matrix. See Remark 2 . (1 ≤ Integer ≤ maximum column number in the displacement matrix.)
SFi	Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)

REMARKS:

1. DVID must be defined on a DESVAR entry.
2. COLi must be a valid column number in the displacement matrix.

3. Multiple references to the same DVID and/or COLi will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the first column.

4. The displacement matrix must have been created by NX Nastran and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN    DISPMAT=' physical filename of MASTER DBset '
DBLOCATE    DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D) ,
            LOGICAL=DISPMAT
```

EBDADD

Element Birth/Death Set Combination (SOLs 601 and 701)

Defines an element birth/death set as a union of element birth/death sets defined on EBDSET entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EBDADD	BDID	BD1	BD2	-etc-					

EXAMPLE:

EBDADD	10	1	2	3					

FIELDS:

Field	Contents
BDID	Element birth/death set identification number. (Integer > 0)
BDi	Identification numbers of element birth/death sets defined via EBDSET entries. (Integer > 0)

REMARKS:

1. To include several element birth/death sets defined via EBDSET entries in a model, EBDADD must be used to combine the element birth/death sets. BDID in EBDADD is then selected with the Case Control command EBDSET.

2. BDi must be unique and may not be the identification of this or any other EBDADD entry.

EBDSET

Element Birth/Death Set Definition (SOLs 601 and 701)

Defines element birth and death times for a set of elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EBDSET	BDID	TBIRTH	TDEATH	EID1	EID2	EID3	EID4	EID5	

CONTINUATION FORMAT 1:

	EID6	EID7	EID8	-etc-					

CONTINUATION FORMAT 2 (“THRU” OPTION IS ONLY AVAILABLE ON A CONTINUATION LINE):

	EID6	“THRU”	EID7	“BY”	INC				

EXAMPLE:

EBDSET	3	0.2	0.5	101	201	300	400	450	
	25	THRU	33						

FIELDS:

Field	Contents
BDID	Element birth/death set identification number. See Remark 2. (Integer > 0)
TBIRTH	Element birth time. (Real ≥ 0.0; Default = 0.0)

Field	Contents
TDEATH	Element death time. (Real > TBIRTH; Default = 1.0E+20)
EIDi	Element identification numbers. See Remark 3. (Integer > 0)

REMARKS:

1. The continuation line is optional.
2. The “THRU” option is only available on a continuation line.
3. BDID may be selected by Case Control command EBDSET. If other element birth/death sets are defined, the EBDADD entry must be used to combine all the EBDSET entries.
4. Element birth/death may be used with the elements CROD, CONROD, CBAR, CBEAM, CQUAD, CQUAD4, CQUAD8, CQUADR, CQUADX4, CQUADX8, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, CHEXA, CPENTA, CPYRAM, CTETRA, CELAS1, CELAS2, CDAMP1, CDAMP2, CMASS1, CMASS2, CGAP, or CBUSH1D.

ECHOOFF

Deactivate Printed Echo

Marks the point or points in the input file to deactivate printed echo of the Bulk Data.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ECHOOFF									

EXAMPLE:

ECHOOFF									

REMARKS:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the “Case Control Commands”.

2. The companion to this entry is the ECHOON entry.

ECHOON

Activate Printed Echo

Marks the point or points in the input file to activate printed echo of the Bulk Data.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ECHOON									

EXAMPLE:

ECHOON									

REMARKS:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in “Case Control Commands”.

2. The companion to this entry is the ECHOOFF entry.

EIGB

Buckling Analysis Set

Defines data needed to perform buckling analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP	NDP	NDN		
	NORM	G	C						

EXAMPLE:

EIGB	13	SINV	0.1	2.5	2	1	1		
	MAX								

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character: “SINV”)
	SINV Enhanced Inverse Power method.
L1, L2	Eigenvalue range of interest. (Real L1 < L2)
NEP	Estimated number of eigenvalues in positive range. Not used if METHOD = “SINV”. (Integer > 0)
NDP, NDN	Desired number of positive and negative eigenvalues. (Integer > 0; Default = 3*NEP)
NORM	Method for normalizing eigenvectors. (Character: “MAX” or “POINT”; Default = “MAX”)

Field	Contents	
	MAX	Normalize eigenvectors by scaling the largest component in the analysis set to unity.
	POINT	Normalize eigenvectors by scaling the component defined in the G and C fields to unity. If the value for the component defined in the G and C fields is zero, it cannot be scaled and NORM defaults to MAX.
G	Grid or scalar point identification number. Required only if NORM = “POINT”, blank otherwise. (Integer > 0 or blank; No default)	
C	Component number. Required only if NORM = “POINT” and G is a geometric grid point, blank otherwise. (1 ≤ Integer ≤ 6 or blank; No default)	

REMARKS:

1. The EIGB entry must be selected by a METHOD case control command that references the SID of the EIGB entry.
2. Each eigenvalue is the factor by which the pre-buckled state of stress is multiplied to produce the buckled shape described by the corresponding eigenvector.
3. The continuation entry is optional. If the continuation is not specified, than NORM = “MAX” normalization is performed.
4. See the *NX Nastran Theoretical Manual* for a discussion of convergence criteria.
5. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
6. The SINV method uses Sturm sequence techniques to ensure that all eigenvalues in the range have been found.
7. The convergence tolerance is 10^{-6}.

EIGC

Complex Eigenvalue Extraction Data

Defines data needed to perform complex eigenvalue analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD				EPS	ND0		

Continuation lines are used to define search regions or shift points for the Inverse Power (INV) method, Complex Lanczos (CLAN) methods, and Iterative Schur-Rayleigh-Ritz (ISRR) method. A maximum of 10 continuation lines can be specified. Continuation lines are not recommended or required for the Hessenberg (HESS) methods.

CONTINUATION LINE FORMAT FOR INVERSE POWER METHOD:

1	2	3	4	5	6	7	8	9	10
	ALPHAAj	OMEGAAj	ALPHABj	OMEGABj	Lj	NEj	NDj		

CONTINUATION LINE FORMAT FOR COMPLEX LANCZOS METHODS:

1	2	3	4	5	6	7	8	9	10
	SHIFTRj	SHIFTIj	MBLKSZj	IBLKSZj	KSTEPSj		NDj		

CONTINUATION LINE FORMAT FOR ITERATIVE SCHUR-RAYLEIGH-RITZ METHOD:

1	2	3	4	5	6	7	8	9	10
	SHIFTR1	SHIFTI1				ISRRFLG	ND1		

EXAMPLES:

EIGC	14	HESS					6		

EIGC	15	INV							
			3.0	4.0	2.5	10			

EIGC	16	CLAN							
		+5.6					4		
		-5.5					3		

EIGC	17	ISRR							
	-1.0	0.0				1	10		

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer>0)
METHOD	Method of complex eigenvalue extraction. (Character: “HESS”, “INV”, “CLAN”, or “ISRR”, No default)
	HESS — Hessenberg methods. See Remark 1.
	INV — Inverse Power method. See Remark 1 and Remark 4.
	CLAN — Complex Lanczos methods. See Remark 1.
	ISRR — Iterative Schur-Rayleigh-Ritz method. See Remark 1.
EPS	Convergence tolerance. (Real ≥ 0.0) Default values are: = 10^{-15} if METHOD = “HESS” = 10^{-4} if METHOD = “INV” = 10^{-8} if METHOD = “CLAN” = 10^{-8} if METHOD = “ISRR”

Field	Contents
ND0	Number of eigenvalues and/or eigenvectors desired. See Remark 3. (Integer > 0 or blank; No default)

CONTINUATION LINE FIELDS FOR INVERSE POWER METHOD: (METHOD = “INV”)

Each continuation line defines a rectangular search region in the complex plane. Each rectangular search region is defined from end points A and B, and width L as indicated in Figure 13-2. Search regions can overlap. Eigenvalues in overlapping search regions will not be extracted more than once.

Field	Contents
ALPHAAj	Real part of end point A of the jth search region in rad/time. (Real; Default = 0.0)
OMEGAAj	Imaginary part of end point A of the jth search region in rad/time. (Real; Default = 0.0)
ALPHABj	Real part of end point B of the jth search region in rad/time. (Real; Default = 0.0)
OMEGABj	Imaginary part of end point B of the jth search region in rad/time. (Real; Default = 0.0)
Lj	Width of the jth search region in rad/time. (Real; Default = 1.0)
NEj	Estimated number of eigenvalues in the jth search region. (Integer > -1; Default = 0)
NDj	Desired number of eigenvalues and eigenvectors to extract from the jth search region. See Remark 3. (Integer > 0; Default = 3 x NEj)

CONTINUATION LINE FIELDS FOR COMPLEX LANCZOS METHODS: (METHOD = “CLAN”)

Each continuation line specifies a shift point and the number of eigenvalues and eigenvectors to compute from the shift. For Single Vector Complex Lanczos method the MBLKSZj and IBLKSZj fields are ignored.

Field	Contents
SHIFTRj	Real part of the jth shift point in rad/time. (Real; Default = 0.0)
SHIFTIj	Imaginary part of the jth shift point in rad/time. (Real; Default = 0.0)
MBLKSZj	Maximum block size for the jth shift point in rad/time. (Integer > 0; Default = 7)
IBLKSZj	Initial block size for the jth shift point in rad/time. (Integer > 0) Default values are: = 1 if N < 1000 = 2 if 1000 ≤ N < 50,000 = 4 if 50,000 ≤ N < 100,000 = 5 if N ≥ 100,000 where N is the linearized problem size. See Remark 10.
KSTEPSj	Frequency of solve for the jth shift point in rad/time. (Integer > 0; Default = 5)
NDj	Desired number of eigenvalues and eigenvectors to extract from the jth shift point. (Integer > 0; No default)

CONTINUATION LINE FIELDS FOR ITERATIVE SCHUR-RAYLEIGH-RITZ METHOD: (METHOD = “ISRR”)

A single continuation line is used to specify a shift point that the software uses to avoid singularity during decomposition and the number of eigenvectors to compute from the shift point.

Field	Contents
SHIFTR1	Real part of the shift point in rad/time if ISRRFLG = 1. Blank otherwise. (Real or blank)
SHIFTI1	Imaginary part of the shift point in rad/time if ISRRFLG = 1. Blank otherwise. (Real or blank)
ISRRFLG	ISRR method instruction flag. ISRRFLG values may be added to obtain a combination of settings. For example ISRRFLG = 323 would indicate options “1”, “2” and a maximum subspace of 10 vectors (1 + 2 + 10*32 = 323). (Integer)

Field	Contents	
	0 (Default)	All of the options below are disabled.
	1	Reserves fields 2 (SHIFTR1) and 3 (SHIFTI1) for shift point specification. The shift point does not redefine the search region, but is only used during decomposition to avoid a singularity. Specifying a shift point is recommended to obtain better performance.
	2	Forces the out-of-core path in the code.
	4	Overrides system cell 405.
	8	Forces balanced iteration for real unsymmetric problems only.
	16	Forces generation of starting vectors for quadratic problems from the values found for the linear case when damping is ignored. In all other cases, the starting vectors are randomly generated.
	M * 32	Forces the maximum size of the subspace to M vectors.
ND1	Desired number of eigenvectors to extract. (Integer > 0; No default)	

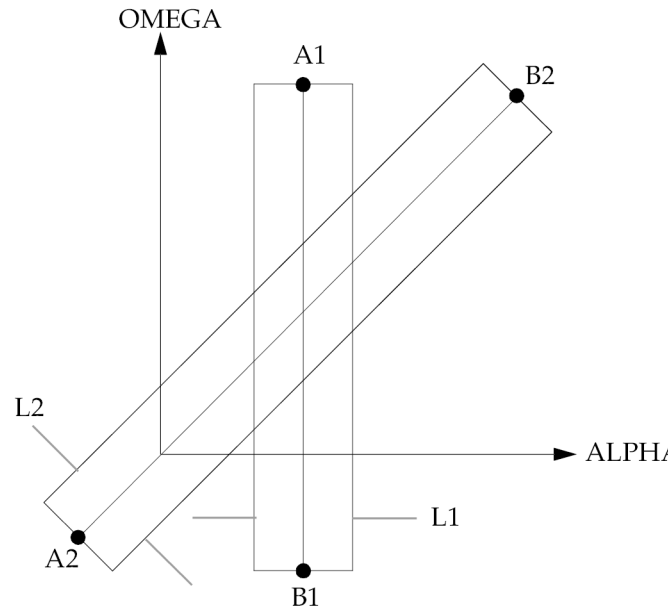


Figure 13-2. Sample Search Regions for Inverse Power Method

REMARKS:

1. A summary of the supported complex eigenvalue methods is provide in the following table.

Method	Description and Applicability
Hessenberg	The Hessenburg methods are generally reliable and economical for small to moderate-size problems. The Hessenberg methods compute all eigenvalues and ND eigenvectors. The QZ Hessenberg method is used by default. The applicability of the QR Hessenberg and LR Hessenberg methods has been superseded by the QZ Hessenberg method. However, the QR Hessenberg and LR Hessenberg methods can still be selected with system cell 108. See Remark 2.
Inverse Power	The applicability of the Inverse Power method has been superseded by the Block Complex Lanczos method.

Method	Description and Applicability
Complex Lanczos	The Block Complex Lanczos method is more reliable and will not accept inaccurate eigenvalues, which the Single Vector Complex Lanczos method has a tendency to do. Given the same input, the Block Complex Lanczos method may often accept fewer eigenvalues. The Block Complex Lanczos method is used by default. The applicability of the Single Vector Complex Lanczos method has been superseded by the Block Complex Lanczos method. However, the Single Vector Complex Lanczos method can still be selected with system cell 108. See Remark 2.
ISRR	The Iterative Schur-Rayleigh-Ritz method works well on sparse matrices, confines the search region to a circle centered on the origin of the complex plane, and provides some reliability that all modes within the circle have been found.

2. The EIGC entry must be selected in the case control section with the command CMETHOD = SID. The complex eigenvalue method that is used depends on the METHOD field of the EIGC entry, but, optionally, can be overridden by specifying system cell 108.

SYSTEM(108)	Specification(1)
0 (Default)	Use QZ Hessenberg method if METHOD = “HESS”, Inverse Power method if METHOD = “INV”, Block Complex Lanczos method if METHOD = “CLAN”, or Iterative Schur-Rayleigh-Ritz method if METHOD = “ISRR”
1	Force QR Hessenberg method (with spill, mass matrix must be non-singular)(2)
2	Force Single Vector Complex Lanczos method(3)
4	Force Block Complex Lanczos method
8	Debugging output for Lanczos methods
16	Turn off block reduction in Block Complex Lanczos method
32	Turn off block augmentation in Block Complex Lanczos method
64	Turn off full orthogonality in Block Complex Lanczos method
128	Turn off preprocessing of initial vectors in Block Complex Lanczos method

SYSTEM(108)	Specification(1)
256	Force LR Hessenberg method (without spill, mass matrix must be non-singular)(2)
512	Force QZ Hessenberg method
65536	Use semi-algebraic sort on imaginary part of roots
(1) In this table, “force” implies that the method selected by the system cell will be used even if another method has been selected on an EIGC entry. Sums of these values will produce two or more actions at once, when feasible. As the numbers get larger, the function is more developer-oriented than user-oriented.	
(2) The applicability of this method has been superseded by the QZ Hessenberg method.	
(3) The applicability of this method has been superseded by the Block Complex Lanczos method.	

3. For all methods, if no continuation lines are present, then ND0 must be specified and the software will calculate ND0 eigenvalues and/or eigenvectors near the origin of the complex plane. If continuation lines are present, it is recommended to leave the ND0 field blank. However, if continuation lines are present and a value is specified in the ND0 field, the value specified in the ND0 field will be ignored. Related method-specific considerations for METHOD = “HESS” and METHOD = “ISRR” are as follows:
   - If METHOD = “HESS” and continuation lines are present, the value specified in the ND1 field is the only value on the continuation lines that is used. All other values specified on the continuation lines are ignored.
   - If METHOD = “ISRR”, only the value in the ND1 field will be used.
4. See the *NX NASTRAN Theoretical Manual* for a discussion of convergence criteria and the search procedure for the Inverse Power method.
5. DIAG 12 prints diagnostics for the Inverse Power, Block Complex Lanczos, Single Vector Complex Lanczos, QZ Hessenberg, QR Hessenberg, LR Hessenberg, and ISRR methods.
6. The normalized eigenvectors may be output with the SDISPLACEMENT and/or DISPLACEMENT case control commands.
7. The eigenvectors are normalized by making the component with the largest magnitude a unit value for the real part and a zero value for the imaginary part. All eigenvectors are then scaled accordingly.
8. The SVD method is provided for DMAP applications. If used in SOL 107 or 110, and mass or damping terms are present, a user fatal exit is taken. The SVD

operation decomposes the input stiffness matrix K into the factors U, S, and V. U and V are collections of vectors of orthogonal functions. S is a rectangular matrix with terms on the diagonal of its left partition. The factors satisfy the equation K = U*S*V', where V' is the complex conjugate transpose of V. The ND1 value has a meaning for the SVD functions which differs from eigensolution.

ND1	OUTPUT
> 0	All vectors of U and V are output.
= 0	U and V are returned in a purged state.
< 0	S is returned as a square matrix whose number of columns is equal to the minimum number of rows or columns of the input matrix. U and V are truncated to be conformable with S. This reduces the computational effort required to solve very rectangular input matrices by providing a partial solution for the most interesting vectors.

9. For DMAP applications there are input parameters, not present in the solution sequences, that may be used to replace the function of the EIGC and CMETHOD entries.

10. The linearized problem size is either twice the dimension of K, M, and B if all three are present, or equal to the dimension of K, M, or B if only two of the three are present.

ALTERNATE FORMAT FOR COMPLEX LANCZOS METHODS: METHOD = "CLAN"

Each continuation line specifies a shift point and the number of eigenvalues and eigenvectors to compute from the shift. For Single Vector Complex Lanczos method the MBLKSZj and IBLKSZj fields are ignored.

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD				EPS	ND0		
	KEYWORD1=<value>, KEYWORD2=<value>, KEYWORD3=<value>, etc.								

Allowable keywords include:

Keyword	Contents
METHOD	Method of complex eigenvalue extraction. See Alternate Format Remark 2. (Character: “CLAN”; No default)
EPS	Convergence tolerance. See Alternate Format Remark 2. (Real ≥ 0.0; Default = 10^{-8})
SHIFTRj	Real part of the jth shift point in rad/time.(Real; Default = 0.0)
SHIFTIj	Imaginary part of the jth shift point in rad/time. (Real; Default = 0.0)
MBLKSZj	Maximum block size for the jth shift point in rad/time. (Integer > 0; Default = 7)
IBLKSZj	Initial block size for the jth shift point in rad/time. (Integer > 0) Default values are: = 1 if N < 1000 = 2 if 1000 ≤ N < 50,000 = 4 if 50,000 ≤ N < 100,000 = 5 if N ≥ 100,000 where N is the linearized problem size.
KSTEPSj	Frequency of solve for the jth shift point in rad/time. (Integer > 0; Default = 5)
NDj	Desired number of eigenvalues and eigenvectors to extract from the jth shift point. See Alternate Format Remark 2. (Integer > 0; No default)

ALTERNATE FORMAT EXAMPLES:

EIGC	1	CLAN							
	SHIFTR1=0.0, SHIFTI1=20.0, MBLKSZ1=5, IBLKSZ1=2, ND1=12, EPS=1.E-12								

EIGC	3	CLAN							
	SHIFTR1=5.0, SHIFTI1=10.0, ND1=5								
	SHIFTR2=0.0, SHIFTI2=20.0, ND2=8, MBLKSZ2=5, IBLKSZ2=4								

ALTERNATE FORMAT REMARKS:

1. The first field of the keyword-driven continuation entry must be blank.

2. If METHOD is specified on the continuation entry, the METHOD field on the original entry must be left blank. If EPS is specified on the continuation entry, the EPS field on the original entry must be left blank. If ND1 is specified on the continuation entry, the ND0 field on the original entry must be left blank.

EIGP

Poles in Complex Plane

Defines poles that are used in complex eigenvalue extraction by the Determinant method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGP	SID	ALPHA1	OMEGA1	M1	ALPHA2	OMEGA2	M2		

EXAMPLE:

EIGP	15	-5.2	0.0	2	6.3	5.5	3		

FIELDS:

Field	Contents
SID	Set identification number. (Integer>0)
ALPHAi, OMEGAi	Coordinates of point in complex plane. (Real)
Mi	Multiplicity of complex root at pole defined by point at ALPHAi and OMEGAi. (Integer>0)

REMARKS:

1. The EIGP entry defines poles in the complex plane that are used with an associated EIGC entry having the same set number.
2. The units of ALPHAi and OMEGAi are radians per unit time.
3. Poles are used only in the determinant method. (METHOD = “DET” on the EIGC entry).
4. One or two poles may be defined on a single entry.

5. See *The NASTRAN Theoretical Manual*.

EIGR

Real Eigenvalue Extraction Data

Defines data needed to perform real eigenvalue analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND			
	NORM	G	C						

EXAMPLE:

EIGR	13	LAN				12			

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer>0)
METHOD	Method of eigenvalue extraction.(Character)
	Modern Methods:
	LAN Lanczos Method
	AHOU Automatic selection of HOU or MHOU method. See Remark 13.
	Obsolete Methods:
	INV Inverse Power method.
	SINV Inverse Power method with enhancements.

Field	Contents	
	GIV	Givens method of tridiagonalization.
	MGIV	Modified Givens method.
	HOU	Householder method of tridiagonalization.
	MHOU	Modified Householder method.
	AGIV	Automatic selection of METHOD = “GIV” or “MGIV”. See Remark 13.
NORM	Method for normalizing eigenvectors. (Character: “MASS,” “MAX,” “AF”, or “POINT”; Default = “MASS”)	
	MASS	Normalize to unit value of the generalized mass. (Default)
	MAX	Normalize to unit value of the largest component in the analysis set.
	AF	Amplitude-Frequency normalization method. See Remark 20. Supported when METHOD=LAN or AHOU.
	POINT	Normalize to a positive or negative unit value of the component defined in fields 3 and 4. POINT is not supported when METH=LAN. (Defaults to “MASS” if defined component is zero).
G	Grid or scalar point identification number. Required only if NORM = “POINT”. (Integer>0)	
C	Component number. Required only if NORM = “POINT” and G is a geometric grid point. (1 ≤ Integer ≤ 6)	

Table 13-13. Relationship Between METHOD Field and Other Fields for Obsolete Methods

Field	METHOD Field	
	INV or SINV	GIV, MGIV, HOU, or MHOU
F1, F2	Frequency range of interest. F1 must be input. If METHOD = “SINV” and ND, is blank, then F2 must be input. (Real ≥ 0.0)	Frequency range of interest. Eigenvectors are found with natural frequencies that lie in the range between F1 and F2. If ND is not blank, at most ND eigenvectors are found. (Real ≥ 0.0; F1<F2)
NE	Estimate of number of roots in range (Required for METHOD = “INV”). Not used by “SINV” method. (Integer>0)	Not used.
ND	Desired number of roots. If this field is blank and METHOD = “SINV”, then all roots between F1 and F2 are searched and the limit is 600 roots. (Integer>0, Default is 3 ≠ NE for METHOD = “INV” only.)	Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPORTi entries. (Integer ≥ 0; Default = 0)

REMARKS:

1. The EIGR entry must be selected with the Case Control command METHOD = SID.
2. See Eigenvalue Extraction Methods in the *NX Nastran Basic Dynamic Analysis User’s Guide* for a discussion of method selection.
3. The units of F1 and F2 are cycles per unit time.
4. The continuation entry is optional. If the continuation entry is not specified, then mass normalization is performed.
5. The contemporary methods are LAN and AHOU. The other methods are in a maintenance-only status, with no enhancements planned for them. They may be eliminated in a future release.
6. The LAN method is the most general-purpose method, and may be used on both small- and large-size problems. It takes advantage of sparsity of input matrices, leading to greater efficiency on large-size problems. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20

degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry. The NE, G, and C fields are ignored for the LAN method. The NORM field may be set to MASS (the default value) or NORM. The conventions used when both the Fi and ND fields are specified are described in Table 1 of the EIGRL entry description. The EIGRL entry is an alternate method to select the LAN method. It has several other input options for special cases. When both and EIGRL and EIGR have the same SID and that SID is selected by a METHOD command the EIGRL entry takes precedence.

7. The AHOU method is competitive with the LAN method when there are small, dense matrices and many eigenvectors are required. This most commonly occurs when static or dynamic reduction is performed. The AHOU method does not take advantage of matrix sparsity, so that computation cost rises with the cube of the number of DOFs. The AHOU method responds to all permitted values for all the other fields except NE, which is ignored.

8. All methods require a positive semi-definite (psd) mass matrix for stable solutions. The mass matrix may be tested for this condition for all methods of solution by setting SYSTEM(303). A value of “-4” should be sufficient to identify problem matrices. A fatal error exit is taken when it is not met. All NX Nastran metric elements are designed to produce psd mass matrices. CMASSi elements, DMIG matrices selected by the M2GG command, and matrices input via INPUTT4 are special methods that allow addition of non-psd terms by use of non-metric element input. If none of this type of special input is present and the fatal error exit is taken you may have encountered an error in a metric element. Contact a Technical Support representative for corrective action in this case.

9. The LAN and AHOU methods allow singular but psd mass matrices.

10. The tridiagonal methods include the xGIV and xHOU methods, where “x” is described in the following comments. All tridiagonal methods compute all eigenvalues, and the number of eigenvectors specified by the Fi and Nd fields, as described in Table 14-14 above.

11. If “x” is blank (for example, the HOU method is selected) the mass matrix must be non-singular.

12. If “x” is M (for example, the MHOU method is selected) the mass matrix may be singular. A modified, shifted problem is solved in an inverse basis with this method. Some precision in the solution and longer computation time is exchanged for a more stable solution.

13. If “x” is A (for example, the AHOU method is selected) an attempt is made to solve the problem without shifting, in the interest of cost reduction and maximum precision. If the mass matrix is determined to be poorly conditioned for inversion the basis is automatically shifted with the modified method.

14. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.

15. If NORM = “POINT”, the selected component should be in the analysis set (a-set). (The program uses NORM = “MAX” when it is not in the analysis set.) The displacement value at the selected component will be positive or negative unity.

16. The “SINV” method is an enhanced version of the “INV” method. It uses Sturm sequence number techniques to make it more likely that all roots in the range have been found. It is generally more reliable and more efficient than the “INV” method.

17. For the “INV” and “SINV” methods, convergence is achieved at 10^{-6}. Convergence is tested by other criteria for the other methods.

18. For the “SINV” method only, if F2 is blank, the first shift will be made at F1, and only one eigensolution above F1 will be calculated. If there are no modes below F1, it is likely that the first mode will be calculated. If there are modes below F1 (including rigid body modes defined by SUPORT entries), a mode higher than the first mode above F1 may be calculated.

19. When F1, F2, and ND are all zero or blank, ND is reset to 1. A User Warning Message is produced for this condition, which is interpreted as likely to be due to an inadvertent omission by the user.

20. In AF (amplitude-frequency) normalization, the normalized eigenvectors are given by:

    $(C / \omega \Delta_{max}) \{x\}$

    where C is an optional user-defined scale factor, ω is the natural frequency for the mode in Hz, Δ_{max} is the magnitude of the maximum grid point translation for the mode, and $\{x\}$ is the un-normalized eigenvector for the mode. The optional user-defined scale factor, C, is specified with the AFNORM parameter.

EIGRL

Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCL	NORM	
	option_1 = value_1 option_2 = value_2, etc.								

EXAMPLE:

EIGRL	1	0.1	3.2	10					
	NORM=MAX NUMS=2								

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer>0)
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remarks 4 and 21. (Real or blank, $-5. \times 10^{16} \le V1 < V2 \le 5. \times 10^{16}$)
ND	Number of roots desired. See Remark 4. (Integer>0 or blank)
MSGLVL	Diagnostic level. ($0 \le$ Integer ≤ 4; Default = 0)
MAXSET	Number of vectors in block or set. Default is machine dependent. See Remark 14.
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark 10. (Real or blank)

Field	Contents
NORM	Method for normalizing eigenvectors (Character: “MASS”, “MAX”, or “AF”)
	MASS — Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)
	MAX — Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)
	AF — Amplitude-Frequency normalization method. See Remark 20.
ALPH	Specifies a constant for the calculation of the frequencies (Fi) at the upper boundary segments for the fdmodes and hdmodes parallel methods or the multisegment serial method based on the following formula: (0.0 < Real ≤ 1.0; Default = 1.0).

ALPH < 1.0,

$$Fi = V1 + (V2 - V1)\frac{1-\mathrm{ALPH}^{i}}{1. - \mathrm{ALPH}^{\mathrm{N}}}$$

ALPH = 1.0,

$$Fi = V1 + (V2 - V1)\left(\frac{i}{\mathrm{N}}\right)$$

The number of frequency segments N is defined as follows:

FDMODES: N is defined either by the NUMSEG keyword or the NASTRAN NUMSEG statement and is limited to 16. See Remark 13.

HDMODES: N is defined by the NCLUST keyword. See Remark 13.

Multisegment serial method: N is defined either by the NUMS field on the EIGRL entry or the NUMSEG keyword. See Remark 19.

Field	Contents
Fi	Frequencies at the upper boundary of each i-th segment for the fdmodes parallel method or the multisegment serial method. Fi is optionally used instead of ALPH to explicitly define the boundaries. Fi cannot be used with hdmodes. The number of Fi boundaries = N – 1 where the number of segments N is defined above in the description of ALPH. (Real or blank; V1 < F1 < F2 <...< F15 < V2)
NUMS	Number of frequency segments for the multisegment serial method. (0 < Integer ≤ 16; Default = 1) See Remark 19.
option_i= value_I	Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGLVL, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry.

REMARKS:

1. Real eigenvalue extraction data sets must be selected with the Case Control command METHOD = SID.

2. The units of V1 and V2 are cycles per unit time in vibration analysis, and are eigenvalues in buckling analysis. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.

3. NORM = “MASS” is ignored in buckling analysis and NORM = “MAX” will be applied.

4. The roots are found in order of increasing magnitude; that is, those closest to zero are found first. V2 should be set to a physically meaningful upper bound on the desired eigenvalues, or omitted. Extremely large values of V2 may result in a fatal message, depending on the model.

   If ND is specified, it is possible that the requested eigenvalues include some but not all from a set of multiple eigenvalues (for example, if ND=3 and there are six rigid modes). In this case, the modal space spanned by the eigenvectors is not uniquely defined, which is undesirable in frequency or transient response. In this situation, the eigensolution is computed as requested, accompanied by a warning message. Alternatively, setting system(509)=1 allows the READ module to increase the number of computed eigenpairs automatically to include the entire set.

The number and type of roots to be found can be determined from Table 13-14.

Table 13-14. Number and Type of Roots Found with EIGRL Entry

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND or all in range, whichever is smaller.
V1	V2	blank	All in range
V1	blank	ND	Lowest ND in range [V1,+∞]
V1	blank	blank	Lowest root in range [V1,+∞]
blank	blank	ND	Lowest ND roots in [-∞,+∞]
blank	blank	blank	Lowest root. See Remark 11.
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

5. In vibration analysis, if V1<0.0, the negative eigenvalue range will be searched. (Eigenvalues are proportional to Vi squared; therefore, the negative sign would be lost.) This is a means for diagnosing improbable models. In buckling analysis, negative V1 and/or V2 require no special logic.

6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.

7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.

8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. It may also be reset if there is insufficient memory available. The default value is recommended.

9. In vibration analysis, if V1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If V1 is set to zero, negative eigenvalues are not calculated.

10. A specification for SHFSCL may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques

can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.

11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. However, this method will not output the additional roots.

12. NASTRAN SYSTEM(146) provides options for I/O in sparse method only:

Table 13-15. SYSTEM(146) Options

SYSTEM(146)	Description
2	Increase memory reserved for sparse method by approximately 100%.
3	Increase memory reserved for sparse method by approximately 300%.
4	Increase memory reserved for sparse method by approximately 400%.

13. For the fdmodes distributed parallel method, the frequency range between V1 and V2 is subdivided into segments and analyzed in parallel. V1 and V2 must be defined, and the number of frequency segments must be specified with the NUMSEG keyword or the NASTRAN NUMSEG statement. NUMSEG should equal the number of processors which is defined with the DMP keyword. If NUMSEG and DMP are not the same, DMP will overwrite the NUMSEG value. The upper frequencies of each segment may be generated automatically by ALPH or specified directly in Fi. If both are specified, then Fi takes precedence over ALPH as long as they are consistent. ALPH, if multiplied by 100, may also be specified on FRQSEQ keyword of the NASTRAN statement. For best performance, ND should not be used with fdmodes. The segment boundaries are distributed automatically in units of frequency, not cycles.

    For the gdmodes distributed parallel method, the geometry is partitioned into segments and analyzed in parallel. ND, V1 and V2 can all be used with gdmodes. On the rare occasion that the geometry partitioning fails, gdmodes will revert to the fdmodes method if the keyword gpart=1 (default).

    For the hdmodes distributed parallel method, the frequency range between V1 and V2 is subdivided into segments, as in fdmodes. Also as in fdmodes, V1 and V2 must be defined. However, the number of segments equals the value of the NCLUST keyword. If desired, the upper frequencies of each segment may be adjusted using ALPH; they cannot be specified directly in Fi. For best performance, ND should not be used with hdmodes.

The rdmodes distributed parallel method uses substructuring technology for very large scale normal modes problems. The rdmodes method generally computes fewer modes with lower accuracy compared to standard Lanczos solution in order to gain performance. The rdmodes method requires V1 and V2, but ND should not be used. This method is not supported in buckling solutions.

Table 13-16. EIGRL Input Summary for DMP Methods

	Modal Solution (SOL 103)	Buckling Solution (SOL 105)
fdmodes	V1 and V2 are required. ND is not supported.	V1 and V2 are required. ND is not supported.
gdmodes	ND, V1 and V2 can all be used. See Table 13-14.	ND, V1 and V2 can all be used. If ND is not specified, both V1 and V2 must be specified.
hdmodes	V1 and V2 are required. ND is not supported.	V1 and V2 are required. ND is not supported.
rdmodes	V1 and V2 are required. ND is not supported.	Not supported.

14. Increasing MAXSET may improve performance for large problems where a large number of eigenvalues are being found. The default is 7 on all machines. SYSTEM(263) may be set in an rcfile to effectively modify the default; however the setting on the EIGRL entry always takes precedence.

15. SYSTEM(196), keyword SCRSAVE, controls reuse of scratch files when segment logic is invoked. SYSTEM(196) is useful only when multiple frequency segments are requested on a Lanczos run. (Multiple frequency segments can be requested via the NUMS field in the EIGRL entry and by SYSTEM(197).) Each frequency segment requires a minimum of three scratch files. When multiple frequency segments are used on a single processor computer then each frequency segment is solved serially. In this case, it makes sense to let segment #2 use the scratch files which were used by segment #1 since work for segment #1 has been completed (otherwise it wouldn't be working on #2). Similarly, when work for segment #2 is finished, segment #3 should be able to use #2's scratch files. SYSTEM(196)=1 allows such file reuse and is considered a safe default on Version 70 and later systems.

16. The new buckling shift logic in Version 70.5 tends to shift to 1.0 first. The logic may have difficulty finding the lowest ND roots if a problem requests a small number of roots (ND) when there are thousands of roots below 1. In this case either the loading should be scaled, SHFSCL specified, or a smaller frequency range requested.

17. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the

method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry.

18. The parameter input VMOPT=2 should not be used with MAX normalization. VMOPT=2 will only yield correct results with MASS normalization.

19. The multisegment serial method subdivides the frequency range between V1 and V2 into segments, as in fdmodes. Unlike fdmodes, the multisegment method analyzes each segment in turn on one processor only. V1 and V2 must be defined, and ND must not be defined. The number of segments may be specified in NUMS, with the NUMSEG keyword, or with the NASTRAN NUMSEG statement. If both NUMS and NUMSEG are specified, NUMS takes precedence. Unlike the parallel methods, the multisegment serial method is not meant to decrease solution times. It is sometimes used to solve an eigenvalue solution which otherwise has numerical problems.

20. In AF (amplitude-frequency) normalization, the normalized eigenvectors are given by:

    $(C / \omega \Delta_{max}) \{x\}$

    where C is an optional user-defined scale factor, ω is the natural frequency for the mode in Hz, Δ_{max} is the magnitude of the maximum grid point translation for the mode, and $\{x\}$ is the un-normalized eigenvector for the mode. The optional user-defined scale factor, C, is specified with the AFNORM parameter.

21. Negative eigenvalues can occur in a SOL 103 analysis if the model contains sufficient preloading to cause buckling. If the V1 field is left blank, the software assumes that all of the eigenvalues are non-negative. Because of this, depending on the magnitude of a negative eigenvalue relative to shift scale, a negative eigenvalue may be filtered out or not computed.

22. In SOL 401, for the cyclic and Fourier subcase types, a unique modal solution occurs for the requested harmonic numbers. As a result of the inherent symmetry with these solutions, mode pairs exist for specific harmonic numbers. For a cyclic subcase, mode pairs exist for harmonic numbers 1 through N/2 -1. For the Fourier subcase, mode pairs exist for all harmonic numbers > 0. The software automatically outputs the mode pairs for these subcase types for the modes requested with the EIGRL entry.

    For example, if you request 10 modes on the EIGRL entry:

    For a cyclic subcase:

    For harmonic index 0 and N/2, 10 modes are computed.

    For harmonic numbers 1 through N/2 -1, 20 modes are computed (10 distinct modes).

    For a Fourier subcase:

For harmonic index 0, 10 modes are computed.

For harmonic index > 0, 20 modes are computed (10 distinct modes).

ELIST

Element List

Defines a list of structural elements for virtual fluid mass.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ELIST	LID	E1	E2	E3	E4	E5	E6	E7	
	E8	E9	E10	-etc.-					

EXAMPLE:

ELIST	3	51	-62	68	THRU	102	122		

FIELDS:

Field	Contents
LID	Identification number of list. (Integer>0)
Ei	Identification number of a structural element. See Remark 1 . for the meaning of the negative sign. The string “THRU” may be used to indicate that all existing elements between those referenced in the preceding and succeeding fields are in the list. (Integer ≠ 0 or “THRU”)

REMARKS:

1. If the ELIST entry is referenced by field 6 of an MFLUID entry, the wetted side of the element is determined by the presence or absence of a minus sign preceding the element’s ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element’s positive normal as determined by applying the right-hand rule to the sequence of its corner points. If the “THRU” option is used, then immediately preceding and succeeding elements must have the same sign.

2. Large open sections with “THRU” should be avoided.

3. The word “THRU” may not appear in fields 3 or 9 on the parent row, or in fields 2 or 9 in any continuation, i.e. no leading or trailing THRU is allowed on the parent entry or on any continuations.

ENDDATA

Bulk Data Delimiter

Designates the end of the Bulk Data Section.

FORMAT:

ENDDATA

REMARKS:

1. ENDDATA is optional.

EPOINT

Extra Point List

Defines extra points for use in dynamics problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

EXAMPLE:

EPOINT	3	18	1	4	16	2			

ALTERNATE FORMAT AND EXAMPLE:

EPOINT	ID1	“THRU”	ID2						
EPOINT	17	THRU	43						

FIELDS:

Field	Contents
IDi	Extra point identification number. (1000000>Integer>0; for “THRU” option, ID1<ID2).

REMARKS:

1. All extra point identification numbers must be unique with respect to all other structural, scalar, and fluid points for direct methods of solution. For modal methods, they must be larger than the number of eigenvectors retained for analysis.

2. EPOINT is used to define coordinates used in transfer function definitions (see the TF and DMIG entries).

3. If the alternate format is used, extra points ID1 through ID2 are also defined to be extra points.

4. See the *NX Nastran Advanced Dynamic Analysis User's Guide* for a discussion of extra points.

EXTRN

Partitioned External Superelement Connection

Defines a boundary connection for an external superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
EXTRN	GID1	C1	GID2	C2	GID3	C3	GID4	C4	
	-etc.-		GID6	“THRU”	GID7	C6	-etc.-		

EXAMPLE:

EXTRN	1001	123	1120	123456	1201	123			

FIELDS:

Field	Contents
GIDi	Grid identification number to which the exterior superelement matrices will be connected.
Ci	Component numbers. (Integer 0, blank, or 1 for scalar points; Integers 1 through 6 with no embedded blanks for grids.)

REMARKS:

1. EXTRN can only be specified in partitioned Bulk Data Sections and is ignored in the main Bulk Data Section.
2. Connection grids must be specified in the partitioned Bulk Data Section following BEGIN SUPER = SEID.
3. “THRU” may be specified only in fields 3, 5, or 7.

4. Pairs of blank fields may be entered to allow easier modification of the EXTRN entry.

Chapter 14: Bulk Data Entries F—L

FLFACT

Aerodynamic Physical Data

Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FLFACT	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

EXAMPLE:

FLFACT	97	.3	.7	3.5					

ALTERNATE FORMAT AND EXAMPLE:

FLFACT	SID	F1	“THRU”	FNF	NF	FMID			
FLFACT	201	.200	THRU	.100	11	.133333			

FIELDS:

Field	Contents
SID	Set identification number. (Unique Integer > 0)
Fi	Aerodynamic factor. (Real)
FNF	Final aerodynamic factor. (Real)
NF	Number of aerodynamic factors. (Integer > 0)
FMID	Intermediate aerodynamic factors. See Remark 4 . (Real)

REMARKS:

1. Only the factors selected by a FLUTTER entry will be used.

2. Embedded blank fields are not allowed in the first format above.

3. The factors must be specified in the order in which they are to be used within the looping of flutter analysis.

4. FMID must lie between F1 and FNF; otherwise, FMID will be set to (*F*1+*FNF*)/2. then

$$Fi = \frac{F1(FNF - FMID) + FNF(FNF - F1)(i-1)}{(FNF - FMID) + (FMID - F1)(i - 1)}$$

   where i = 1, 2, ..., NF

   The use of FMID (middle factor selection) allows unequal spacing of the factors.

$$FMID = \frac{2 \cdot F1 \cdot FNF}{F1 + FNF}$$

   gives equal values to increments of the reciprocal of Fi.

5. If method = PK and this entry specifies velocities, then the velocities must be non-zero. Input of negative values produces eigenvector results at a velocity equal to the positive value of the input. Input of positive values provide eigenvalues results without eigenvectors.

FLUTTER

Aerodynamic Flutter Data

Defines data needed to perform flutter analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FLUTTER	SID	METHOD	DENS	MACH	RFREQ	IMETH	NVALUE	EPS	

EXAMPLE:

FLUTTER	19	K	119	219	319	S	5	1.-4	

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
METHOD	Flutter analysis method. (Character: “K” for K-method, “PK” for PK method, “PKNL” for PK method with no looping, “KE” for the K-method restricted for efficiency.)
DENS	Identification number of an FLFACT entry specifying density ratios to be used in flutter analysis. (Integer > 0)
MACH	Identification number of an FLFACT entry specifying Mach numbers (*m*) to be used in flutter analysis. (Integer > 0)
RFREQ (or VEL)	Identification number of an FLFACT entry specifying reduced frequencies (*k*) to be used in flutter analysis; for the PK and PKNL method, the velocities FLFACT entry is specified in this field. (Integer > 0)
IMETH	Choice of interpolation method for aerodynamic matrix interpolation. Used in the “K” and “KE” methods only. See Remark 6 . (Character: “L” = linear, “S” = surface; Default = “L”.)

Field	Contents
NVALUE	Number of eigenvalues beginning with the first eigenvalue for output and plots. [Integer > 0; Default is the number of modal degrees-of-freedom (u_h)]
EPS	Convergence parameter for k. Used in the PK and PKNL methods only. See Remark 4 . (Real > 0.0; Default = 10^{-3})

REMARKS:

1. The FLUTTER entry must be selected with the Case Control command METHOD = SID.

2. The density is given by DENS · RHOREF, where RHOREF is the reference value specified on the AERO entry and DENS is the density ratio specified on the FLFACT entry.

3. The reduced frequency is given by $k = (REFC \cdot \omega/2 \cdot V)$, where REFC is given on the AERO entry, ω is the circular frequency, and V is the velocity. If k= 0.0, as specified on the FLFACT entry, then only the K-method may be specified and the Inverse Power method of eigenvalue extraction (INV on the EIGC entry) must be used. Aeroelastic divergence analysis is more appropriately performed using the PK-method or PKNL method.

4. An eigenvalue is accepted in the PK and PKNL methods when:

$$\left|k - k_{estimate}\right| < EPS \qquad \text{for } k_{estimate} < 1.0$$

$$\left|k - k_{estimate}\right| < EPS \cdot k_{estimate} \qquad \text{for } k_{estimate} \geq 1.0$$

5. When the PK or PKNL method is selected, physical displacements will only be generated for the velocities on the FLFACT that are specified as negative values of the requested velocity. Also, structural damping as specified on the GE field of MATi entries is ignored.

6. If IMETH = “L”, a linear interpolation is performed on reduced frequencies at the Mach numbers specified on the FLFACT entry using the MKAEROi entry Mach number that is closest to the FLFACT entry Mach number. For IMETH = “S”, a surface interpolation is performed across Mach numbers and reduced frequencies. For METHOD = “PK” or “PKNL”, linear interpolation is always performed.

7. For the “K”, “KE” and “PK” methods, all combinations of the FLFACT entry are analyzed. For the “PKNL” method, only ordered pairs are analyzed; i.e., (M_1, V_1), (M_2, V_2)...(M_n, V_n). For the PKNL method, equal number of densities, Mach numbers and velocities must be specified.

8. Only the “PK” and “PKNL” methods are supported for design sensitivity and optimization.

FORCE

Static Force

Defines a static concentrated force at a grid point by specifying a vector.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

EXAMPLE:

FORCE	2	5	6	2.9	0.0	1.0	0.0		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one Ni ≠ 0.0.)

REMARKS:

1. The static force applied to grid point G is defined by **f** = $F \cdot$ **N** where **N** is the vector with components N1, N2, and N3 defined in fields 6, 7 and 8. The magnitude of **f** is equal to F times the magnitude of **N**.

2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD = SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. A CID of zero or blank (the default) references the basic coordinate system.

4. For scalar points see SLOAD.

5. For interpretation of a FORCE entry in an axisymmetric analysis, see the listing for the axisymmetric element type.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply a force with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

2. To apply a time-dependent force, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

FORCE1

Static Force, Alternate Form 1

Defines a static concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

EXAMPLE:

FORCE1	6	13	-2.93	16	13				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident.)

REMARKS:

1. The static force applied to grid point G is defined by $\mathbf{f} = F \cdot \mathbf{n}$ where **n** is a unit vector parallel to a vector from G1 to G2.
2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be

referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The linear solution sequences are SOLs 103, 105, 107 to 112, 115 and 116. (See also the parameter FOLLOWK.) Follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

   For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. If G1=G or G2=G, the force is a follower force in large displacement analysis (i.e., PARAM,LGDISP,1). Otherwise, the direction of the force is defined by the original positions of G1 and G2 and does not change even for large displacement analysis.

2. To apply a force with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analysis.

3. To apply a time-dependent force, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

FORCE2

Static Force, Alternate Form 2

Defines a static concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

EXAMPLE:

FORCE2	6	13	-2.93	16	13	17	13		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident; G3 and G4 cannot be coincident.)

REMARKS:

1. The direction of the force is parallel to the cross product of vectors from G1 to G2 and G3 to G4.
2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be

referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The linear solution sequences are SOLs 103, 105, 107 to 112, 115 and 116. (See also the parameter FOLLOWK.) Follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

   For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. The direction of the force is defined by the original positions of G1 to G4 and does not change during the analysis.

2. To apply a force with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

3. To apply a time-dependent force, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

FREQ

Frequency List

Defines a set of frequencies to be used in the solution of frequency response problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc.-					

EXAMPLE:

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
Fi	Frequency value in units of cycles per unit time. (Real ≥ 0.0)

REMARKS:

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.

2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \mathrm{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|,$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ1

Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

EXAMPLE:

FREQ1	6	2.9	0.5	13					

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency in set. (Real ≥ 0.0)
DF	Frequency increment. (Real > 0.0)
NDF	Number of frequency increments. (Integer > 0; Default = 1)

REMARKS:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.

3. The frequencies defined by this entry are given by

$$f_i = F1 + DF \cdot (i-1)$$

where i = 1 to (NDF + 1).

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \mathrm{DFREQ} \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ2

Frequency List, Alternate Form 2

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

EXAMPLE:

FREQ2	6	1.0	8.0	6					

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency. (Real > 0.0)
F2	Last frequency. (Real > 0.0, F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0; Default = 1)

REMARKS:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.

3. The frequencies defined by this entry are given by

$$f_i = F1 \cdot e^{(i-1)d}$$

where

$$d = \frac{1}{NF}\ln\frac{F2}{F1}$$

and i = 1, 2, ..., (NF+1)

In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \mathrm{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right| \quad ,$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ3

Frequency List, Alternate 3

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			

EXAMPLE:

FREQ3	6	20.0	200.0	LINEAR	10	2.0			

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of modal frequency range in cycles per unit time. F1 is required. (Real ≥ 0.0 for TYPE = LINEAR and Real > 0.0 for TYPE = LOG)
F2	Upper bound of modal frequency range in cycles per unit time. If F2=F1, F1 is the only excitation frequency resulting from that FREQ3 entry. (Real > 0.0, F2 ≥ F1, Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = “LINEAR”)

Field	Contents
NEF	Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1, Default = 10)
CLUSTER	Specifies clustering of the excitation frequency near the end points of the range. See Remark 6 . (Real > 0.0; Default = 1.0)

REMARKS:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.

2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.

3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.

4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

5. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \mathrm{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|$$

   where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} an f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

6. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER > 1.0 provides closer spacing of excitation frequency towards the ends of the frequency range, while values of less than 1.0 provide closer spacing towards the center of the frequency range. For example, if the frequency range is between 10 and 20, NEF = 11, TYPE = “LINEAR”; then, the excitation frequencies for various values of CLUSTER would be as shown in Table 14-1.

$$\hat{f}_k = \frac{1}{2}(\hat{f}_1 + \hat{f}_2) + \frac{1}{2}(\hat{f}_2 - \hat{f}_1)|\xi|^{1/\mathrm{CLUSTER}} \cdot \mathrm{SIGN}(\xi)$$

where

ξ = −1 + 2(*k*−1)/ (NEF−1) is a parametric coordinate between -1 and 1

k = varies from 1 to NEF (*k* = 1, 2, ..., NEF)

$\hat{f}_1$ = is the lower limit of the frequency subrange

$\hat{f}_2$ = is the upper limit o the subrange

$\hat{f}_k$ = is the k-th excitation frequency

$\hat{f}$ = is the frequency, or the logarithm of the frequency, depending on the value specified for TYPE

Table 14-1. CLUSTER Usage Example

Excitation Frequency Number	ξ	CLUSTER				
		c=0.25	c-0.50	c-1.0	c-2.0	c-4.0
		Excitation Frequencies in Hertz				
1	-1.0	10.00	10.0	10.0	10.00	10.00
2	-0.8	12.95	11.8	11.0	10.53	10.27
3	-0.6	14.35	13.2	12.0	11.13	10.60
4	-0.4	14.87	14.2	13.0	11.84	11.02
5	-0.2	14.99	14.8	14.0	12.76	11.66
6	0.0	15.00	15.0	15.0	15.00	15.00
7	0.2	15.01	15.2	16.0	17.24	18.34
8	0.4	15.13	15.8	17.0	18.16	18.98
9	0.6	15.65	16.8	18.0	18.87	19.40
10	0.8	17.05	18.2	19.0	19.47	19.73
11	1.0	20.00	20.0	20.0	20.00	20.00

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.

8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ4

Frequency List, Alternate Form 4

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of “spread” around each natural frequency and the number of equally spaced excitation frequencies within the spread.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

EXAMPLE:

FREQ4	6	20.0	200.0	0.30	21				

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0, Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0, F2 > F1, Default = 1.0E20)
FSPD	Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0, Default = 0.10)
NFM	Number of evenly spaced frequencies per “spread” mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

REMARKS:

1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency-response solutions.

2. FREQ4 entries must be selected with the Case Control command FREQUENCY=SID.

3. There will be NFM excitation frequencies between (1−*FSPD*) f_N and (1+*FSPD*) f_N, for each natural frequency in the range F1 to F2.

4. In the example above there will be 21 equally spaced frequencies across a frequency band of 0.7 · f_N to 1.3 · f_N for each natural frequency that occurs between 20 and 2000. See Figure 14-1 for the definition of frequency spread.

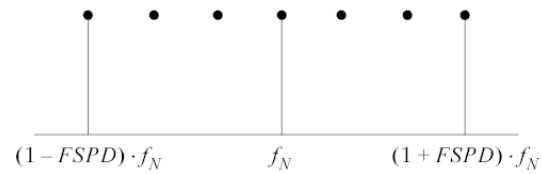


Figure 14-1. Frequency Spread Definition

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by 2 · ξ · f_N, where ξ is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See Figure 14-2 for the definition of half-power bandwidth.

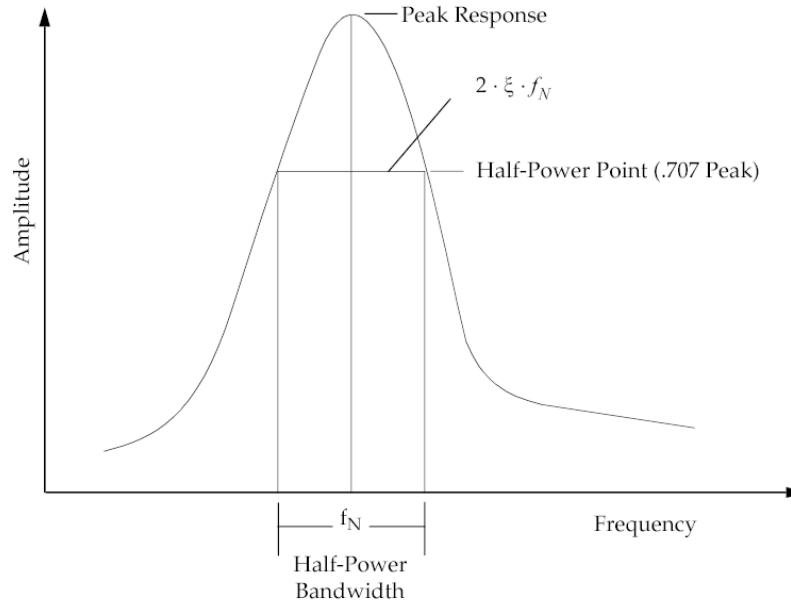


Figure 14-2. Half-Power Bandwidth Definition

6. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

7. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

   where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.

9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ5

Frequency List, Alternate Form 5

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

FORMAT:

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						

EXAMPLE:

FREQ5	6	20.0	200.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0, F2 > F1, Default = 1.0E20)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

REMARKS:

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.

2. FREQ5 entries must be selected with the Case Control command FREQUENCY=SID.

3. The frequencies defined by this entry are given by

$$f_i = FRi \cdot f_{N_i}$$

   where f_{Ni} are the natural frequencies in the range F1 through F2.

4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less then F1 and greater than F2, those computed excitation frequencies are ignored.

   Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \mathrm{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|$$

   where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.

8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

GENEL

General Element

Defines a general element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	-etc.-				

UIm – The last item in the UI list will appear in one of fields 2, 4, 6, or 8.

	“UD”		UD1	CD1	UD2	CD2	-etc.-		

UDn –The last item in the UD list will appear in one of fields 2, 4, 6, or 8.

	“K” or “Z”	KZ11	KZ21	KZ31	-etc.-	KZ22	KZ32		
	-etc.-		KZ33	KZ43	-etc.-				

KZmm – The last item in the K or Z matrix will appear in one of fields 2 through 9.

	“S”	S11	S12	-etc.-		S21	-etc.-		

Smn – The last item in the S matrix will appear in one of fields 2 through 9.

EXAMPLE:

GENEL	629		1	1	13	4	42	0	
	24	2							
	UD		6	2	33	0			
	Z	1.0	2.0	3.0	4.0	5.0	6.0	7.0	
	8.0	9.0	10.0						
	S	1.5	2.5	3.5	4.5	5.5	6.5	7.5	
	8.5								

FIELDS:

Field	Contents
EID	Unique element identification number. (Integer > 0)
UIi, CIi, UDj, CDj	Identification numbers of coordinates in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. UIi and UDi are grid point numbers, and CIi and CDj are the component numbers. If a scalar point is given, the component number is zero. (Integer ≥ 0)
KZij	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list. (Real)
Sij	Values of the [S] matrix ordered by rows according to the UD list. (Real)
“UD”, “K”, “Z”, and “S”	Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices.

REMARKS:

1. The stiffness approach:

$$\left\{\begin{array}{c} f_1 \\ \hline f_d \end{array}\right\} = \left[\begin{array}{c|c} K & -KS \\ \hline -S^T K & S^T K S \end{array}\right] \left\{\begin{array}{c} u_i \\ \hline u_d \end{array}\right\}$$

The flexibility approach:

$$\left[\begin{array}{c} u_i \\ \hline f_d \end{array}\right] = \left[\begin{array}{c|c} Z & S \\ \hline -S^T & O \end{array}\right] \left[\begin{array}{c} f_i \\ \hline u_d \end{array}\right] \left[\begin{array}{c} u_i \\ \hline f_d \end{array}\right] = \left[\begin{array}{c|c} Z & S \\ \hline -S^T & O \end{array}\right] \left[\begin{array}{c} f_i \\ \hline u_d \end{array}\right] \left[\begin{array}{c} u_i \\ \hline f_d \end{array}\right] = \left[\begin{array}{c|c} Z & S \\ \hline -S^T & O \end{array}\right] \left[\begin{array}{c} f_i \\ \hline u_d \end{array}\right]$$

where

$\{u_i\} = [u_{i1}, u_{i2}, ..., u_{im}]^T$

$\{u_d\} = [u_{d1}, u_{d2}, ..., u_{dn}]^T$

$$[KZ] = [K] \text{ or } [Z] = \begin{bmatrix} KZ11 & \dots & \dots & \dots \\ KZ21 & KZ22 & \dots & \dots \\ KZ31 & KZ32 & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ]$$

$$[S] = \begin{bmatrix} S11 & \dots & S_{1n} \\ S21 & \dots & \dots \\ S31 & \dots & \dots \\ \vdots & \vdots & \vdots \\ S_{m1} & \dots & S_{mn} \end{bmatrix}$$

The required input is the $\{u_i\}$ list and the lower triangular portion of [*K*] or [*Z*]. Additional input may include the $\{u_d\}$ list and [*S*]. If [*S*] is input, $\{u^d\}$ must also be input. If $\{u_d\}$ is input but [*S*] is omitted, [*S*] is internally calculated. In this case, $\{u_d\}$ must contain six and only six degrees-of-freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body with rigid body motions that are defined by $\{u_i\} = [S]\{u_d\}$. See “General Element Capability (GENEL)” in the *NX Nastran Element Library*.

2. When the stiffness matrix K is input, the number of significant digits should be the same for all terms.

3. Double-field format may be used for input of K or Z.

4. The DMIG entry or the INPUTT4 module offer alternative methods for inputting large matrices.

5. The general element entry in the example above defines the following:

   $[u_i] = [1–1, 13–4, 42, 24–2]^T$

   $\{u_d\} = [6–2, 33]^T$

   where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \quad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

GRAV

Acceleration or Gravity Load

Defines acceleration vectors for gravity or other acceleration loading.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

EXAMPLE:

GRAV	1	3	32.2	0.0	0.0	-1.0			

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 10 . (Integer; Default = 0)

REMARKS:

1. The acceleration vector is defined by **a** = $A \cdot$ **N** where **N** is the vector with components N1, N2, and N3. The magnitude of **a** is equal to *A* times the magnitude of **N**. The static loads generated by this entry are in the direction of **a**.

2. A CID of zero references the basic coordinate system.

3. Acceleration or gravity loads may be combined with “simple loads” (e.g. FORCE, MOMENT) only by specification on a LOAD entry. That is, the SID on a GRAV entry may not be the same as that on a simple load entry.

4. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD = SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

5. At most 100 GRAV entries can be selected in a given run either by Case Control or the LOAD Bulk Data entry. Multiples or reflections of a given acceleration or gravity load can be economically accomplished by use of the LOAD Bulk Data entry.

6. In cyclic symmetry solution sequences, the T3 axis of the coordinate system referenced in field 3 must be parallel to the axis of symmetry. In dihedral cyclic symmetry (where STYPE = “DIH” on the CYSYM entry), the T1 axis must, in addition, be parallel to Side 1 of segment 1R of the model.

7. For image superelements, the coordinate system must be rotated if the image is rotated relative to its primary superelement.

8. Acceleration or gravity loads do not include effects due to mass on scalar points.

9. The RFORCE entry may be used to specify rotational accelerations.

10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed. If a superelement defined by a partitioned Bulk Data Section is rotated using an SELOC bulk entry, the GRAV loads do not necessarily update on restart because nothing in the partitioned superelement has changed. To force the GRAV loads to update correctly they can be assigned new SID’s in the restart run.

11. When applying a load with a GRAV, ACCEL, or ACCEL1 entry to axisymmetric elements, any component in the radial direction is treated as a radial acceleration load.

12. The software treats cylindrical and spherical coordinate systems selected in the CID field on the GRAV, ACCEL, or ACCEL1 entries as a Cartesian coordinate system. For example, if you define an acceleration vector as 1.0, 0.0, 0.0 and a cylindrical or spherical coordinate system is selected in the CID field, the software will not vary the load vector radially at the grid points.

REMARKS RELATED TO SOLS 601 AND 701:

1. MB is ignored.
2. To apply an acceleration or gravity load with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.
3. To apply a time-dependent acceleration or gravity load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

GRDSET

GRID Entry Defaults

Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS	SEID	

EXAMPLE:

GRDSET		16				32	3456		

FIELDS:

Field	Contents
CP	Identification number of coordinate system in which the location of the grid points are defined. (Integer ≥ 0 or blank)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors of the grid point are defined. (Integer ≥ 0 or blank)
PS	Permanent single-point constraints on the grid point. (Any combination of Integers 1 through 6 with no embedded blanks, or blank.)
SEID	Superelement identification number. (Integer ≥ 0 or blank)

REMARKS:

1. The contents of fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, and 9 are blank. If any of these

fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired, one of the coordinate systems is basic, or the grid is assigned to the residual structure then the default may be overridden on the GRID entry by making one of fields 3, 7, 8, or 9 zero (rather than blank). Only one GRDSET entry may appear in the Bulk Data Section.

2. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).

3. At least one of the fields CP, CD, PS, or SEID must be specified.

GRID

Grid Point

Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

EXAMPLE:

GRID	2	3	1.0	-2.0	3.0		316		

FIELDS:

Field	Contents
ID	Grid point identification number. (0 < Integer)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank*)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ -1 or blank *)
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank*.)
SEID	Superelement identification number. (Integer ≥ 0; Default = 0)

* See the GRDSET entry for default options for the CP, CD, PS, and SEID fields.

REMARKS:

1. All grid point identification numbers must be unique with respect to all other structural, scalar, and fluid points.

2. The meaning of X1, X2, and X3 depends on the type of coordinate system CP as follows (see the CORDij entry descriptions):

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ(degrees)	Z
Spherical	R	θ(degrees)	φ(degrees)

   See “Overview of Coordinate Systems in NX Nastran” in the *NX Nastran User’s Guide* for a definition of coordinate system terminology.

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system. See the chapter “Understanding Coordinate Systems” in the *NX Nastran User's Guide* for more details.

4. The SEID field can be overridden by use of the SESET entry.

5. If CD = -1, then this defines a fluid grid point in coupled fluid-structural analysis (see Defining the Fluid in the *NX Nastran Acoustics User's Guide*). This type of point may only connect the CAABSF, CHACBR, CHACAB, CHEXA, CPENTA, and CTETRA elements to define fluid elements.

6. A zero (or blank if the GRDSET entry is not specified) in the CP and CD fields refers to the basic coordinate system.

REMARKS RELATED TO SOLS 601 AND 701:

1. SEID is ignored.

2. The use of GRDSET entry for default options for the CP, CD, and PS fields is not supported.

GROUP

Group definition

Defines a group of grid points, elements, or physical properties.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GROUP	ID	Descriptor							
	META	Meta Information							
	TYPEi	List of IDs							

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
GROUP	10	Assembly AA4							
	META	100 RPM							
	META	Optionally continue the meta data							
	GRID	1	2	3	4	5	6	7	
		8							
	GRID	10	THRU	20					
	GRID	100	THRU	200					
	GRID	341	THRU	360	BY	2			
	ELEM	30	THRU	40					
	PROP	ALL							

FIELDS:

Field	Contents
ID	Group identification number. (0 < Integer)

Field	Contents
META	Optional character data to store with the group definition. META can be continued on multiple continuation lines. The META keyword in the 2nd field must appear on every line where metadata is entered. (Character)
TYPEi	Designates the type of IDs defined in a specific row, or in a set of rows. Multiple TYPE can exist on the same GROUP entry. (Character) ”GRID” Designates the row includes grid point IDs. ”ELEM” Designates the row includes element IDs. ”PROP” Designates the row includes property IDs.

REMARKS:

1. Because of the descriptor field, free field data format is not supported for this bulk entry. For large field fixed format, the maximum number of characters is reduced to 48.

REMARKS RELATED TO SOLS 601 AND 701:

1. TYPE = GRID is not supported.

GUST

Aerodynamic Gust Load Description

Defines a stationary vertical gust for use in aeroelastic response analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
GUST	SID	DLOAD	WG	X0	V				

EXAMPLE:

GUST	133	61	1.0	0.	1.+4				

FIELDS:

Field	Contents
SID	Gust set identification number. (Integer > 0)
DLOAD	Set identification number of a TLOADi or RLOADi entry that defines the time or frequency dependence. (Integer > 0)
WG	Scale factor (gust velocity/forward velocity) for gust velocity. (Real ≠ 0.0)
X0	Streamwise location in the aerodynamic coordinate system of the gust reference point. (Real)
V	Velocity of vehicle. See Remark 5 . (Real > 0.0)

REMARKS:

1. The GUST entry must be selected with the Case Control command GUST = SID.

2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is

$$WG \cdot T\left(t - \frac{X - X0}{V}\right)$$

where *T* is the tabular function.

3. In random analysis, a unit gust velocity (WG = 1/velocity) is suggested. The actual rms value is entered on the TABRNDG entry.

4. X0 and V may not change between subcases under one execution.

5. V must be equal to VELOCITY on the AERO Bulk Data entry.

INCLUDE

Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

FORMAT:

INCLUDE ’filename’

DESCRIBER:

filename	External file to be inserted. The ‘*directory_path/filename*’ must begin and end with the single quote character.

EXAMPLE:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE ‘MYBULK.DATA’
ENDDATA
```

REMARKS:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.

2. Each line has a 72 character limit. Multiple lines can be used when file names are long. The entire string must begin and end with the single quote character.

   For example,

   D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\folder8\folder9\mydata.dat

can be defined on multiple lines:

```
INCLUDE 'D:\folder1\folder2\folder3\folder4\folder5\folder6\folder7\fold
er8\folder9\mydata.dat'
```

The following input format is also supported:

```
INCLUDE 'D:\folder1\folder2\folder3\
         folder4\folder5\
     folder6\folder7\
        folder8\folder9\mydata.dat'
```

INITADD

Combines multiple initial stress or initial strain sets in SOL 401.

Combines multiple initial stress or initial strain sets in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
INITADD	SID								
	SID1	SID2	SID3	SID4	SID5	SID6	SID7	SID8	
	SID9	SID10	...						

EXAMPLE:

INITADD	80								
	100	101	102	103	104	105	106	107	
	108								

FIELDS:

Field	Contents
SID	Set identification number for INITADD bulk entry. (Integer > 0)
SIDi	Set identification number of INITS bulk entries. (Integer > 0)

REMARKS:

1. The ID of all INITS and INITADD entries must be unique.
2. The INITS case control must be defined above the subcases (globally). If you want to include the effects of initial stress or strain in a modal subcase, you should include a static subcase before the sequentially dependent modal subcase.

3. The INITS entries selected by the INITADD entry must be all TYPE=STRESS or all TYPE=STRAIN. You cannot mix initial stress and initial strain definitions in the same input file.

4. If data is defined on the same grid or element corner location, a fatal error will occur.

INITS

Defines initial stress or strain state in SOL 401.

Defines initial stress or strain state in SOL 401.

FORMAT OF INITIAL STRESS DEFINED ON GRID POINTS (TYPE=STRESS AND LOC=GRID):

1	2	3	4	5	6	7	8	9	10
INITS	SID	STRESS	GRID	CSYS					
	GRID ID	Sxx	Syy	Szz	Sxy	Syz	Szx		
	GRID ID	Sxx	Syy	Szz	Sxy	Syz	Szx		
	...	...	...	...	...	...	...		

FORMAT OF INITIAL STRESS DEFINED AT ELEMENT GRID LOCATIONS (TYPE=STRESS AND LOC = NOE):

1	2	3	4	5	6	7	8	9	10
INITS	SID	STRESS	NOE	CSYS					
	Elem ID	GRID ID	Sxx	Syy	Szz	Sxy	Syz	Szx	
	Elem ID	GRID ID	Sxx	Syy	Szz	Sxy	Syz	Szx	
	...	...	...	...	...	...	...	...	

FORMAT OF INITIAL STRAIN DEFINED ON GRID POINTS (TYPE=STRAIN AND LOC=GRID):

1	2	3	4	5	6	7	8	9	10
INITS	SID	STRAIN	GRID	CSYS					
	GRID ID	Exx	Eyy	Ezz	Exy	Eyz	Ezx		
	GRID ID	Exx	Eyy	Ezz	Exy	Eyz	Ezx		
	...	...	...	...	...	...	...		

FORMAT OF INITIAL STRAIN AT ELEMENT GRID LOCATIONS (TYPE=STRAIN AND LOC = NOE):

1	2	3	4	5	6	7	8	9	10
INITS	SID	STRAIN	NOE	CSYS					
	Elem ID	GRID ID	Exx	Eyy	Ezz	Exy	Eyz	Ezx	
	Elem ID	GRID ID	Exx	Eyy	Ezz	Exy	Eyz	Ezx	
	...	...	...	...	...	...	...	...	

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
INITS	SID	STRAIN	NOE	0					
	12	3	.0045	.0342	.00015				
	12	28	.00012	.0035	.00015				
	12	65	.00012	.0035	.00015				
	12	72	.00012	.0035	.00015				

FIELDS:

Field	Contents
SID	Initial stress or strain set identification number. (Integer>0; also see Remark 3)
TYPE	Defines if the defined data is initial stress or initial strain. There is no default value for this field. This field must be defined. (Character) = STRAIN for initial strain. = STRESS for initial stress.

Field	Contents
LOC	The location where the data is defined. There is no default value for this field. This field must be defined. (Character) = GRID for data on grid points. = NOE for data on an element at grid locations. This can include corner and/or midside grid locations.
CSYS	Selects a coordinate system to define the initial stress or strain orientation. (Integer) = -1 Material coordinate system is used. = 0 Basic coordinate system is used. (Default) > 0 Selects a specific CSYS.
Exx,Eyy,Ezz, Exy,Eyz,Ezx	Engineering strains representing the symmetric portion of the strain tensor. (Real)
Sxx,Syy,Szz, Sxy,Syz,Szx,	Engineering stress representing the symmetric portion of the stress tensor. (Real)
When LOC=GRID:	
GRID ID	Grid point ID where data is applied. (Integer>0)
When LOC=NOE:	
Elem ID	Element where the data is applied. (Integer>0)
GRID ID	Grid point ID defining a location on the element. (Integer>0)

REMARKS:

1. The ID of all INITS and INITADD entries must be unique.
2. The option to define an initial stress or strain is available on all elements in SOL 401 except for the plane strain elements including the generalized plane strain elements, solid composite elements defined with the PCOMPS entry, and rigid elements.

3. The INITS case control must be defined above the subcases (globally), and is reapplied in each static subcase. If you want to include the effects of initial stress or strain including any other load in a normal modes subcase, you should include a static subcase before the sequentially dependent normal modes subcase.

4. For the plane stress and axisymmetric elements, if you select a coordinate system other than the basic system in the CSYS field on the INITS entry, the software first transforms the data into the basic system, then uses the components consistent with the formats described below.

   For the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, both in-plane and out-of-plane initial *strain* values are used by the software. Although, only in-plane initial *stress* values are used. For example, when the plane stress elements are defined on the XY plane and the basic coordinate system is used to define the initial stress or strain, Sxx, Syy, Sxy or Exx, Eyy, Ezz, Exy should be defined. For elements defined on the XZ plane, Sxx, Szz, Szx or Exx, Eyy, Ezz, Ezx should be defined.

   For the axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6, in-plane (radial and axial) and out-of-plane (theta) initial stress or strain values are used by the software. For example, when the axisymmetric elements are defined on the XY plane and the basic coordinate system is used to define the initial stress or strain, Sxx, Syy, Szz, Sxy or Exx, Eyy, Ezz, Exy should be defined. For elements defined on the XZ plane, Sxx, Syy, Szz, Szx or Exx, Eyy, Ezz, Ezx should be defined.

5. Initial stress and strain can be defined on a subset of the model. The software assumes a value of 0.0 at the locations where data is undefined. An exceptiion is when data is undefined at a mid-side grid point, and data is defined at both or either of the related corners. In this case, the software will interpolate a value for that mid-side grid point.

6. The software converts initial stress to strain using the elastic modulus. Stress/strain data and the yield point on the MATS1 bulk entry, if defined, are not used to convert stress to strain.

7. Multiple INITS bulk entries can be defined, each with a unique ID, then combined with the INITADD entry. The INITADD entry is selected with the ID on the INITS case control. The INITS entries selected by the INITADD entry must be all TYPE=STRESS or all TYPE=STRAIN. As a result, you cannot mix initial stress and initial strain definitions in the same input file.

8. If data is defined on the same grid or element location, a fatal error will occur.

IPLANE

Infinite Plane Definition

Defines infinite plane for acoustic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
IPLANE	IPID	TYPE	SID	CID	CSP				

EXAMPLE:

IPLANE	100		20						

IPLANE	200		0	10	1				

FIELDS:

Field	Contents
IPID	Identification number of infinite plane. (Integer > 0; No default)
TYPE	Type of reflective plane. See Remark 1. (Integer 0 or 1; Default = 0)
	0 = symmetric
	1 = anti-symmetric
SID	Surface region identification number. (Integer ≥ 0; See Remark 2 for default behavior)
CID	Rectangular coordinate system identification number. (Integer ≥ 0; Default = 0 for basic coordinate system)

Field	Contents
CSP	Plane of coordinate system. (Integer 1, 2, or 3; No default)
	1 = YZ
	2 = ZX
	3 = XY

REMARKS:

1. The TYPE field designates the infinite plane as a zero velocity or a zero pressure acoustic reflection boundary. You can select up to three infinite planes with the AMLREG entry. If you select multiple infinite planes, they must be perpendicular to one another.

   - When TYPE = 0, the infinite plane is a rigid, reflective boundary in which the velocity is zero. This is also known as a symmetric acoustic boundary. For example, air-to-ground is a zero velocity reflective boundary.

   - When TYPE = 1, the infinite plane is a pressure release reflective boundary in which the pressure is zero. This is also known as an anti-symmetric acoustic boundary. For example, air-to-water is a zero pressure reflective boundary.

2. The infinite plane is defined as follows:

   - SID > 0

     When SID > 0, the SID references a BSURFS bulk entry containing element faces that are used to define the flat infinite plane. The referenced BSURFS entry **must** contain element faces in a flat plane.

   - SID = 0 or blank

     When SID = 0 or blank, a rectangular coordinate system defined by the CID and CSP fields is used to define the flat infinite plane.

   If SID > 0, and the CID and CSP fields are specified, the SID specification takes precedence.

3. The AML region defined by the AMLREG bulk entry (that references this IPLANE entry) **must** not protrude through the infinite plane.

ITER

Iterative Solver Options

Defines options for the global iterative solver in SOLs 101, 106, 108, 111, 153 and the element iterative solver in SOLs 101 and 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ITER	SID								
	OPTION1VALUE1 OPTION2VALUE2 -etc.-								

EXAMPLE:

ITER	100		
	ITSEPS=1.0E0-7, MSGFLG=YES, PRECOND=BICWELL, IPAD=3		

FIELDS:

Field	Contents
Fields for both the global and element iterative solution:	
SID	Set identification number. (Integer > 0).
PRECOND	Preconditioner option for global and element iterative solution. (Global default = “BIC” for real analysis and “BICCMPLX” for complex analysis. Element default is to leave option blank.) See Remark 4.

Global iterative solution preconditioner options:

J	Jacobi
JS	Jacobi with diagonal scaling.
C	Incomplete Cholesky.

Field	Contents	
	CS	Incomplete Cholesky with diagonal scaling.
	RIC	Reduced incomplete Cholesky.
	RICS	Reduced incomplete Cholesky with diagonal scaling.
	BIC	Block incomplete Cholesky for real problems.
	BICCMPLX	Block incomplete Cholesky for complex problems.
	USER	User given preconditioning. For direct frequency response (SOL 108): a decomposition will be done for 1st frequency and the factor will be used for all subsequent frequencies as a preconditioner with the iterative solver. Other solutions require a DMAP alter. Please refer to the *NX Nastran Numerical Methods User's Guide* description of the SOLVIT module.
	Element iterative solution preconditioner options:	
	STD	Standard preconditioning option for the element iterative solver.
	PRIMAL	Optional preconditioning option for the element iterative solver. This option may produce faster convergence for models which contain a large number of elements with bad aspect ratios.
	blank	The default preconditioning option is used for the element iterative solver: – The STD option is used by default when contact conditions are *not* defined. – The PRIMAL option is used by default when contact conditions are defined.
CONV	Convergence criterion. (Character; Default = "AREX")	
	Global iterative solution convergence options:	
	AR	$\|\|r\|\|/\|\|b\|\|$ where r is the residual vector of current iteration and b is the initial load vector; internal criterion.
	GE	Alternative convergence criterion using geometric progression and the differences between two consecutive solution updates; internal criterion.

Field	Contents	
	AREX	Same criterion as AR but with the additional consideration of the external convergence criterion. See Remark 3 . (Default).
	GEEX	Same criterion as GE but with the additional consideration of the external convergence criterion. See Remark 3 .
	Element iterative solution convergence options:	
	AR	For relative reduction in the residual 2-norm.
	GE	For relative reduction in the residual preconditioned norm.
	AREX	For one of the other or energy norm of error. Also each must fall below a looser stopping threshold (Default).
	GEEX	For relative reduction in the residual infinity norm.
MSGFLG	Message flag. (Character; Default = “NO”)	
	YES	Messages will be printed for each iteration.
	NO	Only minimal messages will be printed from the iterative solver (Default).
ITSEPS	User-given convergence parameter epsilon for the global and element iterative solvers. (Real>0.0; Global Iterative Solver Default=1.0E-6; Element Iterative Solver Default=1.0E-8) See remark 6 for information on setting ITSEPS with the element iterative solver.	
ITSMAX	Maximum number of iterations for the global and element iterative solvers. (Integer > 0; Global Iterative Solver Default = N/4 where N is the number of rows in the matrix; Element Iterative Solver Default = 1000.) See Remark 1.	
Fields specific to the global iterative solution:		
IPAD	Global iterative solution padding value for RIC, RICS, BIC, and BICCMPLX preconditioning. The default values are recommended. (Integer)	
	Default = 0	PRECOND = “RIC” or “RICS”

Field	Contents	
	Default = 2	PRECOND = “BIC” and purely three-dimensional models.
	Default = 3	PRECOND = “BIC” and mixed element models.
	Default = 5	PRECOND = “BICCMPLX”.
IEXT	Global iterative solution extraction level in reduced or block incomplete Cholesky factorization. The default values are recommended. (Integer = 0–7)	
	Default = 0	Purely three-dimensional model. Requires USET and SIL.
	Default = 1	Purely two-dimensional model. Heuristic block structure.
	Default = 2	Mixed element types including rigid elements.
PREFONLY	Specifies early termination of the global iterative solver. (Integer = 0 or -1; Default = 0)	
	0	Runs to completion (Default).
	-1	Terminates after preface giving resource estimates.
Field specific to the element iterative solution:		
ZPIVOT	Used to adjust the singularity tolerance during the preconditioner phase of an element iterative solution. (Real, Default=1.0e-9, Default=1.0e-10 when contact (BCSET) or glue (BGSET) conditions exist)	

REMARKS:

1. The global iterative solver is supported in solutions 101, 106, 108, 111, and 153. The element iterative solver is supported in solutions 101 and 401. The ITER bulk entry is optional and only required to override the defaults. The ITER bulk entry is selected by the SMETHOD = SID case control command. Note that, if the ITER

bulk entry is defined, the default CONV for the serial global iterative solver in SOLs 108 and 111 is GEEX instead of AREX.

2. The global iterative solver is requested by specifying ITER=YES on the NASTRAN statement.

   The element iterative solver is requested in SOL 101 by specifying both ELEMITER=YES and ITER=YES on the NASTRAN statement.

   The element iterative solver is requested in SOL 401 by specifying the SOLVER parameter as ELEMITER on the NLCNTL entry. The ELEMITER and ITER keywords on the NASTRAN statement are not supported by SOL 401.

3. The global and element iterative solvers compute the external epsilon as follows:

$$\varepsilon = \frac{(r, x)}{(b, x)}$$

   where r is the final residual vector, x is the final solution vector and b is the initial load vector (*r*, *x*) indicates the inner product of *r* and *x* and (*b*, *x*) indicates the inner product of *b* and *x*.

4. When using the global iterative solver with the SMP or DMP parallel solution options, USER preconditioning is recommended with SMP, and Jacobi preconditioning with DMP.

5.

Element iterative solver restrictions in SOL 101
Superelements are not supported.
The CQUAD8, CTRIA6, and RBE2 elements are only supported if the parameter setting ELITASPC=YES is defined.
The CQUAD4 and CTRIA3 elements are supported only if the parameter K6ROT is defined (results obtained will be the same as if the parameter snorm = 0.0 was entered).
Inertia relief is not supported.
DMIG is not supported.
Differential stiffness is not supported.
The combination of a bolt preload in which the bolt is meshed with solid elements, contact conditions, and the element iterative solver is not supported.
GROUNDCHECK and WEIGHTCHECK case control requests are not supported.
The DMP parallel solution options are not supported. The SMP parallel solution option is supported.

Element iterative solver restrictions in SOL 401
Differential stiffness is not supported.
The combination of a bolt preload in which the bolt is meshed with solid elements and contact conditions is not supported.
GROUNDCHECK and WEIGHTCHECK case control requests are not supported.
The DMP parallel solution options are not supported. The SMP parallel solution option is supported.

6. When using the element iterative solver, the convergence tolerance (ITSEPS option) is used to control when the solution terminates. The equation to be solved is Kx=b, the residual is r=b-K(x0), where x0 is the current solution.

   The solution stops iterating when the minimum of the following four error measures is below the specified value:

   - The relative reduction in the 2–Norm of the residual (the 2–Norm of r divided by the 2–Norm of b).
   - The relative reduction in the preconditioned norm of the residual.
   - The relative reduction in the infinity norm of the residual.
   - The estimated energy norm of the error.

   For each error measure, the software also applies a looser tolerance than the tolerance you define here.

   This table shows typical results oned for convergence tolerances:

Convergence Tolerance	Typical Results
1e-5	Most displacement within 1% accuracy. Unreliable stresses.
1e-6	All displacements within 1% accuracy. Some stresses off 10–20%.
1e-7	Some stresses off as much as 5%.
1e-8	All stresses (even the small values) within 1%.

7. For continuation rows defined for either the global or the element iterative solver, no commas can appear in columns 1 through 8, and the data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

LOAD

Static Load Combination (Superposition)

Defines a static load as a linear combination of load sets defined with the ACCEL, ACCEL1, DAREA, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADE1, PLOADX1, QBDY1, QBDY2, QBDY3, QHBDY, QVECT, QVOL, RFORCE, RFORCE1, SELOAD, or SLOAD entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

EXAMPLE:

LOAD	101	-0.5	1.0	3	6.2	4			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Overall scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)

REMARKS:

1. The load vector {*P*} is defined by

$$\{P\} = S\sum_{i} Si\{P_{Li}\}$$

2. Load set IDs (Li) must be unique.

3. ACCEL, ACCEL1, and GRAV entries cannot have the same set identification number as any other loading entries. If you want to combine these with other static loads, you must use the LOAD bulk entry.

4. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

5. A LOAD entry may not reference a set identification number defined by another LOAD entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. LOAD may be used in a static or transient analysis to combine loads with constant magnitude.

2. Both LOAD and DLOAD can be used in the same analysis to define some loads with constant magnitude and some time-dependent loads.

3. SELOAD entries are not supported.

LOADCYH

Harmonic Load Input for Cyclic Symmetry

Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

EXAMPLE:

LOADCYH	10	1.0	7	C	0.5	15			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
HID	Harmonic. See Remark 6. (Integer or blank)
HTYPE	Harmonic type. (Character: “C”, “S”, “CSTAR” “SSTAR”, “GRAV”, “RFORCE”, or blank)
Si	Scale factor on Li. (Real)
Li	Load set identification number. See Remark 10. (Integer > 0)

REMARKS:

1. The LOADCYH entry is selected with the Case Control command LOAD = SID.

2. If HTYPE is blank, the load will be applied to all applicable types in the problem.

3. If HTYPE is “GRAV” or “RFORCE”, GRAV or RFORCE entry loading will be used. Harmonic loads for appropriate available harmonics will be generated automatically in these cases.

4. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.

5. If L1 refers to a set ID defined by an SPCD entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.

6. If HTYPE is “GRAV” or “RFORCE”, the entry in HID will be ignored and therefore may be blank. S2 and L2 must be blank for this case.

7. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYN entries.

8. If HTYPE = “C”, “S”, “CSTAR”, or “SSTAR”, the load on component (HTYPE) of harmonic (HID) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.

9. S must be nonzero. In addition, either S1 or S2 must be nonzero.

10. L1 and L2 may reference any of the static or dynamic loading entries including GRAV and RFORCE.

LOADCYN

Physical Load Input for Cyclic Symmetry

Defines a physical static or dynamic load for use in cyclic symmetry analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYPE	S1	L1	S2	L2	

EXAMPLE:

LOADCYN	10	1.0	1	R	0.5	17			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
SEGID	Segment identification number. (Integer)
SEGTYPE	Segment type. (Character: “R”, “L”, or blank)
Si	Scale Factors. (Real)
Li	Load set ID numbers. See Remark 8. (Integer > 0)

REMARKS:

1. The LOADCYN entry is selected by the LOAD Case Control command.
2. If SEGTYPE is blank, both R and L segments will be used in DIH-type symmetry.

3. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.

4. If L1 refers to a set ID defined by SPCD loading entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.

5. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYH entries.

6. The load on the segment (or half-segment) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.

7. S must be nonzero. In addition, either S1 or S2 must be nonzero.

8. L1 and L2 may reference any of the static or dynamic loading entries except GRAV and RFORCE.

LOADCYT

Table Load Input for Cyclic Symmetry

Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = “AXI” on the CYSYM entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TID1	LSET1	METHOD1	TID2	LSET2	METHOD2		

EXAMPLE:

LOADCYT	10	19	27		21	26	1		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
TIDi	Table ID for table load input for load set Li. See Remark 3. (Integer > 0)
LSETi	Load set Li. (Integer > 0)
METHODi	Method of interpolation. See Remark 5. (Integer: 0 or 1, Default = 0)
	0 interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. (Default)
	1 interpolate the magnitude of the load at corresponding grid points in all segments.

REMARKS:

1. The LOADCYT entry is selected by the LOAD Case Control command.

2. The load set ID given in fields 4 or 7 of this entry may refer to FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, SPCD, TEMP, or TEMPP1 Bulk Data entries.

3. Either TABLED1 or TABLED2 type tabular data of azimuth angle (Xi) versus scale factors (Yi) may be used. The azimuth angle values must be in degrees.

4. The scale factors given in the tables referenced by TIDi entries will be applied only to the magnitudes of the loads defined by LOADSET IDs given in fields 4 or 7.

5. For grid point loading entries, (like FORCE, MOMENT, SPCD, and TEMP Bulk Data entries) METHODi = 1 option should be used. For element loading entries (like PLOAD, PLOAD2, PLOAD4, and TEMPP1 Bulk Data entries) either METHODi = 0 or METHODi = 1 option can be used. In particular, if harmonic output of element stresses under temperature loading via TEMPP1 Bulk Data entry, METHODi = 0 option should be used to specify TEMPP1 load set.

LSEQ

Static Load Set Definition

Defines a sequence of static load sets.

FORMAT:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	EXCITEID	LID	TID					

EXAMPLE:

LSEQ	100	200	1000	1001					

FIELDS:

Field	Contents
SID	Set identification of the set of LSEQ entries. See Remark 5 . (Integer > 0)
EXCITEID	The EXCITEID set identification assigned to this static load vector. See Remark 5 . (Integer > 0)
LID	Load set identification number of a set of static load entries such as those referenced by the LOAD Case Control command. (Integer > 0 or blank)
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control command. (Integer > 0 or blank)

REMARKS:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.

2. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEID IDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries in the Bulk Data.

3. EXCITEID may be referenced by CLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, and SELOAD entries in order to apply the static load in nonlinear, static and dynamic analysis.

4. Element data recovery for thermal loads is not currently implemented in dynamics.

5. The SID-EXCITEID number pair must be unique with respect to similar pairs on all other LSEQ entries in the Bulk Data.

6. In a nonsuperelement analysis, LID and TID cannot both be blank. In superelement analysis, they may both be blank as long as static loads are prescribed in the upstream superelements.

7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LSEQ Bulk Data entries, all static loads whose load set IDs match the EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries in the Bulk Data are automatically processed.

Chapter 15: Bulk Data Entries M—N

MAT1

Isotropic Material Property Definition

Defines the material properties for linear isotropic materials.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

EXAMPLE:

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37+2	0.23	
	20.+4	15.+4	12.+4	1003					

FIELDS:

Field	Contents
MID	Material identification number. See Remark 11. (Integer > 0)
E	Young's modulus. (Real ≥ 0.0 or blank)
G	Shear modulus. (Real ≥ 0.0 or blank)
NU	Poisson's ratio. (-1.0 < Real ≤ 0.5 or blank)
RHO	Mass density. See Remark 6. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 10 and 11. (Real; Default = 0.0 if A is specified.)

Field	Contents
GE	Structural element damping coefficient. See Remarks 9 and 10. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied, used only to compute margins of safety in certain elements; and have no effect on the computational procedures. (Real ≥ 0.0 or blank)
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See Parameters and Parameter Applicability Tables for more information. (Integer ≥ 0 or blank)

REMARKS:

1. The material identification number must be unique for all MAT1, MAT2, MAT3, and MAT9 entries.

2. The following rules apply when E, G, or NU are blank in solutions other than SOL 401.

   - E and G may not both be blank.

   - If NU and E, or NU and G, are both blank, then both are set to 0.0.

   - If only one E, G, or NU is blank, then it will be computed from the equation: E = 2 · (1 + NU) · G. If this is not desired, then the MAT2 entry is recommended. If E, G, or NU are made temperature dependent by the MATT1 entry, then the equation is applied to the initial values only.

   Note that these rules are validated at the beginning of the solution, even when a MATS1 entry is used.

3. For solutions other than SOL 401, if values are specified for all of the properties E, G, and NU, then it is recommended that the following relationship be satisfied:

$$\left| 1 - \frac{E}{2 \cdot (1 + NU) \cdot G} \right| < 0.01$$

   If this relationship is not desired, then the MAT2 entry is recommended.

It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements as indicated in Table 15-1. Therefore, it is recommended that only the applicable properties be specified for a given element.

Table 15-1. Material Property Usage Versus Element Types

Element Entry	E	NU	G
CROD CBEAM CBAR	Extension and Bending	Not Used	Torsion Transverse Shear
CQUADi CTRIAi	Membrane, including In-plane Shear, and Bending		Transverse Shear
CSHEAR	Not Used		Shear
CRAC2D	All Terms		Not Used
CHEXA CPENTA CTETRA CRAC3D	All Terms		Not Used
CQUADX4 CQUADX8 CTRAX3 CTRAX6	Radial, Axial, Circumferential	All Coupled Ratios	Shear

4. SOL 401 uses the relationship G = E / [2 (1 + NU)] to compute E, NU, or G.
   - If only E is specified, the software sets NU = 0.0, and computes G.
   - If only G is specified, the software sets E, NU, and G to 0.0, and ignores the specified G.
   - If only E and NU are specified, the software computes G.
   - If only E and G are specified, the software computes NU.
   - If only NU and G are specified, the software computes E.

- If E, G, and NU are specified, the software uses E and NU to compute G, and ignores the specified G.

5. MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE case control command.

6. The mass density RHO will be used to compute mass for all structural elements automatically.

7. Weight density may be used in field 6 if the value 1/g is entered on the PARAM,WTMASS entry, where g is the acceleration of gravity (see Parameters and Parameter Applicability Tables for more information).

8. MCSID must be nonzero if PARAM,CURV is specified to calculate stresses or strains at grid points on plate and shell elements only.

9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o, by 2.0.

10. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP or PCOMPG entry.

11. When PCOMP or PCOMPG bulk entries are used, the MATi IDs must be less than 100,000,000. Furthermore, when only PSHELL bulk entries exist and MID4 is greater than 400,000,000, the field A has a special format. It must be defined as (C11+C12+C13)(*alpha*), where *alpha* is the MID4 thermal expansion coefficient. If MIDs larger than 99,999,999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.

12. TREF is used in two different ways:

    - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection.

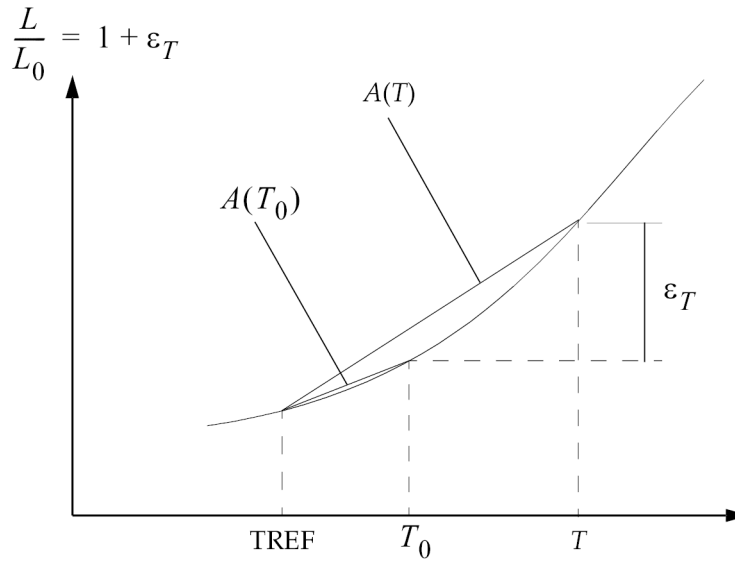


Figure 15-1. Use of TREF in Calculation of Thermal Loads

$\varepsilon_T = A(T) \cdot (T - TREF) - A(T_0) \cdot (T_0 - TREF)$

where $T$is requested by the TEMPERATURE(LOAD) command and T_0 is requested by the TEMPERATURE(INITIAL) command.

Note

a. A is a secant quantity.

b. TREF is obtained from the same source as the other material properties; e.g., ASTM, etc.

c. If $A(T)$ constant, then $\varepsilon_T = A \cdot (T - T_0)$

   o In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.

d. If PARAM,W4 is not specified, GE is ignored in transient analysis. For more information, see Parameters and Parameter Applicability Tables.

REMARKS RELATED TO SOLS 601 AND 701:

1. GE, ST, SC, SS, and MCSID are ignored.
2. RHO is always used as mass density regardless of PARAM,WTMASS entry.
3. TREF is used only used when MAT1 is made temperature dependent by use of the MATT1 entry.
4. For CQUADi and CTRIAi elements, the transverse shear modulus is equal to the in-plane shear modulus.
5. E and NU may not be both blank.

MAT2

Shell Element Anisotropic Material Property Definition

Defines the material properties for linear anisotropic materials for two-dimensional elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	C11	C12	C13	C22	C23	C33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID								

EXAMPLE:

MAT2	13	6.2+3			6.2+3		5.1+3	0.056	
	6.5-6	6.5-6		-500.0	0.002	20.+5			
	1003								

FIELDS:

Field	Contents
MID	Material identification number. See Remark 13. (Integer > 0)
Cij	The material property matrix. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 10 and 11. (Real or blank)

Field	Contents
GE	Structural element damping coefficient. See Remarks 7, 10 and 12. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. (Real or blank)
MCSID	Material coordinate system identification number. Used only for PARAM, CURV processing. See Parameters and Parameter Applicability Tables for more information. (Integer ≥ 0 or blank)

REMARKS:

1. The material identification numbers must be unique for all MAT1, MAT2, MAT3, and MAT9 entries.

2. MAT2 materials may be made temperature dependent by use of the MATT2 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.

3. The mass density, RHO, will be used to automatically compute mass for all structural elements.

4. The convention for the Cij in fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C11 & C12 & C13 \\ C12 & C22 & C23 \\ C13 & C23 & C33 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A1 \\ A2 \\ A3 \end{Bmatrix} \right)$$

5. If this entry is referenced by the MID3 field (transverse shear) on the PSHELL, then C13, C23, and C33 must be blank. See *The NASTRAN Theoretical Manual.*

6. MCSID must be nonzero if PARAM,CURV is specified to extrapolate element centroid stresses or strains to grid points on plate and shell elements only. CQUAD4 element corner stresses are not supported by PARAM,CURV.

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

8. If the MAT2 entry is referenced by a PCOMP or PCOMPG entry, the transverse shear flexibility for the referenced lamina is zero.

9. Unlike the MAT1 entry, data from the MAT2 entry is used directly without adjustment of equivalent E, G, or NU values.

10. TREF and GE are ignored if this entry is referenced by a PCOMP or PCOMPG entry.

11. TREF is used in two different ways:

    - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10 in the MAT1 description.

    - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.

12. If PARAM,W4 is not specified, GE is ignored in transient analysis. See “**Parameters** ”.

13. PCOMP or PCOMPG entries generate MATi entries greater than 100,000,000. Explicitly specified MAT2 IDs must not conflict with internally generated MAT2 IDs. When PCOMP or PCOMPG bulk data entries are used, the MATi IDs must be less than 100,000,000. Furthermore, when only PSHELL bulk data entries exist and the ID of a MAT2 referenced on the PSHELL MID4 field is greater than 400,000,000, A1, A2, and A3 are a special format. They are [*C*ij] · [*alpha*], and not [*alpha*]. If MIDs larger than 99,999,999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.

REMARKS RELATED TO SOLS 601 AND 701:

1. MAT2 defines an orthotropic material for SOLs 601 and 701 with properties calculated from C11, C12, C22, and C33. See Section 3.2.2 of the Advanced Nonlinear Theory and Modeling Guide.

2. C13, C23, A3, GE, ST, SC, SS, and MCSID are ignored.

3. TREF is used only when MAT2 is made temperature dependent by use of the MATT2 entry.

MAT3

Orthotropic Material Property Definition for Axisymmetric, Plane Stress and Plane Strain Elements

Defines linear orthotropic materials for the axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6; the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8; and the plane stress elements CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8.

Note

Except for axisymmetric models in SOL 601, 701, the use of MAT11 is recommended for defining orthotropic materials for axisymmetric elements and 2D solid elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	EY	EZ	NUXY	NUYZ	NUZX	RHO	
	GXY		GZX	AX	AY	AZ	TREF	GE	

EXAMPLE:

MAT3	23	1.0+7	1.1+7	1.2+7	.3	.25	.27	1.0-5	
			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
EX, EY, EZ	Young's moduli in the X-, Y-, and Z-principal material directions. (Real > 0.0)

Field	Contents
NUXY, NUYZ, NUZX	Poisson's ratios: (Real) NUXY = Poisson's ratio for strain in the Y-principal material direction when stressed in the X-principal material direction. NUYZ = Poisson's ratio for strain in the Z-principal material direction when stressed in the Y-principal material direction. NUZX = Poisson's ratio for strain in the X-principal material direction when stressed in the Z-principal material direction.
RHO	Mass density. (Real)
GZX	Shear modulus when elements are defined on the XZ-plane of the basic coordinate system. (Real > 0.0)
GXY	Shear modulus when elements are defined on the XY-plane of the basic coordinate system. (Real > 0.0)
AX, AY, AZ	Thermal expansion coefficients in the X-, Y-, and Z-principal material directions. (Real)
TREF	Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark 6. (Real or blank)
GE	Structural element damping coefficient. See Remarks 7 and 8. (Real)

REMARKS:

1. The material identification number must be unique with respect to the collection of all MAT1, MAT2, MAT3, MAT8, MAT9, and MAT11 entries.

2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE case control command.

3. Note the following:

$$\frac{\nu_{xy}}{E_x} = \frac{\nu_{yx}}{E_y} \qquad \frac{\nu_{xz}}{E_x} = \frac{\nu_{zx}}{E_z} \qquad \frac{\nu_{yz}}{E_y} = \frac{\nu_{zy}}{E_z}$$

See Understanding the MAT3 Bulk Entry in the *NX Nastran User's Guide* for the MAT3 strain-stress relations for the axisymmetric, plane stress, and plane strain elements.

4. A warning message will be issued if any value of NUXY or NUYZ has an absolute value greater than 1.0.

5. For CPLSTNi, CPLSTSi, CQUAX4, CQUADX8, CTRAX3, and CTRAX6 elements, the X,Y,Z–principal material directions on the MAT3 bulk entry correspond to the X_m,Y_m,Z_m–directions on the elements, respectively.

6. TREF is used for two different purposes:

   - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See the remarks on the MAT1 bulk entry.

   - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

8. If PARAM,W4 is not specified, GE is ignored in a transient analysis.

REMARKS RELATED TO SOL 601:

1. GE is ignored.

2. TREF is used only when MAT3 is made temperature dependent by use of the MATT3 entry.

MAT4

Heat Transfer Material Properties, Isotropic

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	RHO	H	m	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

EXAMPLE:

MAT4	1	204.	.900	2700.					

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real ≥ 0.0)
RHO	Density. (Real > 0.0 or blank; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
m	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0; Default = 1.0)

Field	Contents
REFENTH	Reference enthalpy. (Real or blank)
TCH	Lower temperature limit at which phase change region is to occur. (Real or blank)
TDELTA	Total temperature change range within which a phase change is to occur. (Real ≥ 0.0 or blank)
QLAT	Latent heat of fusion per unit mass associated with the phase change. (Real > 0.0 or blank)

REMARKS:

1. MAT4 may specify material properties for any conduction elements as well as properties for a forced convection fluid (see CONVM). MAT4 also provides the heat transfer coefficient for free convection (see CONV).

2. The material identification number may be the same as a structural material (for example, MAT1, MAT2, MAT3, MAT8, MAT9, MAT11 entries) but must be unique with respect to other MAT4 or MAT5 entries.

3. For a forced convection fluid, μ must be specified.

4. REFENTH is the enthalpy corresponding to zero temperature if the heat capacity CP is a constant. If CP is obtained through a TABLEM lookup, REFENTH is the enthalpy at the first temperature in the table.

5. Properties specified on the MAT4 entry may be defined as temperature dependent by use of the MATT4 entry.

REMARKS RELATED TO SOL 601:

1. m, REFENTH, TCH, TDELTA and QLAT are ignored.

MAT5

Thermal Material Property Definition

Defines the thermal material properties for anisotropic materials.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT5	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ	CP	
	RHO	HGEN							

EXAMPLE:

MAT5	24	.092			.083		0.20	0.2	
	2.00								

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
Kij	Thermal conductivity. (Real)
CP	Heat capacity per unit mass. (Real ≥ 0.0 or blank)
RHO	Density. (Real > 0.0; Default = 1.0)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0; Default = 1.0)

REMARKS:

1. X, Y, and Z on the MAT5 fields designate the material coordinates defined on the element or property entries. The general thermal conductivity matrix has the form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

   The thermal conductivity matrix for axisymmetric (see Remark 2), plane stress, and plane strain elements defined on the XZ-plane has the form:

$$K = \begin{bmatrix} KXX & KXZ \\ KXZ & KZZ \end{bmatrix}$$

   The thermal conductivity matrix for axisymmetric (see Remark 2), plane stress, and plane strain elements defined on the XY-plane has the form:

$$K = \begin{bmatrix} KXX & KXY \\ KXY & KYY \end{bmatrix}$$

   The axisymmetric, plane stress, and plane strain elements defined on the XZ plane of the basic coordinate system have different material coordinates than elements defined on the XY plane. As a result, MAT5/MATT5 entries defined for elements in one plane should not be used for elements in the other plane. See the remarks on the element entries, for example CQUADX4, for the definition of the material coordinates.

2. The material identification number may be the same as a structural material (for example, MAT1, MAT2, MAT3, MAT8, MAT9, MAT11 entries) but must be unique with respect to other MAT4 or MAT5 entries.

3. MAT5 materials may be made temperature-dependent by use of the MATT5 entry.

REMARKS RELATED TO SOL 601:

1. MAT5 defines the thermal properties for material with orthotropic thermal conductivity. KXY, KXZ and KYZ are ignored.

MAT8

Shell Element Orthotropic Material Property Definition

Defines the material property for an orthotropic material for isoparametric shell elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						

EXAMPLE:

MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						

FIELDS:

Field	Contents
MID	Material identification number. See Remark 8. (0 < Integer)
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Real ≠ 0.0)
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Real ≠ 0.0)
NU12	Poisson's ratio ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). Note that $\nu_{21}=\varepsilon_1/\varepsilon_2$ for uniaxial loading in 2-direction is related to ν_{12}, E_1, and E_2 by the relation $\nu_{12}E_2=\nu_{21}E_1$. (Real)
G12	In-plane shear modulus. (Real ≥ 0.0; Default = 0.0)

Field	Contents
G1Z	Transverse shear modulus for shear in 1-Z plane. (Real > 0.0; Default implies infinite shear modulus.)
G2Z	Transverse shear modulus for shear in 2-Z plane. (Real > 0.0; Default implies infinite shear modulus.)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient in i-direction. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 4 and 6. (Real or blank)
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. See the FT field on the PCOMP or PCOMPG entry. (Real > 0.0; Default value for Xc is Xt.)
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if failure index is desired. (Real > 0.0; Default value for Yc is Yt.)
S	Allowable stress or strain for in-plane shear. See the FT field on the PCOMP or PCOMPG entry. (Real > 0.0)
GE	Structural damping coefficient. See Remarks 4 and 7. (Real)
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Default = 0.0. See the FT field on the PCOMP or PCOMPG entry. See Remark 5. (Real)
STRN	For the maximum strain theory only (see STRN in PCOMP or PCOMPG entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables. (Real = 1.0 for strain allowables; blank (Default) for stress allowables.)

REMARKS:

1. If G1Z and G2Z values are specified as zero or blank, then transverse shear flexibility calculations will not be performed, which is equivalent to zero shear flexibility (i.e., infinite shear stiffness).

2. MAT8 materials may be made temperature dependent by use of the MATT8 entry.

3. An approximate value for G1Z and G2Z is the in-plane shear modulus G12. If test data are not available to accurately determine G1Z and G2Z for the material and transverse shear calculations are deemed essential; the value of G12 may be supplied for G1Z and G2Z. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed in the TEMPERATURE Case Control command.

4. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP or PCOMPG entry. Xc and Yc are also used but not required for failure calculations. Regardless if the FT field is defined or not, you must specify values for Xt, Yt and S if any of Xt, Xc, Yt, Yc, and S are specified.

5. If you enter a nonzero value for F_{12} and the stability criterion below is not satisfied, the software uses F_{12} = 0.0.

$$\left(\frac{1}{X_t X_c}\right)\left(\frac{1}{Y_t Y_c}\right) - F_{12}^2 > 0$$

6. TREF and GE are ignored if this entry is referenced by a PCOMP or PCOMPG entry.

7. TREF is used in two different ways:

   - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10. in the MAT1 description.

   - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.

8. If PARAM,W4 is not specified, GE is ignored in transient analysis. See “**Parameters** ”.

9. When PCOMP or PCOMPG bulk entries are used, the MATi IDs must be less than 100,000,000. Furthermore, when only PSHELL bulk entries exist and MID4 is greater than 400,000,000, A1 and A2 have a special format. They must be defined as A1=C11(*alpha1*)+C12(*alpha2*) and A2=C21(*alpha1*)+C22(*alpha2*) where *alpha1* and *alpha2* are the MID4 thermal expansion coefficients. If MIDs larger than 99,999,999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.

REMARKS RELATED TO SOLS 601 AND 701:

1. Xt, Xc, Yt, Yc, S, GE, F12, and STRN are ignored.
2. TREF is used only when MAT8 is made temperature dependent by use of the MATT8 entry.

MAT9

Solid Element Anisotropic Material Property Definition

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	C11	C12	C13	C14	C15	C16	C22	
	C23	C24	C25	C26	C33	C34	C35	C36	
	C44	C45	C46	C55	C56	C66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

EXAMPLE:

MAT9	17	6.2+3						6.2+3	
					6.2+3				
	5.1+3			5.1+3		5.1+3	3.2	6.5-6	
	6.5-6					125.	.003		

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
Cij	Elements of the 6 × 6 symmetric material property matrix in the material coordinate system. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient. (Real)

Field	Contents
TREF	Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark 7. (Real or blank)
GE	Structural element damping coefficient. See Remarks 6 and 8. (Real)

REMARKS:

1. The material identification numbers must be unique for all MAT1, MAT2, MAT3, and MAT9 entries.

2. MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In nonlinear static analysis (e.g., SOL 106), linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.

3. The mass density RHO will be used to compute mass in a structural dynamics problem automatically.

4. The third continuation entry is optional.

5. The subscripts 1 through 6 refer to x, y, z, xy, yz, and zx of the material coordinate system (see the CORDM field on the PSOLID entry description). The stress-strain relationship is

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \begin{bmatrix} C11 & C12 & C13 & C14 & C15 & C16 \\ & C22 & C23 & C24 & C25 & C26 \\ & & C33 & C34 & C35 & C36 \\ & & & C44 & C45 & C46 \\ & \text{symmetric} & & & C55 & C56 \\ & & & & & C66 \end{bmatrix} \left\{ \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} - \begin{bmatrix} A1 \\ A2 \\ A3 \\ A4 \\ A5 \\ A6 \end{bmatrix} (T - \text{TREF}) \right\}$$

6. The damping coefficient GE is given by

$$\text{GE} = \frac{2.0 \cdot C}{C_0}$$

7. TREF is used in two different ways:

   - In nonlinear static analysis (e.g., SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10 in the MAT1 description.

   - In all solutions except nonlinear static analysis, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.

8. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters** ".

REMARKS RELATED TO SOLS 601 AND 701:

1. MAT9 defines an orthotropic material for SOLs 601 and 701 with properties calculated from C11, C12, C13, C22, C23, C33, C44, C44, C55, and C66. See Section 3.2.2 of the Advanced Nonlinear Theory and Modeling Guide.

2. C14, C15, C16, C24, C25, C26, C34, C35, C36, C45, C46, C56, A4, A5, A6, and GE are ignored.

3. TREF is used only when MAT9 is made temperature dependent by use of the MATT9 entry.

MAT10

Fluid or Absorber Material Property Definition

Defines material properties for fluid or absorber elements in coupled fluid-structural analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C	GE	GAMMA			
		TIDBULK	TIDRHO		TIDGE	TIDGAMMA			

EXAMPLE:

MAT10	103	0.656	0.011						

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
BULK	Bulk modulus. (Real > 0.0)
RHO	Mass density. (Real > 0.0)
C	Speed of sound. (Real > 0.0)
GE	Fluid element damping coefficient. (Real or blank; Default = 0.0)
GAMMA	Ratio of imaginary bulk modulus to real bulk modulus. (Real or blank; Default = 0.0)
TIDBULK	Identification number of TABLEDi bulk entry containing bulk modulus versus frequency data. (Integer > 0 or blank; No default)
TIDRHO	Identification number of TABLEDi bulk entry containing mass density versus frequency data. (Integer > 0 or blank; No default)

Field	Contents
TIDGE	Identification number of TABLEDi bulk entry containing damping coefficient versus frequency data. (Integer > 0 or blank; No default)
TIDGAMMA	Identification number of TABLEDi bulk entry containing ratio of imaginary bulk modulus to real bulk modulus versus frequency data. (Integer > 0 or blank; No default)

REMARKS:

1. In solutions 103, 107-112, and 200, the MAT10 entry can be referenced by PSOLID entries.

2. The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT9, and MAT10 entries.

3. BULK, RHO, and C are always used in modal calculations. Any two of BULK, RHO, and C must be specified. The third is then calculated from:

$$\text{BULK} = C^2 \cdot \text{RHO}$$

   If all three are specified, BULK and RHO are used.

4. A 3-D frequency dependent absorber can be defined by specifying a nonzero value in the GAMMA field. In this case, the solution becomes complex and frequency dependent with the imaginary part of the governing matrix equation representing acoustic damping. The TIDBULK, TIDRHO, TIDGE, and TIDGAMMA fields allow you to model frequency dependence of the bulk modulus, density, damping coefficient, and ratio of imaginary to real bulk modulus, respectively. See Frequency-dependent acoustic materials or absorbers in the *Acoustics User's Guide* for details.

5. Specifying any combination of GE, GAMMA, TIDBULK, TIDRHO, TIDGE, and TIDGAMMA is optional.

6. If all of TIDBULK, TIDRHO, TIDGE, and TIDGAMMA are left blank, frequency dependence is ignored and BULK, RHO, GE, and GAMMA are used at all frequencies. If any of TIDBULK, TIDRHO, TIDGE, and TIDGAMMA are specified, frequency dependence is assumed. When frequency dependence is assumed, BULK is used at all frequencies if TIDBULK is left blank, RHO is used at all frequencies if TIDRHO is left blank, GE is used at all frequencies if TIDGE is left blank, and GAMMA is used at all frequencies if TIDGAMMA is left blank.

7. If PARAM,W4FL is not specified, GE is ignored in transient analysis.

8. PARAM,OMACHPR,NO (default) selects the pre-NX Nastran 8.5 format for the MAT10 entry and any continuation fields that are specified are removed with the MAKEOLD DMAP module. PARAM,OMACHPR,YES selects the current NX Nastran format for the MAT10 entry and enables the frequency-dependent material capability.

REMARKS RELATED TO SOL 601:

1. In a SOL 601,106 solution, the MAT10 entry can be referenced by a PSOLID entry selected by the 3D solid elements CHEXA, CTETRA, CPENTA, CPYRAM, or the axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6. It can also be referenced by a PPLANE entry selected by the plane strain elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8.

MAT11

Solid Element Orthotropic Material Property Definition

Defines the orthotropic material properties for axisymmetric elements, 2D solid elements, 3D solid elements, and 3D solid composite elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MAT11	MID	E1	E2	E3	NU12	NU13	NU23	G12	
	G13	G23	RHO	A1	A2	A3	TREF	GE	

EXAMPLE:

MAT11	101	2.1E7	2.2E7	2.3E7	0.31	0.29	0.33	2.1E6	
	2.2E6	2.3E6	0.34	0.35	0.36	0.37	0.38	0.39	

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
E1	Modulus of elasticity in the 1–principal material direction. See Remark 1. (Real > 0.0; No default, must be defined)
E2	Modulus of elasticity in the 2–principal material direction. See Remark 1. (Real > 0.0; No default, must be defined)
E3	Modulus of elasticity in the 3–principal material direction. See Remark 1. (Real > 0.0; No default, must be defined)
NU12	Poisson's ratio (ε_2 / ε_1 for uniaxial loading in 1-principal material direction). (Real > 0.0; No default, must be defined)

Field	Contents
NU13	Poisson's ratio (ε_3 / ε_1 for uniaxial loading in 1-principal material direction). (Real > 0.0; No default, must be defined)
NU23	Poisson's ratio (ε_3 / ε_2 for uniaxial loading in 2-principal material direction). (Real > 0.0; No default, must be defined)
G12	Shear modulus relating τ_{12} and γ_{12}. (Real > 0.0; No default, must be defined)
G13	Shear modulus relating τ_{13} and γ_{13}. (Real > 0.0; No default, must be defined)
G23	Shear modulus relating τ_{23} and γ_{23}. (Real > 0.0; No default, must be defined)
RHO	Mass density. (Real or blank; Default = 0.0)
A1	Thermal expansion coefficient in 1–principal material direction. (Real or blank; Default = 0.0)
A2	Thermal expansion coefficient in 2–principal material direction. (Real or blank; Default = 0.0)
A3	Thermal expansion coefficient in 3–principal material direction. (Real or blank; Default = 0.0)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. (Real or blank; Default = 0.0)
GE	Structural damping coefficient. (Real or blank; Default = 0.0)

REMARKS:

1. For CCHOCKi, CHEXA, CPENTA, CPLSTNi, CPLSTSi, CPYRAM, CQUAX4, CQUADX8, CTETRA, CTRAX3, and CTRAX6 elements, the 1,2,3–principal material directions on the MAT11 bulk entry correspond to the X_m,Y_m,Z_m–directions on the elements, respectively.

2. In general, NUij is not equal to NUji, but they are related as follows:

   NUij / Ei = NUji / Ej

3. Material stability requires that the following relations are satisfied:

   $Ei > NUij^2 Ej$

   and

   $1\text{-}NU12\ NU21 - NU23\ NU32 - NU31\ NU13 - 2\ NU21\ NU32\ NU13 > 0.0$

4. MAT11 materials may be made temperature dependent by use of the MATT11 entry.

5. MAT11 entries cannot be used as design variables in SOL 200 (via the DVMREL1 and DVMREL2 bulk entries).

REMARKS RELATED TO SOLS 601 AND 701:

1. GE is ignored.

2. TREF is used only when MAT11 is made temperature dependent by use of the MATT11 entry.

MATCID

Material Coordinate System for Solid Elements

Overrides the material coordinate system for CHEXA, CPENTA, CTETRA, and CPYRAM solid elements when the elements reference a PSOLID property.

Overrides the material coordinate system for CHEXA and CPENTA solid elements when the elements reference a PCOMPS property.

Overrides the material coordinate system for CHEXCZ and CPENTCZ solid elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	
	EID8	EID9	EID10	-etc.-					

EXAMPLE:

MATCID	20	101	102	103	104	105			

ALTERNATE FORMATS AND EXAMPLES:

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	“THRU”	EID2					
MATCID	20	101	THRU	105					

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	“THRU”	EID2	“BY”	N			
MATCID	20	101	THRU	105	BY	2			

1	2	3	4	5	6	7	8	9	10
MATCID	CID	“ALL”							
MATCID	20	ALL							

FIELDS:

Field	Contents
CID	Material coordinate system identification number. (Integer > -2) See Remarks 1, 2, and 3.
EIDi	Element identification number. (Integer > 0 or “ALL” or “THRU” or ”THRU” with “BY”; For “THRU” options, EID1 < EID2)
N	Element selection increment for use with ”THRU” with “BY” option. (Integer ≥ 1)

REMARKS:

1. For CHEXA, CPENTA, CTETRA, and CPYRAM solid elements that reference a PSOLID property, CID = -1 refers to the element coordinate system and CID = 0 refers to the basic coordinate system.

2. For CHEXA and CPENTA solid elements that reference a PCOMPS property, CID = -1 and CID = 0 are treated the same with both referring to the basic coordinate system. Ply stresses and strains are output in the ply coordinate system rather than the CID coordinate system.

3. For CHEXCZ and CPENTCZ solid elements that reference a PSOLCZ property, CID = -1 and CID = 0 are treated the same with both referring to the basic coordinate system.

MATCRP

Creep Material Definition

Defines coefficients for creep constitutive model in SOLs 401 and 601.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATCRP	MID	“301”	THRESH	HARD					
	A	B		D					

EXAMPLE:

MATCRP	20	301		STRAIN	104	105			
	0.5	3.2		0.8					

FIELDS:

Field	Contents
MID	Identification number of a MAT1, MAT3, MAT9, or MAT11 bulk entry. See Remark 1. (Integer > 0)
THRESH	Factor that when multiplied by the elastic modulus yields the stress threshold below which creep does not occur. (0.0 < Real < 1.0E-3; Default = 1.0E-5)
HARD	Hardening rule. (Character) “STRAIN” for strain hardening (Default) “TIME” for time hardening

Field	Contents
A,B,D	Coefficients for creep constitutive model. See Remarks 2 and 3. (Real or Integer > 0) If real entry, value of coefficient used at all temperatures. If integer entry, identification number of TABLEM1 bulk entry that defines the coefficient as a function of temperature.

REMARKS:

1. A MAT1, MAT3, MAT9, or MAT11 bulk entry that is referenced by a MATCRP bulk entry can in turn reference a corresponding MATT1, MATT3, MATT9, or MATT11 bulk entry.

2. The A, B, and D coefficients are used to define the Bailey-Norton creep model as follows:

$$\varepsilon_e^c = A\sigma_e^B t^D$$

   where ε^c is the effective creep strain, σ is effective stress, and t is time.

3. A fatal error occurs if any of the A, B, or D fields are left blank.

REMARKS RELATED TO SOL 601:

1. THRESH and HARD are ignored.

2. The MID field of the MATCRP bulk entry must reference a MAT1 bulk entry.

3. For a specific solution temperature, the software interpolates the coefficients such that the creep strain varies linearly with respect to the temperature points.

MATCZ

Cohesive Element Material

Defines damage model and material properties for cohesive elements (CHEXCZ and CPENTCZ).

FORMAT FOR POLYNOMIAL LAW (DLAW = “POLY”):

1	2	3	4	5	6	7	8	9	10
MATCZ	MID	DLAW	TANG	TAU	ADEL				
	K03S	K02	K01	GIC	GIIC	GIIIC	DCOU	Y0S	
	EXPN								

FORMAT FOR BI-TRIANGULAR LAW (DLAW = “BITR”):

1	2	3	4	5	6	7	8	9	10
MATCZ	MID	DLAW	TANG	TAU	ADEL				
	K03S	K02	K01	GIC	GIIC	GIIIC	DCOU	Y0S	

FORMAT FOR EXPONENTIAL LAW (DLAW = “EXPO”):

1	2	3	4	5	6	7	8	9	10
MATCZ	MID	DLAW	TANG	TAU	ADEL				
	K03S	K02	K01	GIC	GIIC	GIIIC	DCOU		

EXAMPLES:

MATCZ	1	POLY	1	1.5	3.0				
	4.0+8	1.0+8	1.0+8	0.32	0.48	0.48	1.0	0.1	
	0.5								

MATCZ	1	BITR	1						
	4.0+8	1.0+8	1.0+8	0.32	0.48	0.48	1.0	0.1	

MATCZ	1	EXPO	1	2.0					
	4.0+8	1.0+8	1.0+8	0.32	0.48	0.48	1.0		

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
DLAW	Damage model. (Character; Default = “POLY”) “POLY” for the Polynomial model. “BITR” for the Bi-Triangular model. “EXPO” for the Exponential model.
TANG	Tangent of constitutive model. (Integer; Default = 1) 1: Use the tangent stiffness when it is positive and use zero when it is negative. 2: Use the secant stiffness. 3: Use the tangent stiffness.
TAU	Time for delay. See Remark 2. (Real; Default = 0.0)
ADEL	Parameter for delay. See Remark 3. (Real > 0.0; Default = 1.0)
K03S	Out-of-plane normal stiffness in force per unit length^3. (Real > 0; No default)
K02	In-plane shear stiffness in force per unit length^3. (Real > 0; No default)
K01	Transverse shear stiffness in force per unit length^3. (Real > 0; No default)
GIC	Critical energy release rate for Mode I. (Real > 0; No default)
GIIC	Critical energy release rate for Mode II. (Real > 0; No default)
GIIIC	Critical energy release rate for Mode III. (Real > 0; No default)
DCOU	Coupling coefficient. (Real ≠ 0.0)

Field	Contents
Y0S	Thermodynamic force. (Real ≥ 0.0)
EXPN	Exponent. (Real > 0.0)

REMARKS:

1. The material identification number must be unique with respect to the collection of all MAT1, MAT11, and MATCZ entries.
2. Use a positive value to define time for delay. The software interprets negative values as zero time for delay.
3. If TAU ≤ 0.0, there is no time for delay and the ADEL field is ignored.

MATDMG

Material Properties for Progressive Ply Failure

Defines material properties and parameters for progressive ply failure in composite solid elements. Used in combination with MAT11 entries that have the same MID. Valid for SOL 401 only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATDMG	MID	PPFMOD							
	Y012	YC12	YS12	YS22	Y11LIMT	Y11LIMC	KSIT	KSIC	
	B2	B3	A	LITK	BIGK	EXPN	TAU	ADEL	
	PLYUNI		HBAR	DMAX	PE				

EXAMPLE:

MATDMG	3	UD							
	0.1	12.0	10.0	0.08	5.7	1.0E16	8.0		
	0.3	0.5	0.9	20.0	750.0	0.4	0.0001		
			0.43						

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
PPFMOD	Progressive ply failure model. Allowable entries are: “UD” for the unidirectional ply model. (Character; No default)
Y012	Energy threshold for shear damage d_{12}. (Real > 0; For default behavior, see Remark 1)

Field	Contents
YC12	Critical value of energy for shear damage d_{12}. (Real > 0; For default behavior, see Remark 1)
YS12	Limit value of energy for shear damage d_{12}. (Real > 0.0; No default)
YS22	Limit energy in the matrix direction. (Real > 0.0; No default)
Y11LIMT	Energy threshold in tension for the fiber direction. (Real > 0.0; No default)
Y11LIMC	Energy threshold in compression for the fiber direction. (Real > 0.0; No default)
KSIT	Nonlinearity coefficient in tension for the fiber direction. See Remark 2. (Real; Default = 0.0)
KSIC	Nonlinearity coefficient in compression for the fiber direction. See Remark 2. (Real; Default = 0.0)
B2	Coupling coefficient. (Real ≥ 0.0; Default = 0.0)
B3	Coupling coefficient between the damage variables. (Real > 0.0; Default = 1.0)
A	Coupling coefficient. (Real ≥ 0.0; Default = 0.0)
LITK	Initial plastic threshold. (Real > 0.0; Default = 1.0E15)
BIGK	Parameter for Plastic Law. (Real ≥ 0.0; Default = 0.0)
EXPN	Exponent for Plastic Law. (Real ≥ 0.0; Default = 0.0)
TAU	Time constant. See Remark 3. (Real; Default = 0.0)
ADEL	Parameter for delay. See Remark 4. (Real > 0.0; Default = 1.0)
PLYUNI	Control parameter for the application of nonlinearity coefficients. (Integer = 0 or 1; Default = 0) 0: Nonlinearity coefficients are applied to stress. 1: Nonlinearity coefficients are applied to strain.
HBAR	Transition thickness of ply. (Real ≥ 0.0; Default = 0.0)

Field	Contents
DMAX	Maximum damage value at a Gauss point. (0.0 < Real ≤ 0.999; Default = 0.999)
PE	3D effect parameter. (Integer = 0 or 1; Default = 0) 0: Plane stress effect option. The effect of damage in the normal and shear stress associated with the out-of-plane direction are ignored. 1: 3D stress effect option. The effect of damage in the normal and shear stress associated with the out-of-plane direction are included.

REMARKS:

1. An error message is issued if the Y012 and YC12 fields are both blank, or if the Y012 field or the YC12 field is defined and the other is not.
2. The software uses the nonlinearity coefficients KSIT and KSIC to calculate the elastic modulus in the fiber direction as a nonlinear function of stress if PLYUNI = 0, or strain if PLYUNI = 1.
3. Use a positive value to define time delay. The software interprets negative values as zero time delay.
4. If TAU ≤ 0.0, there is no time delay and the ADEL field is ignored.

MATFT

Material Properties for Composite Ply Failure Theories

Defines material properties for use with composite ply failure theories for solid composites defined with the PCOMPS entry.

FORMAT FOR HILL FAILURE THEORY (FT = “HILL”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

FORMAT FOR HOFFMAN FAILURE THEORY (FT = “HOFF”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

FORMAT FOR TSAI-WU FAILURE THEORY (FT = “TSAI”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23	F12	F13	F23			

FORMAT FOR MAXIMUM STRAIN FAILURE THEORY (FT = “STRN”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	Xet	Xec	Yet	Yec	Zet	Zec	Se12	
		Se13	Se23						

MATFT

FORMAT FOR MAXIMUM STRESS FAILURE THEORY (FT = “STRS”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

FORMAT FOR MAXIMUM TRANSVERSE SHEAR STRESS FAILURE THEORY (FT = “TS”)

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	FT	S13	S23						

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
MATFT	20								
	HILL	1.0E5	5.4E6	5.0E4	2.0E4	2.0E4	2.0E4		
	HOFF	7.2E6	5.6E6	5.2E6	4.4E6	4.0E5	3.5E5	2.1E6	
		2.1E6	2.1E6						

FIELDS:

Field	Contents
MID	Material identification number. See Remark 1. (Integer > 0)

Field	Contents
FT	Ply failure theory. Allowable entries are: “HILL” for the Hill failure theory. “HOFF” for the Hoffman failure theory. “TSAI” for the Tsai-Wu failure theory. “STRN” for the Maximum Strain failure theory. “STRS” for the Maximum Stress failure theory. “TS” for the Maximum Transverse Shear Stress failure theory. For a detailed explanation of each failure theory, see “Laminates” in the *NX Nastran User's Guide*. (Character or blank)
Xc, Yc, Zc	Allowable compressive stress in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xc for Yc and Zc)
Xt, Yt, Zt	Allowable tensile stress in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xt for Yt and Zt)
S12, S13, S23	Allowable in-plane shear stress (S12) and transverse shear stresses (S13 and S23). (Real > 0.0)
F12, F13, F23	Interaction terms in the tensor polynomial theory of Tsai-Wu. Required if the failure index is desired and either F12, F13, or F23 are non-zero. See Remark 5. (Real; Default = 0.0)
Xec, Yec, Zec	Allowable compressive strain in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xec for Yec and Zec)
Xet, Yet, Zet	Allowable tensile strain in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xet for Yet and Zet)
Se12, Se13, Se23	Allowable in-plane shear strain (Se12) and transverse shear strains (Se13 and Se23). (Real > 0.0 or blank)

REMARKS:

1. The MID on a MATFT bulk entry must be the same as the MID on a MAT1, MAT9, or MAT11 bulk entry. Only one MATFT bulk entry is allowed per MID.

2. When a MATFT bulk entry is used in combination with a MAT1 bulk entry, stress limits defined on the MAT1 bulk entry are ignored and the stress limits defined on the MATFT bulk entry are used.

3. Multiple failure theories can be defined on a single MATFT bulk entry, and each failure theory can only be specified once.

4. If any failure modulus related to the transverse direction is not specified, failure indices and strength ratios are only calculated for the two-dimensional plane of the ply. If any failure modulus related to the longitudinal or lateral direction is not specified, failure indices and strength ratios are not calculated.

5. If you enter a nonzero value for F_{12} and the first stability criterion below is not satisfied, the software uses F_{12} = 0.0. This behavior is the same for F_{23} and F_{13} using the second and third stability criterion below, respectively. Each is evaluated independently.

$$\left(\frac{1}{Xt\ Xc}\right) * \left(\frac{1}{Yt\ Yc}\right) - F_{12}^2 > 0$$

$$\left(\frac{1}{Zt\ Zc}\right) * \left(\frac{1}{Yt\ Yc}\right) - F_{23}^2 > 0$$

$$\left(\frac{1}{Xt\ Xc}\right) * \left(\frac{1}{Zt\ Zc}\right) - F_{13}^2 > 0$$

6. Failure index or strength ratio results for the maximum stress and maximum strain failure theories are ordered differently than the corresponding allowable values are listed on the MATFT bulk entry.

Order on MATFT bulk entry	Order in results file
Value corresponding to 11 component	Value corresponding to 11 component
Value corresponding to 22 component	Value corresponding to 22 component
Value corresponding to 33 component	Value corresponding to 33 component
Value corresponding to 12 component	Value corresponding to 12 component
Value corresponding to 13 component	Value corresponding to 23 component
Value corresponding to 23 component	Value corresponding to 13 component

MATG

Gasket Material Properties (SOL 601 only)

Defines the material properties for gasket materials.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATG	MID	IDMEM	BEHAV	TABLD	TABLU1	TABLU2	TABLU3	TABLU4	
	TABLU5	TABLU6	TABLU7	TABLU8	TABLU9	TABLU10	YPRESS	EPL	
	GPL								

EXAMPLE:

MATG	3	1	0	101	102	103	104		
							350	2000	
	1270								

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
IDMEM	ID of MAT1 providing material behavior for the membrane (in-plane) direction. (Integer > 0)
BEHAV	Behavior type. Only type 0 is supported. (Integer)
TABLD	Identification number of a TABLES1 entry providing the main loading curve of the gasket (pressure vs. closure). (Integer > 0)
TABLUi	Identification number of a TABLES1 entry providing the unloading curve(s) of the gasket (pressure vs. closure). At least one unloading curve must be specified. Leave fields blank for tables that are not required. See Remark 3. (Integer > 0)

Field	Contents
YPRESS	Initial yield pressure. See Remark 5. (Real >0.0)
EPL	Tensile modulus (pressure per unit length). (Real > 0.0)
GPL	Transverse shear modulus (force per unit area). (Real > 0.0)

REMARKS:

1. MATG can only be used with CHEXA elements with 8 grid points or CPENTA elements with 6 grid points. There should be only one layer of element elements in the direction of the gasket thickness.

2. MATG defines the compressive behavior in the thickness direction. The software will attempt to automatically determine the thickness direction and set it as the material X-axis if it is not defined by CORDM of PSOLID entry. The in-plane behavior is defined by the MAT1 entry referenced by IDMEM.

3. All unloading curves must have the same number of points as the elastic portion of the loading curve (i.e., up to the initial yield pressure). The first point must be defined at pressure=0.0 and the last point must coincide with a yield point on the loading curve. Not all yield points require unloading curves. The unloading behavior at intermediate points is interpolated from adjacent curves. See Figure 15-2.

4. MID, IDMEM, TABLD, TABLU1, YPRS, EPL, and GPL must be specified.

5. The initial yield stress must match a point on the loading curve of TABLD.

6. Closure (strain) is defined as the change in gasket thickness divided by the original gasket thickness.

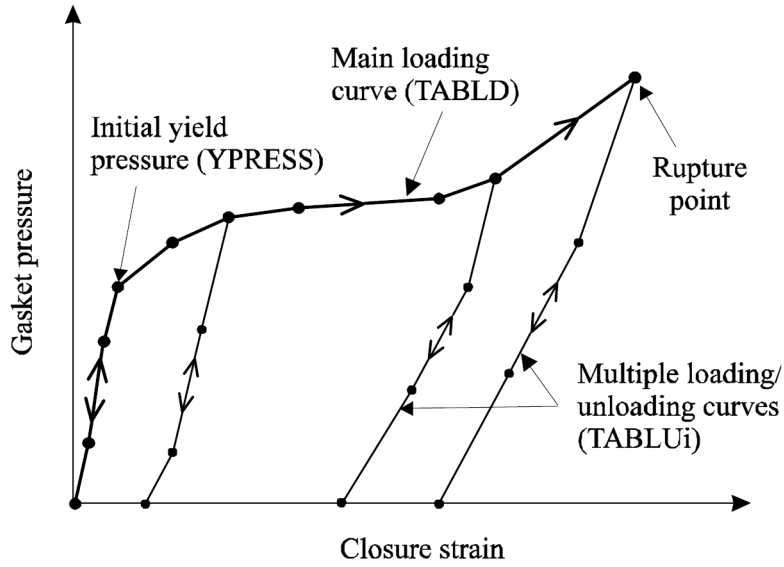


Figure 15-2. Pressure-Closure Relationship for a Gasket Material

MATHE

Hyperelastic Material Properties (SOLs 601 and 701 only)

Specifies hyperelastic (rubber-like) material properties for advanced nonlinear analysis.

FORMAT FOR GENERALIZED MOONEY-RIVLIN MODEL (DEFAULT) (MODEL = MOONEY):

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	TEXP			
	C10	C01							
	C20	C11	C02						
	C30	C21	C12	C03					

FORMAT FOR OGDEN MODEL OR HYPERFOAM MODEL (MODEL = OGDEN OR FOAM):

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	TEXP			
	MU1	ALPHA1	BETA1						
	MU2	ALPHA2	BETA2	MU3	ALPHA3	BETA3			
	MU4	ALPHA4	BETA4	MU5	ALPHA5	BETA5			
	MU6	ALPHA6	BETA6	MU7	ALPHA7	BETA7			
	MU8	ALPHA8	BETA8	MU9	ALPHA9	BETA9			

FORMAT FOR ARRUDA-BOYCE MODEL (MODEL = ABOYCE):

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	TEXP			
	NKT	N							

FORMAT FOR SUSSMAN-BATHE MODEL (MODEL=SUSSBAT):

1	2	3	4	5	6	7	8	9	1O
MATHE	MID	Model		K	RHO	TEXP			
	TAB1	SSTYPE	RELERR						

FIELDS:

Field	Contents
MID	Material identification number (Integer > 0)
Model	Hyperelastic material model type (Character; Default = Mooney)
	Mooney: Generalized Mooney-Rivlin model
	Ogden: Ogden model
	Foam: Hyperfoam model
	Aboyce: Arruda-Boyce model
	Sussbat: Sussman-Bathe model
K	Specifies a bulk modulus. Not used in hyperfoam material. See Remark 8. (Real > 0.0 or blank; Default (when blank) = automatically set for nearly incompressible condition).
RHO	Mass density in reference configuration. (Real; Default = 0.0)
Texp	Coefficient of thermal expansion. See Remark9. (Real; Default = 0.0)
Cij	Material constants related to distortional deformation for generalized Monney-Rivlin model. See Remark 3. (Real; Default = 1.0 for C10 and C01, = 0.0 for other constants).
Muk	Coefficients μ_k of the strain energy function for Ogden or hyperfoam material. See Remarks 4 and 5. (Real; Default = 0.0)
Alphak	Coefficients α_k of the strain energy function for Ogden or hyperfoam material. See Remarks 4 and 5. (Real; Default = 0.0)

Field	Contents
Betak	Coefficients β_k of the strain energy function for hyperfoam material. These fields are unused for the Ogden model. See Remark 5. (Real; Default = 0.0)
NKT	Material constant for Arruda-Boyce strain energy model. See Remark 6. (Real > 0; Default = 1.0)
N	Material constant representing the number of statistical links of the chain for Arruda-Boyce model. See Remark 6. (Real > 0; Default = 1.0)
TAB1	Table identification number of a TABLES1 entry that contains the stress-strain data associated with the material model. The stress-strain data are assumed to correspond to a uniaxial tension/compression condition. Note that both uniaxial tension and compression data must be provided to properly describe the material. (Integer > 0)
SSTYPE	The type of stress-strain data entered in TAB1. (Character; Default = Eng) Eng: engineering strains, engineering stresses True: true strains, true stresses Stretch: stretches, engineering stresses
RELERR	The relative error used to determine the number of splines used to fit the stress-strain data. (Real > 0.0; Default = 0.01)

REMARKS:

1. Model = Foam or Aboyce is not supported for SOL 701.
2. Minimum input lines: 1 for Mooney or Aboyce, 2 for Ogden, foam or Sussbat.
3. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$W(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{3} C_{ij}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + 0.5K(J - 1)^2$$

with

$$\bar{I}_1 = \bar{\lambda}_1^{\,2} + \bar{\lambda}_2^{\,2} + \bar{\lambda}_3^{\,2} \quad , \bar{I}_2 = \bar{\lambda}_1^{\,2}\bar{\lambda}_2^{\,2} + \bar{\lambda}_2^{\,2}\bar{\lambda}_3^{\,2} + \bar{\lambda}_3^{\,2}\bar{\lambda}_1^{\,2}$$

$\bar{\lambda}_i$

is the deviatoric stretch ratio defined as

$$\bar{\lambda}_i = J^{-1/3}\lambda_i ,$$

and K and J are the bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is related to the Mooney-Rivlin constants by $G = 2(C_{10}+C_{01})$.

The model reduces to a Mooney-Rivlin material with only two constants (C10 and C01), and to a Neo-Hookean material with one constant (C10).

4. For the Ogden material model, the strain energy function is

$$W = \sum_{k=1}^{9} \frac{\mu_k}{\alpha_k}\left(\bar{\lambda}_1^{\,\alpha_k} + \bar{\lambda}_2^{\,\alpha_k} + \bar{\lambda}_3^{\,\alpha_k} - 3\right) + 0.5K(J-1)^2$$

where the μ_k and α_k are material constants.

$\bar{\lambda}_i$ is the deviatoric stretch ratio defined as

$$\bar{\lambda}_i = J^{-1/3}\lambda_i ,$$

and K and J are the bulk modulus and volume ratio, respectively. A two-term Ogden model can be made equivalent to a simple Mooney-Rivlin model by choosing $\mu_1=2C_{10}$, $\alpha_1=2$, $\mu_2=2C_{01}$, $\alpha_2=-2$.

For small strains, the shear modulus G is related to the Ogden constants by

$$G = \frac{1}{2}\sum_{k=1}^{9} \mu_k \alpha_k$$

5. For the hyperfoam material model, the strain energy function is

$$W = \sum_{k=1}^{9} \frac{\mu_k}{\alpha_k}\left(\lambda_1^{\alpha_k} + \lambda_2^{\alpha_k} + \lambda_3^{\alpha_k} - 3 + \frac{1}{\beta_k}\left(J^{-\alpha_k\beta_k} - 1\right)\right)$$

where the μ_k, α_k and β_k are material constants, and J is the volume ratio.

For small strains, the shear modulus G and bulk modulus K are related to the hyperfoam constants by

$$G = \frac{1}{2}\sum_{k=1}^{9} \mu_k\alpha_k, \; K = \sum_{k=1}^{9}\left(\beta_k + \frac{1}{3}\right)\mu_k\alpha_k$$

These moduli must be greater than zero, hence we note that β_k should be greater than -1/3.

When all of the β_k are equal to each other, then β is related to the Poisson's ratio by

$$\beta = \frac{\nu}{1 - 2\nu}.$$

6. For the Arruda-Boyce model, the strain energy function is

$$W = N_{KT}[\frac{1}{2}\left(\overline{I}_1 - 3\right) + \frac{1}{20N}\left(\overline{I}_1^2 - 9\right) + \frac{11}{1050N^2}\left(\overline{I}_1^3 - 27\right)$$
$$+ \frac{19}{7000N^3}\left(\overline{I}_1^4 - 81\right) + \frac{519}{673750N^4}\left(\overline{I}_1^5 - 243\right)] + K\left(J\ln J - (J-1)\right)$$

with

$$\overline{I}_1 = \overline{\lambda}_1^2 + \overline{\lambda}_2^2 + \overline{\lambda}_3^2$$

where N_{kt} is a material constant and N is a material parameter representing the number of statistical links of the material chain. K and J are the bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is related to N_{kt} by

$$G = N_{KT}[1 + \frac{3}{5N} + \frac{99}{175N^2} + \frac{513}{875N^3} + \frac{42039}{67375N^4}]$$

7. For the Sussman-Bathe model, the strain energy function is

$W = w(e_1) + w(e_2) + w(e_3) + K(J \ln J - (J-1))$

with e_i = principal logarithmic strains and w(e) = a spline determined by the given uniaxial tension-compression data. K and J are the bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is equal to $\frac{1}{2}\frac{d^2w(e)}{de^2}$.

8. Although the conventional Mooney-Rivlin, Ogden, Arruda-Boyce and Sussman-Bathe materials are fully incompressible, the Mooney-Rivlin, Ogden, Arruda-Boyce and Sussman-Bathe material models in SOL 601/701 are compressible. Nearly incompressible materials may be simulated with a large value of K.

   The default value of K is chosen so that the small-strain Poisson's ratio is 0.499.

For each material model, K is defined as follows:	
Mooney-Rivlin material:	$K=1000(C_{10}+C_{01})$
Ogden material:	$K = 250\sum_{k=1}^{9}\mu_k\alpha_k$
Arruda-Boyce material:	$K=500N_{kt}$
Sussman-Bathe material:	$K = 250\frac{d^2w(e)}{de^2}$

9. The thermal strain is computed by $\varepsilon_{th}=\alpha\,(T-T_0)$ where T_0 is an initial temperature. ε_{th} is an engineering strain, thus, if the material is free to expand,

$$\varepsilon_{th} = \frac{L(t) - L(0)}{L(0)}$$

   where L(t) is the final length and L(0) is the initial length.

10. The stress and strain measures for both input and output is described in section 3.1 of the *Advanced Nonlinear Theory and Modeling Guide*.

MATHEM

Mullins Effect on Hyperelastic Material (SOL 601 only)

Specifies Mullins effect (Ogden-Roxburgh model) on hyperelastic (MATHE) material.

FORMAT

1	2	3	4	5	6	7	8	9	10
MATHEM	MID	R	M	HGEN					

EXAMPLE

MATHEM	11	2.0	22.0	0.8					

FIELDS:

Field	Contents
MID	Material identification number of a MATHE entry. (Integer > 0).
R, M	Material constants of the Ogden-Roxburgh model for the Mullins effect. (Real > 0.0)
HGEN	Fraction of energy dissipated by the Mullins effect model that is considered as heat generation. (0.0 ≤ Real ≤ 1.0, Default=0.0)

REMARKS:

1. HGEN is used only in a thermo-mechanical coupled analysis, i.e., SOL 601,153 or SOL 601,159.

MATHEV

Viscoelastic Effect on Hyperelastic Material (SOL 601 only)

Specifies viscoelastic effect (Holzapfel model) on hyperelastic (MATHE) material.

FORMAT

1	2	3	4	5	6	7	8	9	10
MATHEV	MID	SHIFT	C1	C2	TREF				
	beta1	tau1	hgen1	usage1					
	beta2	tau2	hgen2	usage2					
	-etc-								

EXAMPLE

MATHEV	11								
	0.5	0.7	0.0	0					
	0.3	0.6	0.0	1					

FIELDS:

Field	Contents
MID	Material identification number of a MATHE entry. (Integer > 0)
SHIFT	Time-temperature superposition shift law. (Integer, Default=0) 0: No time-temperature superposition 1: Use WLF (Williams-Landel-Ferry) shift function 2: Use Arrhenius shift function
C1, C2	Material constants for the WLF or Arrhenius shift functions. When SHIFT=1 or 2 is specified, C1 must be defined and C2 defaults to 0.0. (Real; C1 > 0.0, C2 ≥ 0.0, Default C2=0.0)

Field	Contents
TREF	Reference temperature used by the WLF or Arrhenius shift function. (Real; Default=0.0)
beta(i)	Factor beta for chain (i) of the viscoelastic model. (Real > 0.0)
tau(i)	Relaxation time for chain (i) of the viscoelastic model. (Real > 0.0)
hgen(i)	Fraction of dissipation energy that is considered as heat generation for chain (i). (0.0 ≤ Real ≤ 1.0, Default=0.0)
usage(i)	Usage of chain (i). (Integer, Default=0) 0: Chain is based on the deviatoric strain energy 1: Chain is based on the volumetric strain energy 2: Chain is based on the total strain energy

REMARKS:

1. hgen(i) is used only in a thermo-mechanical coupled analysis, i.e., SOL 601,153 or SOL 601,159.

MATHP

Hyperelastic Material Properties

Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATHP	MID	A10	A01	D1	RHO	AV	TREF	GE	
		NA	ND						
	A20	A11	A02	D2					
	A30	A21	A12	A03	D3				
	A40	A31	A22	A13	A04	D4			
	A50	A41	A32	A23	A14	A05	D5		
	TAB1	TAB2	TAB3	TAB4				TABD	

FIELDS:

Field	Contents
MID	Identification number of a MATHP entry. (Integer > 0; No Default)
Aij	Material constants related to distortional deformation. (Real; Default = 0.0)
Di	Material constants related to volumetric deformation. [Real ≥ 0; Default for D1 is $10^3 \cdot$ (A10+A01); Default for D2 through D5 is 0.0.]
RHO	Mass density in original configuration. (Real; Default = 0.0)
AV	Coefficient of volumetric thermal expansion. (Real; Default = 0.0)
TREF	Reference temperature. See “**MAT1** ”. (Real; Default = 0.0)
GE	Structural damping element coefficient. (Real; Default = 0.0)
NA	Order of the distortional strain energy polynomial function. (0 < Integer ≤ 5; Default = 1)

Field	Contents
ND	Order of the volumetric strain energy polynomial function. (0 < Integer ≤ 5; Default = 1)
TAB1	Table identification number of a TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0 and yi values must be values of the engineering stress F/A_0. Stresses are negative for compression and positive for tension. If this convention is not followed the job may fail to converge or issue a traceback. (Integer > 0 or blank)
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0. yi values must be values of the engineering stress F/A_0. l is the current length, F is the current force, A_0 is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $Pr_0\lambda^2/2t_0$ where P=current value of the pressure and r_0, t_0=initial radius and thickness. (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent γ and yi values must be values of the engineering shear stress F/A_0 ·. (Integer > 0 or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Aij. xi and yi values in the TABLES1 entry must be stretch ratios $\lambda_1=l/l_0$ and values of the nominal stress F/A_0.l is the current length, F is the current force, l_0 and A_0 are the initial length and cross-sectional area, respectively in the 1-direction. (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J=\lambda^3$ where $\lambda=l/l_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression. (Integer > 0 or blank)

REMARKS:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J,\bar{I}_1,\bar{I}_2) = \sum_{i+j=1}^{\mathrm{NA}} Aij(\bar{I}_1-3)^i(\bar{I}_2-3)^j + \sum_{i=1}^{\mathrm{ND}} Di(J-1-AV(T-T_0))^{2i}$$

$$\mathrm{A00} = 0$$

   where $\bar{I}_1$ and $\bar{I}_2$ are the first and second distortional strain invariants, respectively; *J*=det *F* is the determinant of the deformation gradient; and 2D1=*K* and 2(A10 + A01)=*G* at small strains, in which *K* is the bulk modulus and *G* is the shear modulus. The model reduces to a Mooney-Rivlin material if NA=1 and to a Neo-Hookean material if NA = 1 and A01 = 0.0. See Remark 2. For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required. T is the current temperature and T_0 is the initial temperature.

2. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not presently available but may be simulated with a large enough value of D1. A value of D1 higher than $10^3 \cdot$ (A10 + A01) is, however, not recommended.

3. Aij and Di are obtained from least squares fitting of experimental data. Aij, as well as Di, are in the units of force per area. One or more of four experiments (TAB1 to TAB4) may be used to obtain Aij. Di may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank, Aij must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters.

4. IF ND=1 and a nonzero value of D1 is provided or is obtained from experimental data in TABD, then the parameter estimation of the material constants Aij takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimating the material constants.

5. See “Hyperelastic Material” in the *NX Nastran Basic Nonlinear Analysis User’s Guide* for further details.

REMARKS RELATED TO SOLS 601 AND 701:

1. AV, TREF, and GE are ignored. Only A10, A01, D1, A20, A11, A02, A30, A21, A12, A03, and RHO are considered for the Mooney-Rivlin material model (NA≤3 and ND=1). TABi and TABD may be used to obtain the material constants.

2. The stress and strain measures for both input and output is described in section 3.1 of the *Advanced Nonlinear Theory and Modeling Guide*.

MATOVR

Disable Nonlinear Effects in Elements

Used to globally turn off creep and/or plasticity effects in groups of elements. Applicable to SOL 401 only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATOVR	GRPID	PL	CR						

EXAMPLE:

MATOVR	110		NO						

FIELDS:

Field	Contents
GRPID	Group identification number. See Remark 1. (Integer > 0)
PL	If nonlinear analysis is enabled, include plasticity effects. See Remark 2. (Character = “YES” or “NO”; Default = “YES”)
CR	If nonlinear analysis is enabled, include creep effects. See Remark 2. (Character = “YES” or “NO”; Default = “YES”)

REMARKS:

1. The GROUP bulk entry referenced in the GRPID field must have TYPE = “ELEM” or “PROP”. If TYPE = “ELEM”, all the elements listed in the group are selected. If TYPE = “PROP”, all the elements that reference the physical properties listed in the group are selected.

2. If the MATNL parameter specification does not enable nonlinear analysis, specifying PL or CR = “YES” does not switch on plasticity or creep effects in the group of elements.

3. Multiple MATOVR bulk entries are permissible so long as the GRPID fields are unique.

MATPLCY

Plastic-cyclic Material Properties (SOLs 601 and 701 only)

Specifies properties for a material used to model cyclic plasticity.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATPLCY	MID	E	NU	RHO	A	ISO	KIN	RUP	
	BETA	MAXITE	RTOL						

EXAMPLE:

MATPLCY	11	2.e5	0.3	.785e-8		1		1	

FIELDS:

Field	Contents
MID	Material identification number of a MATPLCY entry. (Integer > 0)
E	Young's modulus. (Real > 0.0)
NU	Poisson's ratio. (-1.0 < Real ≤ 0.5; Default = 0.0)
RHO	Mass density. (Real ≥ 0.0; Default = 0.0)
A	Thermal expansion coefficient. (Real ≥ 0.0; Default = 0.0)
ISO	Identification number of a PLCYISO entry that provides the isotropic hardening rule. It specifies the dependence of the radius of the yield surface on the plastic strains. (Integer > 0)
KIN	Identification number of a PLCYKIN entry that provides the kinematic hardening rule. It specifies the dependence of the back stresses on the plastic strains. KIN = 0 indicates no kinematic hardening. (Integer ≥ 0; Default = 0)

Field	Contents
RUP	Identification number of a PLCYRUP entry that specifies the rupture criterion. RUP = 0 indicates no rupture. (Integer ≥ 0; Default = 0)
BETA	Factor used in the stress integration. (0.0 ≤ Real ≤ 1.0 or blank; Default = blank). If blank, the program sets BETA = 1.0 for SOL 601 and 0.0 for SOL 701.
MAXITE	Maximum number of iterations used to solve for the incremental plastic strains. (Integer > 0; Default = 100)
RTOL	Relative tolerance used to assess convergence of the iterations. RTOL can be thought of as a reference incremental plastic strain. (Real > 0.0, Default = 1.0E-12)

REMARKS:

1. MID can be referenced by the CONROD, PROD, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PSHELL*, PPLANE and PSOLID entries.

   * If a PSHELL is associated to a CQUAD8 entry, the plastic-cyclic material is only supported when ELCV=1 is included in the NXSTRAT entry. ELCV=1 converts the CQUAD8 to a 9-node element.

MATPOR

Porous Material Property Definition

Defines material properties for porous materials used as acoustic absorbers.

FORMAT FOR CRAGGS MODEL (MODEL = “CRAGGS”):

1	2	3	4	5	6	7	8	9	10
MATPOR	MID	MODEL	RHO	C					
	RES	POR	TORT						

FORMAT FOR DELANEY-MIKI MODEL (MODEL = “DELMIKI”):

1	2	3	4	5	6	7	8	9	10
MATPOR	MID	MODEL	RHO	C	FRAME				
	RES	POR		DENS					

FORMAT FOR JOHNSON-CHAMPOUX-ALLARD MODEL (MODEL = “JCA”):

1	2	3	4	5	6	7	8	9	10
MATPOR	MID	MODEL	RHO	C	FRAME	GAMMA	PR	MU	
	RES	POR	TORT	DENS	L1	L2			

EXAMPLES:

MATPOR	20	CRAGGS	1.225E-9	340000.0					
	87000.0	0.97	2.52						

MATPOR	20	DELMIKI	1.225E-9	340000.0	LIMP				
	87000.0	0.97		31.0E-9					

MATPOR	20	JCA	1.225E-9	340000.0	LIMP	1.4	0.71	1.84E-5	
	87000.0	0.97	2.52	31.0E-9	0.037	0.119			

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0)
MODEL	Porous material model. (Character; No default) See Remark 2.
	“CRAGGS” for the Craggs model.
	“DELMIKI” for the Delany-Miki model.
	“JCA” for the Johnson-Champoux-Allard model.
RHO	Density of the fluid. (Real; Default = 0.0)
C	Speed of sound in the fluid. (Real; Default = 0.0)
FRAME	Compliance of solid. (Character; No default)
	“RIGID” for a rigid solid.
	“LIMP” for an extremely soft solid.
GAMMA	Ratio of constant pressure specific heat to constant volume specific heat. (Real ; Default = 0.0)
PR	Prandtl number. (Real; Default = 0.0)
MU	Dynamic viscosity. (Real; Default = 0.0)
RES	Flow resistivity. (Real; Default = 0.0)
POR	Porosity. (Real; Default = 0.0)
TORT	Tortuosity. (Real; Default = 0.0)
DENS	Density of the frame. (Real; Default = 0.0)
L1	Characteristic viscous length. (Real; Default = 0.0)
L2	Characteristic thermal length. (Real; Default = 0.0)

REMARKS:

1. MATPOR bulk entries can only be referenced by PSOLID bulk data entries and associated with CHEXA, CPENTA, CPYRAM, and CTETRA bulk data entries.

2. For a detailed explanation of each material model, see the *Acoustics User's Guide*.

MATS1

Material Stress Dependence

Specifies stress-dependent material properties for use in applications involving nonlinear materials. This entry is used if a MAT1, MAT2, MAT8, MAT9, or MAT11 entry is specified with the same MID in SOLs 106 and 129, or a MAT1, MAT3, MAT9, or MAT11 entry is specified with the same MID in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	

EXAMPLE:

MATS1	17	28	PLASTIC	0.0	1	1	2.+4		

FIELDS:

Field	Contents
MID	Identification number of a MAT1, MAT2, MAT3, MAT8, MAT9, or MAT11 entry. (Integer > 0)
TID	Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark 3. (Integer ≥ 0 or blank)
TYPE	Type of material nonlinearity. (Character: “NLELAST” or “PLASTIC” or “PLSTRN”) “NLELAST” for nonlinear elastic. Not valid for SOL 401. See Remarks 1 and 3. “PLASTIC” for elasto-plastic. Valid for SOLs 106, 129, and 401. See Remarks 2 and 3. “PLSTRN” for plastic strain. Valid for SOL 401 only. See Remarks 2 and 3.

Field	Contents
H	Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, H = 0.0. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark 2. (Real or blank; Default = 0.0 if TID field is blank)
YF	Yield function. See Remark 6. (Integer) 1 = von Mises (Default) 2 = Tresca 3 = Mohr-Coulomb 4 = Drucker-Prager
HR	Hardening rule. See Remark 7. (Integer) 1 = Isotropic (Default) 2 = Kinematic 3 = Combined isotropic and kinematic hardening
LIMIT1	Initial yield point. See Table 15-2. (Real > 0.0 or blank)
LIMIT2	Internal friction angle, measured in degrees, for the Mohr-Coulomb and Drucker-Prager yield criteria. See Table 15-2. (0.0 ≤ Real < 45.0 or blank)

Table 15-2. Yield Functions Versus LIMIT1 and LIMIT2

Yield Function (YF)	LIMIT1	LIMIT2
von Mises or Tresca	Initial Yield Stress in Tension, σ_y	Not used
Mohr-Coulomb or Drucker-Prager	2 x Cohesion, 2*c* (in units of stress)	Angle of Internal Friction φ (in degrees)

REMARKS:

1. If TYPE = “NLELAST”, then MID may reference a MAT1 entry only. The stress-strain data in the TABLES1 entry is used to determine the stress for a given value of strain. The values H, YF, HR, LIMIT1, and LIMIT2 will not be used in this case.

   Thermo-elastic analysis with temperature-dependent material properties is available for isotropic linear and nonlinear elastic materials (TYPE = “NLELAST”) and anisotropic linear elastic materials. Four options for the constitutive relation exist. These options are listed in Table 15-3 along with the required bulk entries.

Table 15-3. Constitutive Relations and Required Material Property Entries

Constitutive Relation	Required Bulk Data Entries
$\{\sigma\} = \left[G_e(T)\right]\{\varepsilon\}$	MATi and MATTi where i = 1, 2, 8, 9, or 11 (MATS1 is not required)
$\{\sigma\} = \frac{\bar{E}(\sigma,\varepsilon)}{E}\left[G_e(T)\right]\{\varepsilon\}$	MAT1, MATT1, MATS1, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T,\sigma,\varepsilon)}{E}\left[G_e\right]\{\varepsilon\}$	MAT1, MATS1, TABLEST, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T,\sigma,\varepsilon)}{E}\left[G_e(T)\right]\{\varepsilon\}$	MAT1, MATT1, MATS1, TABLEST, and TABLES1

   In Table 15-3, $\{\sigma\}$ is the stress vector, $\{\varepsilon\}$ is the strain vector, $[G_e]$ is the elasticity matrix, T is temperature, $\bar{E}$ is the effective elastic modulus, and E is the reference elastic modulus.

2. If TYPE = “PLASTIC” or “PLSTRN”, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The work hardening slope H is related to the tangential modulus E_T by:

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where E is the elastic modulus and $E_T = d\sigma/d\varepsilon$ is the slope of the stress-strain curve in the plastic region. See Figure 15-3.

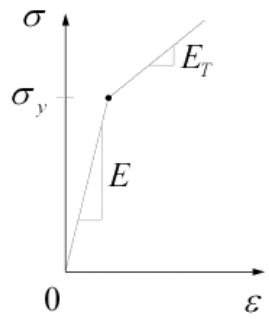


Figure 15-3. Stress vs. total strain curve when H is specified

3. If TID is specified, the tabular (Xi, Yi) data listed on the TABLES1 bulk entry represents a stress-strain curve. The strains are the Xi values and the stresses are the Yi values. The tabular data must conform to the following rules:

   - If TYPE = “NLELAST”, the stress-strain curve can be defined in the first and third quadrants to accommodate differences in tension and compression. If the stress-strain curve is defined only in the first quadrant, then the first data point must be the origin, (X1, Y1) = (0.0, 0.0), and the behavior in compression is assumed to be the mirror image of the behavior in tension.

   - If TYPE = “PLASTIC”, the TID cannot reference a TABLEST entry for SOL 106 or 129. The stress-strain curve must be defined in the first quadrant as indicated in Figure 15-4. The first data point must be the origin, (X1, Y1) = (0.0, 0.0). The second data point depends on the yield function as indicated in Table 15-4.

Table 15-4. Second Data Point Versus Yield Function

Yield Function	Applicable SOLs	Second Data Point
von Mises	SOLs 106, 129, and 401	(X2, Y2) = (ε_1, σ_y)
Tresca	SOLs 106 and 129	(X2, Y2) = (ε_1, σ_y)
Mohr-Coulomb	SOLs 106 and 129	(X2, Y2) = (ε_1, 2c)
Drucker-Prager	SOLs 106 and 129	(X2, Y2) = (ε_1, 2c)

The slope of the line joining the first and second data points must equal to the value of E on the MAT1 entry. The work hardening slope, H_k, is calculated for each successive pair of data points in the plastic region from the following formula:

$$H_k = \frac{\sigma_{k+1} - \sigma_k}{\varepsilon_{k+1}^p - \varepsilon_k^p}$$

where ε_k^p is the plastic strain at point k. The plastic strain at point k is related to the total strain at point k as follows:

$$\varepsilon_k^p = \varepsilon_k - \frac{\sigma_k}{E}$$

For SOL 106, if the LIMIT1 value is different from the value for the yield strength in the TABLES1 data, the LIMIT1 value takes precedence for the purpose of calculating the strain at yield. However, the work hardening slopes, H_k, are still calculated from the data points on the TABLES1 entry.

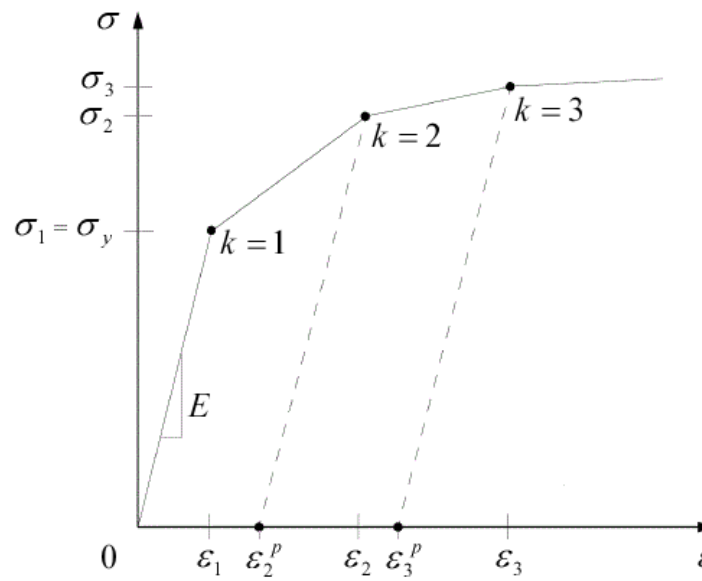


Figure 15-4. Stress vs. total strain curve when TID is specified and TYPE = "PLASTIC"

- If TYPE = "PLSTRN", the TID can reference a TABLEST entry and the stress-plastic strain curve must be defined in the first quadrant as indicated in Figure 15-5. The work hardening slope, H_k, is calculated for each successive pair of data points from the following formula:

$$H_k = \frac{\sigma_{k+1} - \sigma_k}{\varepsilon_{k+1}^p - \varepsilon_k^p}$$

where ε_k^p is the plastic strain at point *k*. The total strain at point *k* is related to the plastic strain at point *k* as follows:

$$\varepsilon_k = \varepsilon_k^p + \frac{\sigma_k}{E}$$

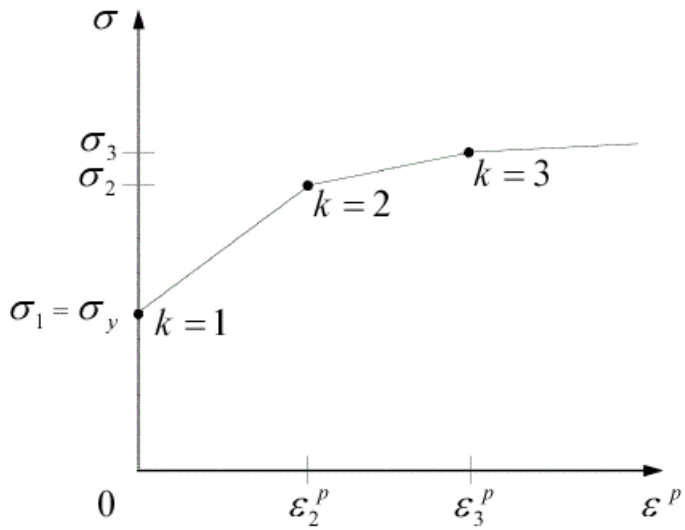


Figure 15-5. Stress vs. plastic strain curve when TID is specified and TYPE = “PLSTRN”

4. The software interprets the data on the TABLES1 entries as engineering stress and strain. See SOLs 601 and 701 Remark 13.

5. If MATS1 is specified, isotropic plasticity theory is used to calculate plastic strains regardless of elastic property type. For anisotropic elastic properties this may be unrealistic and should be used with caution. For more information, see the *NX Nastran Hanbook of Nonlinear Analysis*.

6. For SOL 401, the von Mises yield criterion is the only valid yield function. Selecting any other yield function results in a fatal error.

7. For SOL 401:

   - When a bilinear stress-strain representation is defined, all three hardening rules are supported.

- When a multilinear stress-strain representation is defined, only isotropic hardening is supported.

REMARKS RELATED TO SOLS 601 AND 701:

1. MID is restricted to the identification number of a MAT1 entry.
2. LIMIT2 is ignored. LIMIT1 is only used if TID is blank and H is specified (i.e. for a bilinear plastic material).
3. Only YF = 1 (von Mises yield criterion) is supported.
4. If HR = 3, a mixed hardening factor of 0.5 is used. HR = 3 may only be used with a bilinear plastic material (i.e. the TID must be blank).
5. TID can reference a TABLEST entry for TYPE = “PLASTIC” but not for TYPE = “NLELAST”.
6. For SOL 601, MATS1 can be combined with the CREEP entry to model a plastic-creep material. TID must be specified instead of H in this case.
7. TYPE = ”NLELAST” does not support temperature-dependent material properties or temperature loading. As a result, Table 15-3 does not apply, and there is no thermal strain.
8. For SOL 701, TYPE = ”NLELAST” can only be used with the rod element.
9. If TYPE = “NLELAST”, the full stress-strain curve (tension and compression) must be specified for the rod element. For other elements, the stress-strain curve is linearly extrapolated using the two starting and end points on the curve.
10. If the slope of the line joining the origin and second point in TABLES1 (for TYPE = “PLASTIC” and a given TID) is not equal to the value of E in MAT1 entry, the strain value at the second point will be adjusted accordingly.
11. If TYPE = “PLASTIC”, all tangent modulii ETi must satisfy the following:
    - HR = 1: $ET_i < E$

      ET_i can be negative when HR=1. Caution: Under certain modeling and loading conditions, a negative ET_i may prevent a solution from converging and cause a failure.
    - HR = 2: $0.0001 * E < E_{Ti} < E$
    - HR = 3: $0.0001 * E < E_{Ti} < E$

12. For beam elements, only the bilinear plastic material (i.e., TYPE=”PLASTIC” with H) is applicable.

13. The stress and strain measures used for input in the TABLES1 entry depends on the kinematic formulation selected in the model. The stress and strain measures for both input and output is described in section 3.1 of the *Advanced Nonlinear Theory and Modeling Guide*.

MATSMA

Shape-Memory Alloy (SMA) Material Properties (SOL 601 only)

Specifies properties for shape-memory alloy material.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATSMA	MID	EM	EA	NUM	NUA	ETMAX	VTM0	RHO	
	AM	AA	TREF	MS	MF	AS	AF		
	CM	CA	SIGMAR	CR	TOL				

EXAMPLE:

MATSMA	111	500.0	1500.0	0.2	0.3	0.1			
				70.0	10.0	90.0	130.0		
	1.0	1.0							

FIELDS:

Field	Contents
MID	Material identification number (Integer > 0)
EM	Young's modulus for martensite phase (Real > 0.0)
EA	Young's modulus for austenite phase (Real > 0.0)
NUM	Poisson's ratio for martensite (0.0 ≤ Real < 0.5; Default = 0.0)
NUA	Poisson's ratio for austenite (0.0 ≤ Real < 0.5; Default = 0.0)
ETMAX	Maximum residual transformation strain (Real > 0.0)

Field	Contents
VTM0	Initial volumetric fraction of martensite (0.0 ≤ Real ≤ 1.0; Default = 0.0)
RHO	Mass density (Real ≥ 0.0; Default = 0.0)
AM	Mean coefficient of thermal expansion for martensite (Real ≥ 0.0; Default = 0.0)
AA	Mean coefficient of thermal expansion for austenite (Real ≥ 0.0; Default = 0.0)
TREF	Reference temperature thermal expansion calculation (Real; Default = 0.0)
MS	Temperature at which the transformation to martensite starts at the stress-free state. (Real)
MF	Temperature at which the transformation to martensite is finished at the stress-free state. (Real)
AS	Temperature at which the transformation to austenite starts at the stress-free state. (Real)
AF	Temperature at which the transformation to austenite is finished at the stress-free state. (Real)
CM	Slope of the martensite transformation conditions (Real > 0.0)
CA	Slope of the austenite transformation conditions (Real > 0.0)
SIGMAR	Martensite re-orientation yield property. SIGMAR = 0.0 means that the martensite re-orientation is not performed. (Real ≥ 0.0; Default = 0.0)
CR	Slope of the martensite re-orientation yield function (Real ≥ 0.0; Default = 0.0)
TOL	Tolerance used in calculation of effective stress (Real > 0.0; Default = 1.0E-8)

REMARKS RELATED TO SOL 601:

1. MATSMA can be selected by PSHELL, PSOLID, PROD, CONROD, PCOMP, or PPLANE bulk entries. It cannot be selected by PBEAM or PBAR entries.

MATSR

Material Strain Rate Dependence (SOLs 601 and 701 only)

Specifies strain-rate dependent properties for use with MATS1 entry with the same MID.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATSR	MID	BVALUE	TSRATE	TID					

EXAMPLES:

MATSR	15	0.361	0.05						

MATSR	15			2					

FIELDS:

Field	Contents
MID	Identification number of a MATS1. (Integer > 0)
BVALUE	Strain-rate hardening parameter. Must be specified if TID is blank or zero. (Real or blank)
TSRATE	Transition strain rate. Must be specified if TID is blank or zero. (Real > 0.0 or blank)
TID	Identification number of a TABLEST entry. (Integer ≥ 0 or blank)

REMARKS:

1. BVALUE and TSRATE may be specified or be determined from the stress-strain curves at two or more strain rate in the TABLEST entry referenced by TID.

2. If TID is specified and TABLEST referenced by TID contains stress-strain curves at two or more strain rates, then the program calculates BVALUE and TSRATE which overwrites any BVALUE and TSRATE specified.

MATT1

Isotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

EXAMPLE:

MATT1	17	32				15			
	52								

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer ≥ 0 or blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)

Field	Contents
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMi entry for the tension compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMi entry for the tension shear limit. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT1 must match the identification number of the associated MAT1 entry.
   - The identification number of the MAT1 and MATT1 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E is modified by TABLEMi 32, A is modified by TABLEMi 15, and ST is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT1 entry.
3. Any quantity modified by this entry must have a value on the MAT1 entry. Initial values of E, G, or NU may be supplied according to Remark 3 on the MAT1 entry.
4. Table references must be present for each item that is temperature dependent. For example, it is not sufficient to give table references only for fields 3 and 4 (Young's modulus and shear modulus) if Poisson's ratio is temperature dependent.

5. SOL 401 computes the non-temperature dependent values E, NU, and G using the MAT1 definition, before it considers the temperature-dependent properties defined on the MATT1 entry. (See the SOL 401 remarks on the MAT1 entry for the details of this computation.)

   Because of this, the software uses the MAT1 and MATT1 values to compute

   G(T) = E' / [2 (1 + NU')]

   where:

   - E' = E(T) if E(T) is specified on the MATT1 entry, otherwise E' = E.
   - NU' = NU(T) if NU(T) is specified on the MATT1 entry, otherwise NU' = NU.

   as follows:

   - If only E(T) is specified, E(T) and NU are used to compute G(T).
   - If only NU(T) is specified, NU(T) and E are used to compute G(T).
   - If only E(T) and NU(T) are specified, E(T) and NU(T) are used to compute G(T).
   - If only E(T) and G(T) are specified, E(T) and NU are used to compute G(T), and the specified G(T) is ignored.
   - If only NU(T) and G(T) are specified, E and NU(T) are used to compute G(T), and the specified G(T) is ignored.
   - If E(T), NU(T), and G(T) are specified, E(T) and NU(T) are used to compute G(T), and the specified G(T) is ignored.
   - If only G(T) is specified, E and NU are used to compute G, and the specified G(T) is ignored. As a result of this, G is not temperature dependent.

REMARKS RELATED TO SOLS 601 AND 701:

1. Temperature-dependent property is supported only for E, NU, and A, i.e. T(RHO) is ignored.
2. As GE, ST, SC, and SS are not supported in MAT1, T(GE), T(ST), T(SC), and T(SS) are also not supported in MATT1
3. Only TABLEM1 entry may be used to define the temperature dependent properties.

4. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is restricted to 16.

MATT2

Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT2	MID	T(C11)	T(C12)	T(C13)	T(C22)	T(C23)	T(C33)	T(RHO)	
	T(A1)	T(A2)	T(A3)		T(GE)	T(ST)	T(SC)	T(SS)	

EXAMPLE:

MATT2	17	32					15		
	62								

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on a MAT2 entry. (Integer > 0)
T(Cij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficient. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)

Field	Contents
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the tension compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT2 must match the identification number of the associated MAT2 entry.
   - The identification number of the MAT2 and MATT2 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, C11 is modified by TABLEMk 32, C33 is modified by TABLEMk 15, and A1 is modified by TABLEMk 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT2 entry.
3. Any quantity modified by this entry must have a value on the MAT2 entry.

REMARKS RELATED TO SOLS 601 TO 701:

1. T(C11), T(C12), T(C22), T(C33), T(A1), and T(A2) are used to define a temperature-dependent orthotropic material. Note that T(RHO) is ignored, i.e. no temperature dependence for density.
2. As C13, C23, A3, GE, ST, SC, and SS are not supported in MAT2, T(C13), T(C23), T(A3), T(GE), T(ST), T(SC), and T(SS) are also not supported in MATT2.

3. Only TABLEM1 entry may be used to define the temperature dependent properties.

4. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is restricted to 16.

5. The temperature range specified in TABLEM1 should be as close as possible to the range of temperature loads in the model. For example, if the temperature load in the model ranges from 100.0 to 200.0, the temperature range in the TABLEM1 entries should be about 100.0 to 200.0 and not 0.0 to 1000.0.

6. To define temperature-dependent orthotropic material for shell elements, it is recommended to use MAT8 with MATT8 instead of MAT2 with MATT2.

MATT3

Orthotropic Material Temperature Dependence for Axisymmetric, Plane Stress and Plane Strain Elements

Specifies temperature-dependent material properties on MAT3 entry fields via TABLEMi entries that are temperature dependent.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT3	MID	T(EX)	T(EY)	T(EZ)	T(NUXY)	T(NUYZ)	T(NUZX)	T(RHO)	
	T(GXY)		T(GZX)	T(AX)	T(AY)	T(AZ)		T(GE)	

EXAMPLE:

MATT3	23	32		15					
			62						

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on MAT3 entry. (Integer > 0)
T(EX), T(EY), T(EZ)	Identification number of a TABLEMi entry for the Young's moduli in the X,Y,Z-principal material directions. (Integer ≥ 0 or blank)
T(NUXY), T(NUYZ), T(NUZX)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)

Field	Contents
T(GXY), T(GZX)	Identification number of a TABLEMi entry for the shear moduli. (Integer ≥ 0 or blank)
T(AX), T(AY), T(AZ)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the X,Y,Z-principal material directions. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT3 must match the identification number of the associated MAT3 entry.
   - The identification number of the MAT3 and MATT3 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT3 entry referenced in field 2. The value recorded in a particular field of the MAT3 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, EX is modified by TABLEMi 32, EZ is modified by TABLEMi 15, and GZX is modified by TABLEMi 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT3 entry.
3. Any quantity modified by this entry must have a value on the MAT3 entry.

REMARKS RELATED TO SOL 601:

1. T(GE) and T(RHO) are ignored.
2. Only TABLEM1 entry may be used to define the temperature dependent properties.

3. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is restricted to 16.

MATT4

Thermal Material Temperature Dependence

Specifies table references for temperature-dependent MAT4 material properties.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)	T(CP)		T(H)	T(μ)	T(HGEN)		

EXAMPLE(S):

MATT4	2	10	11						

FIELDS:

Field	Contents
MID	Identification number of a MAT4 entry that is temperature dependent. (Integer > 0)
T(K)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal conductivity. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(H)	Identification number of a TABLEMj entry that gives the temperature dependence of the free convection heat transfer coefficient. (Integer ≥ 0 or blank)
T(μ)	Identification number of a TABLEMj entry that gives the temperature dependence of the dynamic viscosity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMj entry that gives the temperature dependence of the internal heat generation property for QVOL. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT4 must match the identification number of the associated MAT4 entry.
   - The identification number of the MAT4 and MATT4 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. The basic quantities on the MAT4 entry are always multiplied by the corresponding tabular function referenced by the MATT4 entry.
3. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the MAT4 entry.

REMARKS RELATED TO SOL 601:

1. T(μ) is ignored.
2. Only TABLEM1 entry may be used to define the temperature dependent properties.
3. The TABLEM1 entries for T(K) and T(CP) must have the same number of temperature points with the same temperature values.

MATT5

Thermal Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT5 entry fields via TABLEMi entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	T(KXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)	T(CP)	
		T(HGEN)							

EXAMPLE:

MATT5	24	73							

FIELDS:

Field	Contents
MID	Identification number of a MAT5 entry that is to be temperature dependent. (Integer > 0)
T(Kij)	Identification number of a TABLEMi entry. The TABLEMi entry specifies temperature dependence of the matrix term. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMi entry that gives the temperature dependence of the internal heat generation property for the QVOL entry. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT5 must match the identification number of the associated MAT5 entry.
   - The identification number of the MAT5 and MATT5 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. The basic quantities on the MAT5 entry are always multiplied by the tabular function referenced by the MATT5 entry.
3. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the basic MAT5 entry.

REMARKS RELATED TO SOL 601:

1. MATT5 defines temperature-dependent thermal properties for material with orthotropic thermal conductivity. T(KXY), T(KXZ) and T(KYZ) are ignored.
2. Only TABLEM1 entry may be used to define the temperature dependent properties.
3. The TABLEM1 entries for T(KXX), T(KYY), T(KZZ) and T(CP) must have the same number of temperature points with the same temperature values.

MATT8

Shell Orthotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT8	MID	T(E1)	T(E2)	T(NU12)	T(G12)	T(G1Z)	T(G2Z)	T(RHO)	
	T(A1)	T(A2)		T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(S)	
	T(GE)	T(F12)							

EXAMPLE:

MATT8	20	10							
	11								

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on MAT8 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for Young's modulus 1. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for Young's modulus 2. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)

Field	Contents
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)
T(G1Z)	Identification number of a TABLEMi entry for transverse shear modulus 1Z. (Integer ≥ 0 or blank)
T(G2Z)	Identification number of a TABLEMi entry for transverse shear modulus 2Z. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(A1)	Identification number of a TABLEMi entry for thermal expansion coefficient 1. See Remark 3. (Integer ≥ 0 or blank)
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient 2. See Remark 3. (Integer ≥ 0 or blank)
T(Xt)	Identification number of a TABLEMi entry for tension stress/strain limit 1. (Integer ≥ 0 or blank)
T(Xc)	Identification number of a TABLEMi entry for compression stress/strain limit1. (Integer ≥ 0 or blank)
T(Yt)	Identification number of a TABLEMi entry for tension stress/strain limit 2. (Integer ≥ 0 or blank)
T(Yc)	Identification number of a TABLEMi entry for compression stress/strain limit 2. (Integer ≥ 0 or blank)
T(S)	Identification number of a TABLEMi entry for shear stress/strain limit. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer ≥ 0 or blank)
T(F12)	Identification number of a TABLEMi entry for Tsai-Wu interaction term. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:

- The identification number of the MATT8 must match the identification number of the associated MAT8 entry.
- The identification number of the MAT8 and MATT8 entries must be referenced by an element property, for example, a PSOLID entry.
- The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.

2. Fields 3,4, etc., of this entry correspond, field-by-field, to fields 3,4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 10 and A1 is modified by TABLEMi 11. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT8 entry.

3. Any quantity modified by this entry must have a value on the MAT8 entry

REMARKS RELATED TO SOLS 601 AND 701:

1. T(RHO), T(Xt), T(Xc), T(Yt), T(Yc), T(S), T(GE) and T(F12) are ignored.

2. Only TABLEM1 entry may be used to define the temperature dependent properties.

3. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is restricted to 16.

MATT9

Solid Element Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT9	MID	T(C11)	T(C12)	T(C13)	T(C14)	T(C15)	T(C16)	T(C22)	
	T(C23)	T(C24)	T(C25)	T(C26)	T(C33)	T(C34)	T(C35)	T(C36)	
	T(C44)	T(C45)	T(C46)	T(C55)	T(C56)	T(C66)	T(RHO)	T(A1)	
	T(A2)	T(A3)	T(A4)	T(A5)	T(A6)		T(GE)		

EXAMPLE:

MATT9	17	32			18			17	
				12					
				5			10		

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
T(Cij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficients. (Integer ≥ 0 or blank)

Field	Contents
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT9 must match the identification number of the associated MAT9 entry.
   - The identification number of the MAT9 and MATT9 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.
2. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value recorded in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, C11 is modified by TABLEMj 32, C14 is modified by TABLEMj 18, etc. If the fields are zero or blank, then there is no temperature dependence of the fields on the MAT9 entry.
3. Any quantity modified by this entry must have a value on the MAT9 entry.
4. The continuation entries are optional.

REMARKS RELATED TO SOLS 601 AND 701:

1. T(C11), T(C12), T(C13), T(C22), T(C23), T(C33), T(C44), T(C55), T(C66), T(A1), T(A2), and T(A3) are used to define a temperature-dependent orthotropic material. Note that T(RHO) is ignored, i.e. no temperature dependence for density.
2. As C14, C15, C16, C24, C25, C26, C34, C35, C36, C45, C46, C56, A4, A5, A6, and GE are not supported in MAT9, T(C14), T(C15), T(C16), T(C24), T(C25), T(C26), T(C34), T(C35), T(C36), T(C45), T(C46), T(C56), T(A4), T(A5), T(A6), and T(GE) are also not supported in MATT9.

3. Only TABLEM1 entry may be used to define the temperature dependent properties.

4. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is restricted to 16.

5. The temperature range specified in TABLEM1 should be as close as possible to the range of temperature loads in the model. For example, if the temperature load in the model ranges from 100.0 to 200.0, the temperature range in the TABLEM1 entries should be about 100.0 to 200.0 and not 0.0 to 1000.0.

6. To define temperature-dependent orthotropic material for solid elements, it is recommended to use MAT11 with MATT11 instead of MAT9 with MATT9.

MATT11

Solid Orthotropic Material Temperature Dependence

Defines the temperature dependent orthotropic material properties for axisymmetric elements, 2D solid elements, 3D solid elements, and 3D solid composite elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATT11	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU13)	T(NU23)	T(G12)	
	T(G13)	T(G23)	T(RHO)	T(A1)	T(A2)	T(A3)		T(GE)	

EXAMPLE:

MATT11	20	10							
			11						

FIELDS:

Field	Contents
MID	Material property identification number that matches the identification number on MAT11 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for modulus of elasticity in the 1-principal material direction. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for modulus of elasticity in the 2-principal material direction. (Integer ≥ 0 or blank)
T(E3)	Identification number of a TABLEMi entry for modulus of elasticity in the 3-principal material direction. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)

Field	Contents
T(NU13)	Identification number of a TABLEMi entry for Poisson's ratio 13. (Integer ≥ 0 or blank)
T(NU23)	Identification number of a TABLEMi entry for Poisson's ratio 23. (Integer ≥ 0 or blank)
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)
T(G13)	Identification number of a TABLEMi entry for shear modulus 13. (Integer ≥ 0 or blank)
T(G23)	Identification number of a TABLEMi entry for shear modulus 23. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(A1)	Identification number of a TABLEMi entry for thermal expansion coefficient in the 1-principal material direction. (Integer ≥ 0 or blank)
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient in the 2-principal material direction. (Integer ≥ 0 or blank)
T(A3)	Identification number of a TABLEMi entry for thermal expansion coefficient in the 3-principal material direction. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer ≥ 0 or blank)

REMARKS:

1. To include temperature dependent material properties:
   - The identification number of the MATT11 must match the identification number of the associated MAT11 entry.
   - The identification number of the MAT11 and MATT11 entries must be referenced by an element property, for example, a PSOLID entry.
   - The TEMPERATURE case control command must select the material evaluation temperatures defined in the bulk data. See the remarks on the TEMPERATURE case control command.

2. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT11 entry referenced in field 2. The value in a particular field of the MAT11 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 10 and A1 is modified by TABLEMi 11. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT11 entry.

3. Any quantity modified by this entry must have a value on the MAT11 entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. T(RHO) is ignored, i.e. no temperature dependence for mass density.

2. Only TABLEM1 entry may be used to define the temperature dependent properties.

3. The TABLEM1 entries must all have the same number of temperature points with the same temperature values. The maximum number of temperature points is 16.

MATTC

Creep Material Temperature Dependence

Specifies temperature-dependent creep coefficients on CREEP entry fields via TABLEM1 entries. Used with SOL 601 only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MATTC	MID	T(a)	T(b)		T(d)				

EXAMPLE:

MATTC	111	3	4						

FIELDS:

Field	Contents
MID	Material identification number of a CREEP entry. (Integer >0). See Remark 1.
T(a), T(b), T(d)	Identification number of a TABLEM1 entry that gives the temperature dependence of the creep coefficient. (Integer > 0). See Remarks 2 and 3.

REMARKS:

1. Only TYPE=300 in CREEP entry can be used with MATTC entry.
2. The TABLEM1 entries must all have the same number of temperature points with the same temperature values.
3. In TABLEM1 entry for T(a), the YAXIS field may be LINEAR or LOG.

4. For a specific solution temperature, the software interpolates the coefficients such that the creep strain varies linearly with respect to the temperature points.

MATVE

Viscoelastic Material Properties (SOL 601 only)

Specifies viscoelastic material properties for advanced nonlinear analysis.

FORMAT

1	2	3	4	5	6	7	8	9	10
MATVE	MID	GFUNC	KFUNC	RHO	ALPHA				
	SHIFT	C1	C2	T0					

EXAMPLE

MATVE	3	1	2						
	2	3800.0	0.003	100.0					

FIELDS:

Field	Contents
MID	Material identification number. (Integer > 0).
GFUNC	Table identification number of a TABVE entry that contains a series of shear modulii and decay coefficients to represent the shear modulus relaxation function of the material.
KFUNC	Table identification number of a TABVE entry that contains a series of bulk modulii and decay coefficients to represent the bulk modulus relaxation function of the material.
RHO	Mass density. (Real; Default=0.0)
ALPHA	Coefficient of thermal expansion. (Real; Default=0.0)

Field	Contents
SHIFT	Time-temperature superposition shift law. (Integer, Default=1) 1: Use WLF (Williams-Landel-Ferry) shift function 2: Use Arrhenius shift function
C1, C2	Material constants used by the WLF or Arrhenius shift function. (Real; Default=0.0)
T0	Reference temperature used by the WLF or Arrhenius shift function. (Real; Default=0.0)

REMARKS:

1. MATVE may be used with the rod, plane stress, plane strain, axisymmetric, solid and shell elements. Hence, it may be referenced directly by PROD, PPLANE, PSOLID, PSHELL and PCOMP entries.

MFLUID

Fluid Volume Properties

Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MFLUID	SID	CID	ZFS	RHO	ELIST1	ELIST2	PLANE1	PLANE2	
	RMAX	FMEXACT							

EXAMPLE:

MFLUID	3	2	15.73	1006.	3	4	S	N	
		100.							

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to X_3) and of planes of symmetry, if any. (Integer ≥ 0 or blank)
ZFS	Intercept of the free surface on the X_3 axis of the coordinate system referenced by CID. If X3 > ZFS then there is no fluid. See Remark 3. (Real; Default means that the free surface is located at an infinitely large positive value of X_3)
RHO	Density of the fluid. (Real)

Field	Contents
ELIST1	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on one side by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. See Remarks 3 and 5. (Integer ≥ 0)
ELIST2	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on both sides by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. (Integer ≥ 0; ELIST1 + ELIST2 > 0)
PLANE1, PLANE2	Planes of symmetry, antisymmetry, or no symmetry. “S” means that plane 1, which is the plane containing the X_1 and X_3 axes of CID, is a plane of symmetry. “A” means that plane 1 is a plane of antisymmetry. “N” means that it is neither. See Remark **5.** Plane 2 uses “S”, “A”, or “N” for the X_2 and X_3 plane. (Character: “S”, “A”, or “N”)
RMAX	Characteristic length. Interactions between elements with separation that is greater than RMAX will be neglected. (Real > 0.0; Default = 1.0E 10)
FMEXACT	Exact integration is used if the distance between two elements is less than FMEXACT times the square root of the area of the larger element. Otherwise, center point integration is used by default. (Real; Default = 1.0E 15)

REMARKS:

1. The MFLUID entry must be selected with the Case Control command MFLUID = SID.

2. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.

3. The wetted side of an element in ELIST1 is determined by the presence or absence of a minus sign preceding the element’s ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element’s positive normal, as determined by applying the right-hand rule to the sequence of its corner points. The same element can appear on two ELIST entries, indicating that it forms a barrier between the unconnected fluids.

4. The fluid volume may be finite (interior) or infinite (exterior). The volume may be bounded by a free surface and one or two planes of structural symmetry.

If structural symmetry is used, the structure must have the symmetric or antisymmetric boundary corresponding to the selection in fields 8 and 9. Interior fluids must have ELIST1 data and a free surface or plane of antisymmetry.

5. The planes of symmetry and/or antisymmetry defined in fields 8 and 9 must be planes of symmetry for the entire analysis. The user may apply appropriate structural boundary conditions at all grid points lying in these planes.

6. The current list of elements that may be placed in ELIST1 and ELIST2 include CTRIA3 and CQUAD4.

7. The continuation entry is optional.

8. If there is ELIST1 data and no free surface nor plane of antisymmetry, the program assumes a special form of external fluid. These special external fluids must have a CID (field 3) such that the origin of the fluid coordinate system is near the center of the enclosed volume, since the singularity for volume change will be placed at the origin. Special external fluids are supported only in SOLs 103, and 107 through 112. If used in conventional solution sequences, System Fatal Message 3001 results for file 205.

9. See the parameter VMOPT. VMOPT controls when the virtual mass is included in the mass matrix.

10. The SPCD method of enforced motion is not recommended for large models. Use the large mass method instead.

MICPNT

Microphone Point Definition

Defines microphone point for acoustic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MICPNT	EID	GID	Name/Descriptor						

EXAMPLE:

MICPNT	100	246	Microphone point 100 on fluid grid 246.	

FIELDS:

Field	Contents
EID	Element ID number for microphone point. (Integer > 0) See Remark 1.
GID	Fluid point grid identification number. (Integer > 0) See Remark 2.
Name/ Descriptor	The Name/Descriptor is an optional character string. (Character; 48 characters maximum; See Bulk Data Syntax Rules.) See Remark 3.

REMARKS:

1. A MICPNT is a 0D microphone element.
2. The GRID bulk entries referenced must be fluid point grids, which are designated on the GRID bulk entry by CD = -1.
3. Because of the name/descriptor field, free-field and large field data formats are not supported for this bulk entry.

MKAERO1

Mach Number - Frequency Table

Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MKAERO1	m1	m2	m3	m4	m5	m6	m7	m8	
	k1	k2	k3	k4	k5	k6	k7	k8	

EXAMPLE:

MKAERO1	.1	.7							
	.3	.6	1.0						

FIELDS:

Field	Contents
mi	List of from 1 to 8 Mach numbers. (Real ≥ 0.0)
kj	List of from 1 to 8 reduced frequencies. (Real > 0.0)

REMARKS:

1. Blank fields end the list, and thus cannot be used for 0.0.
2. All combinations of (mi, kj) will be used.
3. The continuation entry is required.
4. Multiple MKAERO1 entries are permitted.

5. For the lifting surface theories (Doublet-Lattice and Mach Box), the maximum value of kj should be less than one quarter of the number of boxes on a representative chord (i.e.,

   $$\mathrm{MAX(kj)} < \overline{C}/4\Delta x \mathrm{MAX(kj)} < \overline{C}/4\Delta x$$

   where $\overline{C}$ is the reference chord and Δx is a typical box chord length).

6. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.

7. The very low nonzero value of kj required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MKAERO2

Mach Number - Frequency Table

Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MKAERO2	m1	k1	m2	k2	m3	k3	m4	k4	

EXAMPLE:

MKAERO2	.10	.30	.10	.60	.70	.30	.70	1.0	

FIELDS:

Field	Contents
mi	Mach numbers. (Real ≥ 0.0)
ki	Reduced frequencies. (Real > 0.0)

REMARKS:

1. MKAERO2 will cause the aerodynamic matrices to be computed for the given sets of parameter pairs. Embedded blank pairs are skipped.
2. No continuation entries are allowed, but multiple MKAERO2 entries are permitted.
3. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.

4. The very low nonzero value of ki required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MODTRAK

Mode Tracking Parameters

Specifies parameters for mode tracking in design optimization (SOL 200).

FORMAT:

1	2	3	4	5	6	7	8	9	10
MODTRAK	SID	LOWRNG	HIGHRNG	MTFILTER					

EXAMPLE:

MODTRAK	100	1	26	0.80					

FIELDS:

Field	Contents
SID	Set identification number that is selected in the Case Control Section with the MODTRAK command. See Remark 1. (Integer; No Default)
LOWRNG	Lowest mode number in range to search. See Remark 2. (Integer ≥ 0, Default = 0. If nonzero, LOWRNG < HIGHRNG.)
HIGHRNG	Highest mode number in range to search. See Remark 2. (Integer > 0, Default = number of eigenvalues extracted. If nonzero, LOWRNG < HIGHRNG.)
MTFILTER	Filtering parameter used in mode cross-orthogonality check. See Remark 3. (Real, Default = 0.9)

REMARKS:

1. Only the designed modes for the subcase will be tracked. A designed mode is one that is used in the design model (in connection with either objective or constraints) and, therefore, identified on a DRESP1 entry.

2. The range of modes LOWRNG through HIGHRNG, inclusive, will be used to track the designed modes. If LOWRNG and HIGHRNG are both blank, then all computed modes will be used to search for the designed modes. Since large numbers of computed modes will result in higher computational costs, limiting the search range with LOWRNG and HIGHRNG is recommended.

3. Modes are considered to correlate if their mass normalized cross orthogonalities are greater than MTFILTER.

MOMENT

Static Moment

Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

EXAMPLE:

MOMENT	2	5	6	2.9	0.0	1.0	0.0		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni ≠ 0.0)

REMARKS:

1. The static moment applied to grid point G is defined by **m** = $M \cdot$ **N** where **N** is the vector with components N1, N2, and N3. The magnitude of **m** is equal to M times the magnitude of **N**.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command, LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. A CID of zero or blank references the basic coordinate system.

4. For scalar points see SLOAD.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply a moment with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

2. To apply a time-dependent moment, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

MOMENT1

Static Moment, Alternate Form 1

Defines a static concentrated moment at a grid point by specifying a magnitude and two grid points that determine the direction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

EXAMPLE:

MOMENT1	6	13	-2.93	16	13				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
G1, G2	Grid point identification numbers used to define the unit vector $\vec{n}$. (Integer > 0; G1 and G2 cannot be coincident.)

REMARKS:

1. The static moment applied to grid point G is defined by **m** = $M \cdot$ **n** where **n** is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "FOLLOWK"). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

REMARKS RELATED TO SOLS 601 AND 701:

1. If G1=G or G2=G, the moment is a follower moment in large displacement analysis (i.e., PARAM,LGDISP,1). Otherwise, the direction of the moment is defined by the original positions of G1 and G2 and does not change even for large displacement analysis.

2. To apply a moment with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analysis.

3. To apply a time-dependent moment, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

MOMENT2

Static Moment, Alternate Form 2

Defines a static concentrated moment at a grid point by specification of a magnitude and four grid points that determine the direction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MOMENT2	SID	G	M	G1	G2	G3	G4		

EXAMPLE:

MOMENT2	6	13	-2.93	16	13	17	13		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
Gi	Grid point identification numbers used to determine the unit vector $\vec{n}$. (Integer > 0; G1 and G2 cannot be coincident; G3 and G4 cannot be coincident.)

REMARKS:

1. The static moment applied to grid point G is defined by **m** = $M \cdot$ **n** where **n** is a unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "FOLLOWK"). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

REMARKS RELATED TO SOLS 601 AND 701:

1. The direction of the moment is defined by the original positions of G1 to G4 and does not change during the analysis.

2. To apply a moment with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

3. To apply a time-dependent moment, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

MONPNT1

Integrated Load Monitor Point

Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system. The integrated loads about this point over the associated nodes will be computed and printed for static aeroelastic analysis (SOL 144) and form integrated loads on the nonlinear static aeroelastic database.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CID	X	Y	Z			

EXAMPLE:

MONPNT1	WING155	Wing Integrated Load to Butline 155							
	34	WING		0.0	155.0	15.0			

FIELDS:

Field	Contents
NAME	A character string of up to eight characters Identifying the monitor point. (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
AXES	Component axes about which to sum. (Any unique combination of the integers 1 thru 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined.
CID	The identification number of a coordinate system in which the (x,y,z) coordinates are defined.

Field	Contents
X,Y,Z	The coordinates in the CID coordinate system about which the forces are to be summed. (Default = 0.0)

REMARKS:

1. The Identification NAME must be unique among all MONPNT1 entries.
2. The LABEL is a 56 character string that should be unique among all MONPNT1 entries (the string is case insensitive). It is used as additional label information in the printed output.

MPC

Multipoint Constraint

Defines a multipoint constraint equation of the form

$$\sum_{i} \mathrm{Aj}\ \mathrm{u}_{j} = 0$$

where u_j represents degree-of-freedom Cj at grid or scalar point Gj.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

EXAMPLE:

MPC	3	28	3	6.2	2		4.29		
		1	4	-2.91					

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
Gj	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank or zero for scalar points.)
Aj	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

REMARKS:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.

2. The first degree-of-freedom (G1, C1) in the sequence is defined to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.

3. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

4. The m-set degrees-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.

5. By default, the grid point connectivity created by the MPC and MPCADD entries is not considered during resequencing, (see the description of parameter "OLDSEQ"). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

REMARKS RELATED TO SOLS 601 AND 701:

1. No output of forces of multipoint constraint may be requested.

2. Multipoint constraints are only approximately satisfied in SOL 701, since imposing the constraints exactly results in a non-diagonal mass matrix.

3. By default, a dependent degree-of-freedom may not be assigned as independent by a rigid link (from RBAR or RBE2 entry) or another multipoint constraint (from RBAR, RBE2, RBE3 or another MPC entry). An error message will be issued in such a case. However, if GENMPC=1 is specified in the NXSTRAT entry, a dependent degree-of-freedom of a multipoint constraint can be assigned as independent by a rigid link or another multipoint constraint.

MPCADD

Multipoint Constraint Set Combination

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

EXAMPLE:

MPCADD	101	2	3	1	6	4			

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0)

REMARKS:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.

3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.

4. By default, the grid point connectivity created by the MPC and MPCADD entries is not considered during resequencing, (see “OLDSEQ” for more information). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,101.

MUMAT

Defines the material properties for the user defined material subroutine in SOL 401.

Defines the material properties for a user defined material subroutine in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
MUMAT	MID	MODNAME1	MODNAME2	NUMSTAT	MATNAME	SETID			
	REAL	REAL1	REAL2	REAL3	...				
	INTEGER	INTEGER1	INTEGER2	INTEGER3	...				
	TABLES1	TIDS1	TIDS2	TIDS3	...				
	TABLEST	TIDST1	TIDST2	TIDST3	...				
	TABLEM1	TIDM1	TIDM2	TIDM3	...				

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
MUMAT	21	MODELA	MODELB	5	TESTMAT1	6			+
+	INTEGER	98	187	574	...				+
+	REAL	1.111	2.222	R3	...				+
+	TABLES1	101	102	103	104	105	106	107	+
+	TABLES1	111	112	113	114	115	116	117	+
+		121	122	123	124	125	126	127	+
+	TABLES1	131	132	133	134	135	136	137	+
+	TABLEST	303	202	203	204	205	206	207	+
+		211	212	213	214	215			+
+	TABLEM1	301	TIDST2	TIDST3	...				+
+	TABLES1	148	149						+

FIELDS:

Field	Contents
MID	Identification number of a MAT1, MAT9, or MAT11 entry. (Integer > 0)
MODNAME1	Material model name. (Character)
MODNAME2	Material model name. (Character)
NUMSTAT	Number of state variables. (Integer, default=0)
MATNAME	Material name. (Character)
SETID	Identification number of a SET1 entry which lists the state variables for NX Nastran to output. SETID=-1 requests that all state variables be output. (Integer, default=0)
REALi	Material constants that are type REAL. (Real) Double precision is allowed. for example, 1.2345D6, but NX Nastran will convert them to REAL before storing.
INTEGERi	Material constants that are type INTEGER. (Integer)
TIDSi	Identification number of a TABLES1 entry. (Integer>0)
TIDSTi	Identification number of a TABLEST entry. (Integer>0)
TIDMi	Identification number of a TABLEM1 entry. (Integer>0)

REMARKS:

1. If a REAL, INTEGER, TABLES1, TABLEST, or TABLEM1 row is repeated, the data type in field 2 (REAL, INTEGER, TABLES1, TABLEST, TABLEM1) is optional in the consecutive row as long as the previous row is fully defined. That is, the previous row has data in fields 3 through 9.

2. You can optionally use the MODNAME1, MODNAME2, and MATNAME character fields however you would like to apply them in your external subroutine. For example, you may have multiple material models in the same routine, and these fields could be used to branch to a specific model.

NLCNTL

Strategy Parameters for SOL 401

Defines solution control parameters for SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NLCNTL	ID	PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3		
	PARAM4	VALUE4	PARAM5	VALUE5	-etc-				

EXAMPLE:

NLCNTL	1	MAXITER	30	CONV	UPW	MAXDIV	2		
	SOLVER	ELEMITER							

FIELDS:

Field	Contents
SID	Identification number. (Integer > 0)
PARAMi	Name of the NLCNTL parameter. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. (Real, Integer, or Character)

NLCNTL PARAMETERS:

Table 15-5. General Convergence and Iteration Control Parameters

Name	Description
CONV	Specifies the convergence criteria. See Remark 8. (Character = “U”, “P”, “W”, or any combination; Default = “W”)
EPSP	Error tolerance on force. (Real > 0.0; Default = 1.0E-2)
EPSU	Error tolerance on displacement. (Real > 0.0; Default = 1.0E-2)
EPSW	Error tolerance on work. (Real > 0.0; Default = 1.0E-6)
MAXBIS	Maximum number of bisections allowed. (0 < Integer < 10; Default = 5)
MAXDIV	Number of divergences before solution is assumed to diverge. (Integer > 0; Default = 3)
MAXITER	Maximum number of iterations per time step. (Integer > 0; Default = 25)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved. (Integer ≥ 0; Default = 10)

Table 15-6. Bolt Preload Parameters

Name	Description
EPSBOLT	Convergence tolerance for a nonzero bolt preload. See Remark 9. (Real > 0.0; Default = 1.0E-3)
ZERBOLT	Convergence tolerance for a zero bolt preload. See Remark 9. (Real>0.0; Default=1.0E-7)
ITRBOLT	Maximum number of bolt iterations before the bolt preload calculation is considered non-converged. See Remark 9. (Integer > 0; Default = 20)

Table 15-6. Bolt Preload Parameters

Name	Description
MISFBLT	Limits the bolt strain from one preload increment to the next. (Real > 1.0E-6; No default. The software does not apply MISFBLT when it is undefined.) MISFBLT is useful, for example, if you model a bolted joint with a gap condition. The bolt preload algorithm increments the bolt strain as the joint compresses. When a gap is present, before the joint is compressed, the axial bolt force will be relatively low. As a result, the preload algorithm will increase the strain increments. Once the gap is closed and the joint begins to compress, the bolt and contact forces will change quickly, possibly causing the contact conditions to destabilized. By limiting the bolt preload strain increments, you can reduce the chance of destablization, and utlimately help achieve convergence. At the start of a bolt preload solution, the software uses your requested preload force and the bolt geometry to estimate a bolt strain. The estimated strain (ε_{es}) is computed with the assumption that everything is rigid except for the bolt. The software then computes the maximum allowable strain increment using the new MISFBLT parameter. $\Delta\varepsilon_{max} = \varepsilon_{es}$ * MISFBLT At each preload increment, the software compares $\Delta\varepsilon_{max}$ with the next computed preload increment $\Delta\varepsilon^{i+1}$. The software uses the smaller of the two at each preload increment.
MSGLVLB	Diagnostic output for bolt preload. (Integer = 0, 1, or 2; Default = 1) 0: Bolt summary messages upon convergence. 1 -Bolt summary messages at every bolt preload iteration. 2 - Bolt summary messages and load/tolerance messages per bolt at every bolt preload iteration. MSGLVLB=1 and 2 requires MSGLVL=1. If MSGLVL= 0, MSGLVLB is also forced to 0, even if MSGLVLB=1 or 2 are defined. MSGLVLB 1 and 2 also output a table of bolt data showing the forces, moments and strains per bolt. This is only after the bolts converge.

Table 15-7. General Solution Parameters

Name	Description
LVAR	Specifies whether time unassigned loads are ramped or stepped. (Character = ”RAMP” or ”STEP”; Default = “RAMP”)
MSGLVL	Diagnostic level. (Integer = 0 or 1; Default = 1) 0: No additional diagnostic output 1: Convergence information is output for each iteration
SOLVER	Specifies the solver. See Remark 10. (Character = “SPARSE”, “PARDISO”, “MUMPS”, or “ELEMITER”; Default = “SPARSE”)
THRMST	Include thermal strain in a static analysis (Character = “YES” or “NO”; Default = “YES”)
TVAR	Specifies whether time unassigned temperature loads are ramped or stepped. (Character = ”RAMP” or ”STEP”; Default = “RAMP”)

Table 15-8. Stiffness Control Parameters

Name	Description
FOLLOWK	Include follower stiffness. Follower stiffness is computed from the follower loads defined with the FORCE1, FORCE2, PLOAD, or PLOAD4 entries. (Character = “YES” or “NO”; Statics default = “NO”; Modal default = “YES”)
KUPDATE	Stiffness update strategy. (Integer; Default = 0) -1: Initial stiffness approach 0: Auto stiffness update 1: Full Newton-Raphson >1: Quasi Newton-Raphson, and KUPDATE is the number of iterations before a stiffness update
SPINK	Include spin softening. See Remark 11. (Character = “YES” or “NO”; Statics default = “NO”; Modal default = “YES”)

Table 15-8. Stiffness Control Parameters

Name	Description
STFOPTN	Material stiffness matrix option. (Integer ≥ 0; Default = 3) 1: The elastic stiffness matrix is used. 2: The tangent stiffness matrix is used. 3: The elastic stiffness matrix is used to start each subcase, the elastic stiffness matrix is used if a stiffness update is requested prior to the beginning of a new time step, and the tangent stiffness matrix is used at any intermediate stiffness update.
STRESSK	Include stress stiffening. (Character = “YES” or “NO”; Statics default = “YES”; Modal default = “YES”)
TSTEPK	Stiffness is updated at the beginning of the time step. Applicable only if KUPDATE>1. (Character = “YES” or “NO”; Default = “NO”)

Table 15-9. Contact Parameters

Name	Description
CNTMDIV	Number of permissible contact divergences before besection is initiated. (Integer > 0; Default = 9)

Friction defined with the FRICi field on the BCTSET bulk entry along with sliding contact conditions can result in unsymmetric stiffness terms in the global stiffness matrix. Including the unsymmetric stiffness terms will slow the solution performance, although the unsymmetric terms are sometimes necessary to achieve solution convergence. You can use the FSYMTOL, KSYM and KSYMTOL parameters to control if the unsymmetric terms are included.

Table 15-9. Contact Parameters

Name	Description
FSYMTOL	Contact friction coefficient threshold to control if the sliding contact stiffness includes the unsymmetric stiffness term. (Real ≥ 0.0; Default = 0.2) When the maximum FRICi for all contact pairs is less than FSYMTOL, the unsymmetric stiffness term is not included, and a symmetric stiffness is generated. When the maximum FRICi for all contact pairs is greater than FSYMTOL, the unsymmetric stiffness term is included, and an unsymmetric stiffness is generated.
KSYM	When an unsymmetric material stiffness is generated as described in the FSYMTOL description, KSYM controls if the unsymmetric matrix is symmetrized. (Character = “AUTO”, “SYM”; or “UNSYM”; Default = “AUTO”) SYM: The unsymmetric matrix is always symmetrized. UNSYM: The unsymmetric matrix is never symmetrized. AUTO: The unsymmetric matrix is symmetrized if the KSYMTOL tolerance is satisfied as described below.
KSYMTOL	Tolerance for symmetrizing the unsymmetric global stiffness when KSYM=AUTO. (Real ≥ 0.0; Default = 0.001) If K_{full} is the unsymmetric global stiffness, the symmetric part K_{sp} is: $$K_{sp} = \frac{1}{2}\left(K_{full} + K_{full}^{\mathrm{T}}\right)$$ The unsymmetric part K_{up} is: $$K_{up} = K_{full} - K_{sp}$$ The ratio of the unsymmetric part to the full matrix is: $$r = \frac{\left\|K_{up}\right\|}{\left\|K_{full}\right\|}$$

Table 15-9. Contact Parameters

Name	Description
	The ratio r is a measure of the matrix asymmetry. It is compared with the KSYMTOL. If r < KSYMTOL, the unsymmetric matrix is symmetrized. If r > KSYMTOL, the unsymmetric matrix is used.
MSGLVLC	Diagnostic output for contact. (Integer = 0, 1 or 2; Default = 1) 0: No additional diagnostic output. 1: Convergence information is printed for each iteration. Changes in contact status are also printed. Before the first iteration, a summary of the number of sliding/sticking/active/inactive elements is printed. The number of augmentations and number of pair updates is also printed. 2: Everything described in option 1 and after each iteration, detailed information for every contact pair is printed.
USOLVER	Selects the unsymmetric solver. (Character = “HYBRID”, “SPARSE”, “PARDISO”, or “MUMPS”; Default = “HYBRID”)

Table 15-10. Plasticity and Creep Control Parameters

Name	Description
CRCERAT	For the ratio of maximum creep increment to elastic strain method, the ratio of maximum creep increment to elastic strain that is used to calculate the next time step. Valid for creep analysis only. (Real ≥ 0.0; Default = 0.4)
CRCINC	For the maximum creep increment method, the maximum creep increment that is used to calculate the next time step. Valid for creep analysis only. (Real ≥ 0.0; Default = 1.0E-4)
CREEP	Include creep effects. (Character = “YES” or “NO”; Default = “YES”)
CRICOFF	Creep strain increment below which the next time step is the product of the current time step and CRMFMX. Valid for creep analysis only. (0.0 < Real < 1.0; Default = 1.0E-6)

Table 15-10. Plasticity and Creep Control Parameters

Name	Description
CRINFAC	Integration factor used to calculate incremental creep strain. Valid for creep analysis only. See Remark 5. (0.0 ≤ Real ≤ 1.0; Default = 0.5)
CRMFMN	Minimum time step multiplying factor. If the next time step calculated by the adaptive time stepping algorithm is smaller than the product of the current time step and CRMFMN, the software halves the current time step, recalculates the current creep strain increment, and reenters the adaptive time stepping algorithm at the point the creep strain increment is compared to CRICOFF. Valid for creep analysis only. (0 ≤ Real ≤ 1.0; Default = 0.1)
CRMFMX	Maximum time step multiplying factor. See the CRICOFF parameter for additional information. Valid for creep analysis only. (Real ≥ 1.0; Default = 5.0)
CRTEABS	Maximum absolute truncation error. Valid for creep analysis only. (0.0 ≤ Real < 1.0; Default = 1.0E-4)
CRTECO	For the error truncation method, use CRTEABS to calculate the next time step if the creep strain is less than CRTECO, and use CRTEREL to calculate the next time step if the creep strain is greater than CRTECO. Valid for creep analysis only. (0.0 ≤ Real < 1.0; Default = 1.0E-4)
CRTEREL	Maximum relative truncation error. Valid for creep analysis only. (0.0 ≤ Real < 1.0; Default = 0.01)
PLASTIC	Include plasticity effects. (Character = “YES” or “NO”; Default = “YES”)

Table 15-11. Time Step Control Parameters

Name	Description
AUTOTIM	Automatic timing scheme ON/OFF parameter. When AUTOTIM=ON (default), automatic time step strategy will be used. Setting AUTOTIM=OFF turns off the automatic time stepping scheme. (Character = "ON" or "OFF"; Default="ON")
DTINIT	Initial time step or constant time step. (Real > 0.0; Default = 0.01)

Table 15-11. Time Step Control Parameters

Name	Description
DTMAX	Maximum time step. If DTMAX is set to 0.0 (default), the software accepts the next time step. If DTMAX is nonzero and the next time step is larger than DTMAX, the software uses DTMAX as the next time step. Otherwise, the next time step is compared to DTMIN. (Real ≥ 0.0; Default = 0.0)
DTMIN	Minimum time step. If the next time step is larger than DTMIN, the software accepts the next time step. For creep analysis, if the next time step is smaller than DTMIN, the software halves the current time step, recalculates the current creep strain increment, and reenters the adaptive time stepping algorithm at the point the creep strain increment is compared to CRICOFF. (0.0 ≤ Real ≤ DTMAX; Default = 0.001*DTINIT)
DTSBCDT	Option to use DTINIT or not in a new subcase. A subcase with a zero start time always uses DTINIT. (Integer; Default = 1) 0: Use the time step calculated at the end of the previous subcase. 1: Use DTINIT
EQMFMIN	Minimum time step factor for equilibrium iteration criterion. (Real > 0.0; Default=0.2)
EQMFMX	Maximum time step factor for equilibrium iteration criterion. (Real > 0.0; Default=5.0)
TSCCR	Specifies the time stepping method. Valid for creep analysis only. See Remarks 6 and 7. (Integer or blank; Default = 12) 0: Use constant time stepping 1: Use adaptive time stepping based on the error truncation method only 2: Use adaptive time stepping based on the ratio of maximum creep increment to elastic strain method only 3: Use adaptive time stepping based on the maximum creep increment method only 12: Use adaptive time stepping based on both the error truncation method and the ratio of maximum creep increment to elastic strain method

Table 15-11. Time Step Control Parameters

Name	Description
	13: Use adaptive time stepping based on both the error truncation method and the ratio of maximum creep increment method
	23: Use adaptive time stepping based on both the ratio of maximum creep increment to elastic strain method and the maximum creep increment method
	123: Use adaptive time stepping based on the error truncation method, the ratio of maximum creep increment to elastic strain method, and the maximum creep increment method
TSCEQ	Flag to turn ON/OFF stepping based on equilibrium iterations. (Character = "ON" or "OFF"; Default="ON")
TSCUMAT	Flag to activate stepping based on UMAT supplied time increments. (Character = "ON" or "OFF"; Default="ON")
UMFMIN	Minimum time step factor for UMAT stepping. (Real > 0.0; Default=0.2)
UMFMX	Maximum time step factor for UMAT stepping. (Real > 0.0; Default=5.0)

REMARKS:

1. The NLCNTL bulk entry must be selected with the NLCNTL = SID case control command.
2. NLCNTL case control commands can be placed inside subcases. Because each NLCNTL case control command can point to a different NLCNTL bulk entry, the NLCNTL parameter settings can vary from subcase to subcase.
3. A fatal error occurs when a PARAMi field is defined and the corresponding VALUEi field is left blank.
4. If an NLCNTL parameter is applicable to a certain type of analysis, but it is not defined on an NLCNTL bulk entry, the default value for the parameter is used.
5. Incremental creep strain, $\Delta\varepsilon^c$, is calculated from the generalized trapezoidal rule as follows:

$$\Delta\varepsilon^c = \Delta t\left[\left(1-\beta\right)\dot{\varepsilon}_t^c + \beta\dot{\varepsilon}_{t+\Delta t}^c\right]$$

where Δt is the current time step, $\dot{\varepsilon}^c_t$ is the creep strain rate at t, $\dot{\varepsilon}^c_{t+\Delta t}$ is the creep strain rate at $t+\Delta t$, and β is the integration factor specified with the CRINFAC parameter.

6. Solution times are the times specified by the TENDi and NINCi fields on TSTEP1 bulk entries. At all times during the creep analysis, if the next time step would result in skipping over a solution time, the software truncates the next time step so that a solve occurs at the solution time. If a time step is truncated to avoid skipping over a solution time, the truncated time step is not subject to the any minimum time step requirement.

7. When you select an adaptive time stepping algorithm that is based on multiple methods, the software calculates a value for the next time step from each selected method. The software compares the values and uses the smallest as the next time step.

8. The convergence test flags (U = displacement error, P = load equilibrium error, and W = work error) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, and/or W) are satisfied upon convergence.

9.

   - When the software determines that your requested preload is nonzero, if the difference between the software computed preload and the user-defined preload is less than EPSBOLT, the bolt preload calculation is considered converged. If the difference is greater than EPSBOLT, the preload strain is modified accordingly for the next bolt preload iteration. The iterations continue until either convergence is satisfied, or the number of iterations reaches ITRBOLT.

   - When the software determines that your requested preload is zero, it uses ZERBOLT to check for convergence by checking if P/AE < ZERBOLT, where P is the axial bolt force from an applied axial strain value, A is the area of the bolt cross section, and E is the modulus of the material selected for the elements on the bolt. EPSBOLT is ignored for a zero bolt preload.

     Note that the software considers a bolt preload force to be zero when F/AE < ZERBOLT, where F is the preload force you requested on the BOLTFOR entry. A preload of 0.0 always satisfies this, but it is also possible for the preload to be nonzero, yet A and E are large enough for the software to determine that the zero preload convergence checking should be used.

10. SOL 401 supports the sparse direct solver (default), the element iterative solver, and the PARDISO solver (NLTRD3 nonlinear solution module). To select the SOL 401 solver type, define the SOLVER parameter as SPARSE (default), ELEMITER,

or PARDISO. When the element iterative solver is selected, you can optionally define the SMETHOD case control command and the ITER bulk entry to alter the default options available on the ITER entry.

11. If spin softening is requested with the SPINK parameter, and a rotational force is defined with the RFORCE or RFORCE1 entry, if a grid point is selected in the G field on the RFORCE or RFORCE1 entry to define the rotation center, and this grid point is not used in the connectivity of any element, all translational DOF for this grid point must be constrained.

NLPARM

Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	MAXBIS				MAXR		RTOLB		

EXAMPLE:

NLPARM	15	5		ITER					

FIELDS:

Field	Contents
ID	Identification number. (Integer > 0)
NINC	Number of increments. See Remark 16. (0 < Integer < 1000; Default=10 with SOL 106; Default =1 with SOLs 153 and 159 when ANALYSIS case control command = “HEAT”)
DT	Incremental time interval for creep analysis. See Remark 3. (Real ≥ 0.0; Default = 0.0 for no creep.)
KMETHOD	Method for controlling stiffness updates. See Remark 4. (Character = “AUTO”, “ITER”, or “SEMI”; Default = “AUTO”.)
KSTEP	Number of iterations before the stiffness update for ITER method. See Remark 5. (Integer ≥ 1; Default = 5)
MAXITER	Limit on number of iterations for each load increment. See Remark 6. (Integer > 0; Default = 25)

Field	Contents
CONV	Flags to select convergence criteria. See Remark 7. (Character = “U”, “P”, “W”, or any combination; Default = “PW”.)
INTOUT	Intermediate output flag. See Remark 8. (Character = “YES”, “NO”, or “ALL”; Default = NO)
EPSU	Error tolerance for displacement (U) criterion. See Remark 16. (Real > 0.0; Default = 1.0E-2;)
EPSP	Error tolerance for load (P) criterion. See Remark 16. (Real > 0.0; Usual default = 1.0E-2)
EPSW	Error tolerance for work (W) criterion. See Remark 16. (Real > 0.0; Usual default = 1.0E-3)
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 9. (Integer ≠ 0; Default = 3)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 10. (Integer ≥ 0; Default = MAXITER)
MAXLS	Maximum number of line searches allowed for each iteration. See Remark 11. (Integer ≥ 0; Default = 4)
FSTRESS	Fraction of effective stress $\text{FSTRESS} \cdot \bar{\sigma}$ used to limit the subincrement size in the material routines. See Remark 12. (0.0 < Real < 1.0; Default = 0.2)
LSTOL	Line search tolerance. See Remark 11. (0.01 < Real < 0.9; Default = 0.5)
MAXBIS	Maximum number of bisections allowed for each load increment. See Remark 13. (-10 ≤ MAXBIS ≤ 10; Default = 5)
MAXR	Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark 14. (1.0 ≤ MAXR ≤ 40.0; Default = 20.0)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15. (Real > 2.0; Default = 20.0)

REMARKS:

1. The NLPARM entry is selected by the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.

2. In cases of static analysis (DT = 0.0) using Newton methods, NINC is the number of equal subdivisions of the load change defined for the subcase. Applied loads, gravity loads, temperature sets, enforced displacements, etc., define the new loading conditions. The differences from the previous case are divided by NINC to define the incremental values. In cases of static analysis (DT = 0.0) using arc-length methods, NINC is used to determine the initial arc-length for the subcase, and the number of load subdivisions will not be equal to NINC. In cases of creep analysis (DT > 0.0), NINC is the number of time step increments.

3. The unit of DT must be consistent with the unit used on the CREEP entry that defines the creep characteristics. Total creep time for the subcase is DT multiplied by the value in the field NINC; i.e., DT*NINC.

4. The stiffness update strategy is selected in the KMETHOD field.

   - If the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 9 and 13. and for diverging solutions.

   - If the SEMI option is selected, the program for each load increment (i) performs a single iteration based upon the new load, (ii) updates the stiffness matrix, and (iii) resumes the normal AUTO option.

   - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if KSTEP ≤ MAXITER. However, if KSTEP > MAXITER, stiffness matrix is never updated. Note that the Newton-Raphson iteration strategy is obtained by selecting the ITER option and KSTEP = 1, while the Modified Newton-Raphson iteration strategy is obtained by selecting the ITER option and KSTEP = MAXITER.

5. For AUTO and SEMI options, the stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness. See Remark 7.

6. The number of iterations for a load increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or MAXBIS = 0) and MAXDIV is positive, the best attainable solution is

computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated.

7. The test flags (U = displacement error, P = load equilibrium error, and W = work error) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, and/or W) are satisfied upon convergence.

   When KMETHOD=AUTO or SEMI, EPSP is used to detemine if the stiffness is updated, even when P is not included in the CONV field. This can have an effect on convergence or results. See “Update Principles” in Section 3.3.1 of the *NX Nastran Handbook of Nonlinear Analysis*.

8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES or ALL must be specified in order to be able to perform a subsequent restart from the middle of a subcase.

INTOUT	Output Processed
YES	For user-specified load increments.
NO	For the last load of the subcase.
ALL	For every computed and user-specified load increment.

   - For the Newton family of iteration methods (i.e., when no NLPCI command is specified), the option ALL is equivalent to option YES since the computed load increment is always equal to the user-specified load increment.

   - For arc-length methods (i.e., when the NLPCI command is specified) the computed load increment in general is not going to be equal to the user-specified load increment, and is not known in advance. The option ALL allows the user to obtain solutions at the desired intermediate load increments.

   If the YES or ALL options are specified and the output is directed to an .xdb file, an error message is issued and the run terminates. This occurs because the .xdb file is incapable of storing multiple outputs of the same data block. If intermediate results are required, use the YES or ALL option and direct the output to an .op2 file. If intermediate results are not required, use the NO option and direct the output to either an .op2 file or an .xdb file.

9. Displacement convergence is not supported when KMETHOD=ITER and KSTEP=1 are defined (which selects the Full Newton-Raphson method). If displacement convergence is requested ("U" on the CONV field) when KMETHOD=ITER and KSTEP=1 are defined, the program will use work convergence instead (EPSW).

10. The ratio of energy errors before and after the iteration is defined as divergence rate (E^i), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

$$\text{If } E^i \geq 1 \text{ or } E^i < -10^{12}, \text{ then NDIV} = \text{NDIV} + 2$$

$$\text{If } -10^{12} < E^i < -1, \text{ then NDIV} = \text{NDIV} + 1$$

The solution is assumed to diverge when NDIV ≥ |MAXDIV|. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIV is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated on the second divergence.

11. The BFGS update is performed if MAXQN > 0. As many as MAXQN quasi-Newton vectors can be accumulated. The BFGS update with these QN vectors provides a secant modulus in the search direction. If MAXQN is reached, no additional ON vectors will be accumulated. Accumulated QN vectors are purged when the stiffness is updated and the accumulation is resumed.

12. The line search is performed as required, if MAXLS > 0. In the line search, the displacement increment is scaled to minimize the energy error. The line search is not performed if the absolute value of the relative energy error is less than the value specified in LSTOL.

    EPSW is used to detemine if line search is performed, even when W is not included in the CONV field. This can have an effect on convergence or results. See “Other Provisions for Line Search” in section 3.4.5 of the *NX Nastran Handbook of Nonlinear Analysis*.

13. The number of subincrements in the material routines (elastoplastic and creep) is determined so that the subincrement size is approximately

    $\text{FSTRESS} \cdot \bar{\sigma}$ (equivalent stress).

    FSTRESS is also used to establish a tolerance for error correction in the elastoplastic material; i.e.,

$$\text{error in yield function} < \text{FSTRESS} \cdot \bar{\sigma}$$

If the limit is exceeded at the converging state, the program will exit with a fatal message. Otherwise, the stress state is adjusted to the current yield surface.

14. The number of bisections for a load increment/arc-length is limited to the absolute value of MAXBIS. Different actions are taken when the solution diverges depending on the sign of MAXBIS. If MAXBIS is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If MAXBIS is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after |MAXBIS| bisections, the analysis is continued or terminated depending on the sign of MAXDIV. See Remark 9.

15. MAXR is used in the adaptive load increment/arc-length method to define the overall upper and lower bounds on the load increment/arc-length in the subcase; i.e.,

$$\frac{1}{\text{MAXR}} \leq \frac{\Delta l_n}{\Delta l_o} \leq \text{MAXR}$$

where Δl_n is the arc-length at step n and Δl_o is the original arc-length. The arc-length method for load increments is selected by an NLPCI Bulk Data entry. This entry must have the same ID as the NLPARM Bulk Data entry.

16. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$, or $\Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.

17. Default tolerance sets are determined based on model type and desired accuracy. Accuracy is under user control and can be specified on the PARAM, NLTOL entry. NLTOL's value is used only if the CONV, EPSU, EPSP and EPSW fields are blank, and if NINC is set to a value of 10 or larger. Otherwise, the NLTOL selection will be overridden. The tables below list tolerances according to NLTOL selections:

Table 15-12. Default Tolerances for Static Nonlinear SOL 106 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
1	High	PW	_____	1.0E-2	1.0E-3
2	Engineering	PW	_____	1.0E-2	1.0E-2
3	Prelim Design	PW	_____	1.0E-1	1.0E-1
None	Engineering	PW	_____	1.0E-2	1.0E-2

Table 15-13. Default Tolerances for Static Nonlinear SOL 106 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
2	Engineering	PW	_____	1.0E-3	1.0E-5
None	Engineering	PW	_____	1.0E-3	1.0E-5

Table 15-14. Default Tolerances for Static Nonlinear SOL 106 or 153 Models With Heat Transfer (Enter NLTOL Value of 0 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
None	Very high	PW	_____	1.0E-3	1.0E-7

NLPCI

Parameters for Arc-Length Methods in Nonlinear Static Analysis

Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106). This entry will be used if a subcase contains an NLPARM command (NLPARM = ID).

FORMAT:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE		DESITER	MXINC	

EXAMPLE:

NLPCI	10	CRIS	1.0	1.0			12	10	

FIELDS:

Field	Contents
ID	Identification number of an associated NLPARM entry. (Integer > 0)
TYPE	Constraint type. See Remark 2. (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3 and 4. (0.0 < Real ≤ 1.0; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3 and 4. (Real ≥ 1.0; Default = 4.0)
SCALE	Scale factor (w) for controlling the loading contribution in the arc-length constraint. (Real ≥ 0.0; Default = 0.0)

Field	Contents
DESITER	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. See Remarks 3 and 4. (Integer > 0; Default = 12)
MXINC	Maximum number of controlled increment steps allowed within a subcase. See Remark 5. (Integer > 0; Default = 20)

REMARKS:

1. The NLPCI entry is selected by the Case Control command NLPARM = ID. There must also be an NLPARM entry with the same ID. However, for creep analysis (DT ≠ 0.0 in NLPARM entry), the arc-length methods cannot be activated, and the NLPCI entry is ignored if specified. The NLPCI entry is not recommended for heat transfer analysis in SOL 153.

2. The available constraint types are as follows:

   TYPE = “CRIS”:

$$\left\{u_n^i - u_n^0\right\}\left\{u_n^i - u_n^0\right\} + w^2(\mu^i - \mu^0)^2 = \Delta l_n^2$$

   TYPE = “RIKS”:

$$\left\{u_n^i - u_n^{i-1}\right\}^T\left\{u_n^1 - u_n^0\right\} + w^2\Delta\mu^i = 0$$

   TYPE = “MRIKS”:

$$\left\{u_n^i - u_n^{i-1}\right\}^T\left\{u_n^{i-1} - u_n^0\right\} + w^2\Delta\mu^i(\mu^{i-1} - \mu^0) = 0$$

   where:

   w = the user-specified scaling factor (SCALE)

μ	=	the load factor
Δl	=	the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (*w*) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor μ. As the value of *w* is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite *w*, the arc-length method is degenerated to the conventional Newton's method.

3. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one load increment to the next by:

$$\text{MINALR} \leq \frac{\Delta l_{\text{new}}}{\Delta l_{\text{old}}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use MINALR = MAXALR = 1.

4. The arc-length Δl for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous load increment (I_{max}) and the number of iterations desired for convergence in the current load increment (DESITER) as follows:

$$\Delta l_{new} = \sqrt{\frac{\text{DESITER} \cdot \Delta l_{old}}{I_{max}}}$$

5. The MXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. This field is useful in limiting the number of increments computed for a collapse analysis.

NLRGAP

Nonlinear Transient Load Proportional to Gap

Defines a nonlinear transient radial (circular) gap.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NLRGAP	SID	GA	GB	PLANE	TABK	TABG	TABU	RADIUS	

EXAMPLE:

NLRGAP	21	3	4	XY	3	10	6	1.6	

FIELDS:

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GA	Inner (e.g., shaft) grid for radial gap. (Integer > 0)
GB	Outer (e.g., housing) grid for radial gap. (Integer > 0)
PLANE	Radial gap orientation plane: XY, YZ, or ZX. (Character, Default = XY.)
TABK	Table ID of gap stiffness vs. time. (Integer > 0)Table ID of gap stiffness vs. penetration. (Integer < 0)
TABG	Table ID for radial gap clearance as function of time. (Integer > 0)
TABU	Table ID for radial coefficient of friction as function of time. (Integer > 0)
RADIUS	Shaft radius. (Real ≥ 0.0, Default = 0.0)

REMARKS:

1. NLRGAP must be selected with the Case Control command NONLINEAR = SID.

2. Multiple NLRGAP entries with the same SID are allowed.

3. The NLRGAP is not an element, but a nonlinear load similar to the NOLINi Bulk Data entries. It computes the relative displacements of GA and GB in the selected plane and applies appropriate nonlinear loads to simulate the radial contact.

4. The degrees-of-freedom in the XY, YZ, or ZX planes (depending on the PLANE) of GA and GB must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation. If RADIUS is > 0.0, then the in-plane rotation degree-of-freedom must also be in the solution set.

5. As with the NOLINi entries, the NLRGAP is limited to use in direct transient response solution sequences.

6. The XY, YZ and ZX planes are relative to the displacement coordinates systems of GA and GB. GA and GB should be coincident grids with parallel displacement coordinate systems. NX Nastran does not check or enforce this. Wrong answers can occur if this rule is not followed.

7. If TABK is negative, the corresponding TID on the referenced TABLEDi entry is input positive and not negative.

8. The shaft radius is used only for the computation of friction induced torque.

9. In the underlying equations, a positive coefficient of friction is consistent with counter-clockwise shaft rotation from axis 1 towards axis 2 (anti-clockwise). A negative coefficient of friction is consistent with clockwise shaft rotation from axis 2 towards axis 1 (clockwise). See Figure 15-6.

10. Nonlinear forces for the grids referenced on the NLRGAP can be output with the NLLOAD Case Control command. See Figure 15-6 for the sign conventions.

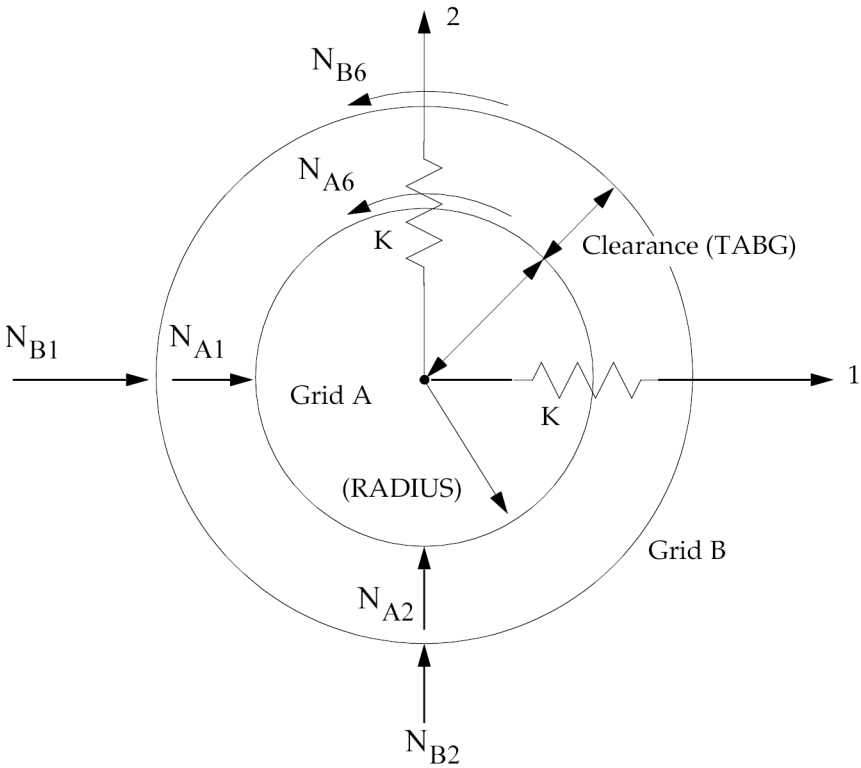


Figure 15-6. Radial Gap Orientation and Nonlinear Load Sign Conventions

NOLIN1

Nonlinear Transient Load as a Tabular Function

Defines nonlinear transient forcing functions of the form

$$\text{Function of displacement: } P_i(t) = S \cdot T(u_j(t))$$

$$\text{Function of velocity: } P_i(t) = S \cdot T(\dot{u}_j(t))$$

where $u_j(t)$ and $\dot{u}_j(t)$ are the displacement and velocity at point GJ in the direction of CJ.

$u_j(t)$ is temperature in SOL 159.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

EXAMPLE:

NOLIN1	21	3	4	2.1	3	10	6		

FIELDS:

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)

Field	Contents
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

TID	Identification number of a TABLEDi entry. (Integer > 0)

REMARKS:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN1 entries must be members of the solution set. This means the e-set (EPOINT entry) for modal formulation and the d-set for direct formulation.

4. Nonlinear loads as a function of velocity (Equation 2) are denoted by components ten greater than the actual component number; i.e., a component of 11 is component 1 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. See the *Advanced Dynamic Analysis User's Guide* for information on using NOLINi bulk entries in solutions 109 and 112. See the *Handbook of Nonlinear Analysis* for information on using NOLINi bulk entries in solution 129. See the *Thermal Analysis User's Guide* for information on using NOLINi bulk entries in solution 159.

NOLIN2

Nonlinear Transient Load as the Product of Two Variables

Defines nonlinear transient forcing functions of the form

$$P_i(t) = S \cdot X_j(t) \cdot X_k(t)$$

where $X_j(t)$ and $X_k(t)$ can be either displacement or velocity at points GJ and GK in the directions of CJ and CK.

$X_j(t)$ and $X_k(t)$ are temperatures in SOL 159.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

EXAMPLE:

NOLIN2	14	2	1	2.9	2	1	2		

FIELDS:

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ, GK	Grid, scalar, or extra point identification number. (Integer > 0)

Field	Contents
CJ, CK	Component number for GJ, GK according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

REMARKS:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR=SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN2 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.

4. GI-CI, GJ-CJ, and GK-CK may be the same point.

5. Nonlinear loads may be a function of displacement ($X = u$) or velocity $(X = \dot{u})$. Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_t = \frac{u_t - u_{t-1}}{\Delta t}$$

   where Δt is the time step interval and u_{t-1} is the displacement of GJ-CJ or GK-CK for the previous time step.

6. See the *Advanced Dynamic Analysis User's Guide* for information on using NOLINi bulk entries in solutions 109 and 112. See the *Handbook of Nonlinear Analysis* for information on using NOLINi bulk entries in solution 129. See the *Thermal Analysis User's Guide* for information on using NOLINi bulk entries in solution 159.

NOLIN3

Nonlinear Transient Load as a Positive Variable Raised to a Power

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} S \cdot [X_j(t)]^A, & X_j(t) > 0 \\ 0, & X_j(t) \le 0 \end{cases}$$

where $X_j(t)$ may be a displacement or a velocity at point GJ in the direction of CJ. $X_j(t)$ is temperature in SOL 159.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

EXAMPLE:

NOLIN3	4	102		-6.1	2	15	-3.5		

FIELDS:

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which the nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)

Field	Contents
GJ	Grid, scalar, extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

Field	Contents
A	Exponent of the forcing function. (Real)

REMARKS:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN3 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.

4. Nonlinear loads may be a function of displacement (X_j=u_j) or velocity (X_j=u_j). Velocities are denoted by a component number ten greater than the actual component number; e.g., a component of 16 is component 6 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN4 entry for the negative range of X_j (t).

6. See the *Advanced Dynamic Analysis User's Guide* for information on using NOLINi bulk entries in solutions 109 and 112. See the *Handbook of Nonlinear Analysis* for information on using NOLINi bulk entries in solution 129. See the

Thermal Analysis User's Guide for information on using NOLINi bulk entries in solution 159.

NOLIN4

Nonlinear Transient Load as a Negative Variable Raised to a Power

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} -S \cdot [-X_j(t)]^A, & X_j(t) < 0 \\ 0, & X_j(t) \geq 0 \end{cases}$$

where $X_j(t)$ may be a displacement or a velocity at point GJ in the direction of CJ. $X_j(t)$ is temperature in SOL 159.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

EXAMPLE:

NOLIN4	2	4	6	2.0	101		16.3		

FIELDS:

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)

Field	Contents
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

Field	Contents
A	Exponent of forcing function. (Real)

REMARKS:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.

2. Nonlinear loads may not be referenced on a DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN4 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.

4. Nonlinear loads may be a function of displacement (X_j=u_j) or velocity X_j=u_j). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,t-1}$ is the displacement of GJ-CJ for the previous time step.

5. Use a NOLIN3 entry for the positive range of $X_j(t)$.

6. See the *Handbook of Nonlinear Analysis* for more information on NOLINi bulk entries.

7. See the *Advanced Dynamic Analysis User's Guide* for information on using NOLINi bulk entries in solutions 109 and 112. See the *Handbook of Nonlinear Analysis* for information on using NOLINi bulk entries in solution 129. See the *Thermal Analysis User's Guide* for information on using NOLINi bulk entries in solution 159.

NSM

Define Non-Structural Mass

Defines a set of non-structural mass by ID.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NSM	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	ID	VALUE	etc.				

EXAMPLE:

NSM	11	PSHELL	16	0.25					

FIELDS:

Field	Contents
SID	Nonstructural mass set identification number. (Integer > 0)
TYPE	Set points to either property entries or element entries. Properties are: PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PCOMP, PCOMPG, PROD, CONROD, PBEND, PSHEAR, PSHELL, PTUBE, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	Nonstructural mass per unit area for 2D elements, or nonstructural mass per unit length for 1D elements. (Real)

REMARKS:

1. Nonstructural mass sets must be selected with Case Control command NSM = SID.

2. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSM1

Define Non-Structural Mass

Alternate form for NSM entry. Defines non-structural mass entries by VALUE, ID list.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.					

EXAMPLE:

NSM1	11	ELEMENT	0.46	1243	1532				

ALTERNATE FORMS:

NSM1	SID	TYPE	VALUE	ID	THRU	ID			

NSM1	SID	TYPE	VALUE	ALL					

NSM1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	

FIELDS:

Field	Contents
SID	Nonstructural mass set identification number. (Integer > 0)

Field	Contents
TYPE	Set points to either property entries or element entries. Properties are: PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PCOMP, PCOMPG, PROD, CONROD, PBEND, PSHEAR, PSHELL, PTUBE, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0 or “ALL” or “THRU” or “BY” or N (the BY increment))
VALUE	Nonstructural mass per unit area for 2D elements, or nonstructural mass per unit length for 1D elements. (Real)

REMARKS:

1. Non structural mass sets must be selected with Case Control command NSM = SID.

2. Defining TYPE = PSHELL, PCOMP, or PCOMPG applies mass to all of these types.
   Defining TYPE = PBEAM, PBEAML, or PBCOMP applies mass to all of these types.
   Defining TYPE = PBAR or PBARL applies mass to both of these types.

   For example, the input “NSML1,12,PSHELL,12.5, ALL” will apply nonstructural mass to the elements associated to all PSHELL, all PCOMP, and all PCOMPG properties.

3. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSMADD

Define Non-Structural Mass Set Combination

Non-structural mass set combination. Defines non-structural mass as the sum of the sets listed.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NSMADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	S10	etc.					

EXAMPLE:

NSMADD	7	30	31	32	33				

FIELDS:

Field	Contents
SID	Nonstructural mass set identification number. (Integer > 0)
Si	Identification numbers of non structural mass sets defined via NSM and NSM1 entries. (Integer > 0)

REMARKS:

1. The nonstructural mass sets must be selected with the Case Control command NSM = SID.
2. No Si may be the identification number of a non structural mass set defined by another NSMADD entry.
3. The NSMADD bulk entry take precedence over NSM, NSM1, NSM1, or NSML1 entries. If both have the same set ID, only the NSMADD entry will be used.

4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSML

Define Lumped Non-Structural Mass

Defines a set of lumped non-structural mass by ID.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NSML	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	ID	VALUE	etc.				

EXAMPLE:

NSML	23	PSHELL	6	0.22					

FIELDS:

Field	Contents
SID	Nonstructural mass set identification number. (Integer > 0)
TYPE	Set points to either property entries or element entries. Properties are: PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PCOMP, PCOMPG, PROD, CONROD, PBEND, PSHEAR, PSHELL, PTUBE, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	A lumped mass value to be distributed. See Remark 2. (Real)

REMARKS:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.

2. VALUE is the total nonstructural mass to be distributed across *all* IDs.

   - Area element calculation (for example, CQUAD4):
     Mass for a particular element = (Element area) x (VALUE / ∑Element area for all IDs)

   - Line element calculation (for example, CBEAM):
     Mass for a particular element = (Element length) x (VALUE / ∑Element length for all IDs)

3. Non structural mass sets must be selected with the case control command NSM = SID.

4. The ELSUM case control command will give a summary of both structural and nonstructural mass by element or property type.

NSML1

Define Lumped Non-Structural Mass

Alternate form for NSML entry. Defines lumped non-structural mass entries by VALUE, ID list.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.					

EXAMPLE:

NSML1	13	ELEMENT	0.72	156	286				

ALTERNATE FORMS:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID1	THRU	ID2			

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ALL					

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID1	THRU	ID2	BY	N	

FIELDS:

Field	Contents
SID	Nonstructural mass set identification number. (Integer > 0)

Field	Contents
TYPE	Set points to either property entries or element entries. Properties are: PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PCOMP, PCOMPG, PROD, CONROD, PBEND, PSHEAR, PSHELL, PTUBE, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0 or “ALL” or “THRU” (ID1 < ID2 for “THRU” option) or “BY” or N (N > 0))
VALUE	A lumped mass value to be distributed. See Remark 2. (Real)

REMARKS:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.

2. VALUE is the total nonstructural mass to be distributed across *all* IDs.

   - Area element calculation (for example, CQUAD4):
     Mass for a particular element = (Element area) x (VALUE / ∑Element area for all IDs)

   - Line element calculation (for example, CBEAM):
     Mass for a particular element = (Element length) x (VALUE / ∑Element length for all IDs)

3. Alternate input forms using THRU, BY, and N can be included on different continuation lines except for ALL. ALL can only be used on a single line input.

4. Non structural mass sets must be selected with the case control command NSM = SID.

5. Defining TYPE = PSHELL, PCOMP, or PCOMPG applies mass to all of these types.
   Defining TYPE = PBEAM, PBEAML, or PBCOMP applies mass to all of these types.
   Defining TYPE = PBAR or PBARL applies mass to both of these types.

   For example, the input “NSML1,12,PSHELL,12.5, ALL” will apply nonstructural mass to the elements associated to all PSHELL, all PCOMP, and all PCOMPG properties.

6. The ELSUM case control command will give a summary of both structural and nonstructural mass by element or property type.

NXSTRAT

Strategy Parameters for SOLs 601 and 701

Defines parameters for solution control and strategy in advanced nonlinear structural analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
NXSTRAT	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	-etc-				

EXAMPLE:

NXSTRAT	1	AUTO	1	MAXITE	30	RTOL	0.005		
	ATSNEXT	3							

FIELDS:

Field	Contents
ID	Identification number. Currently not used. (Integer > 0)
PARAMi	Name of the NXSTRAT parameter. Allowable names are given in the parameter listing below. See Remark 4 for a table summarizing the supported solutions for each parameter. (Character)
VALUEi	Value of the parameter. (Real or integer)

NXSTRAT PARAMETERS

Analysis Control

Name	Description
SOLVER	Selects the solver to use. (Integer; Default = 0) 0 – Direct sparse solver 1 – Multigrid solver 2 – 3D iterative solver. This solver is effective for models with large numbers of higher order 3D solid elements, i.e., CTETRA and CHEXA elements with mid-side nodes.
AUTO	Indicates whether automatic incrementation scheme is enabled. (Integer; Default = 0) 0 – No automatic incrementation scheme is used 1 – Automatic time stepping (ATS) scheme is enabled 2 – Automatic load-displacement control (LDC) scheme is enabled 3 – Total load application (TLA) scheme is enabled. Program ignores any time step (TSTEP) and time function (TABLEDi) specified. Instead, 50 time steps of size 0.2 is used with a linear ramp time function to reach a load factor of 1.0 at time of 10.0. Hence, loads selected with the LOAD case control and loads selected with the DLOAD case control are ramped up in the same way. In addition, the follow settings are used: • maximum number of equilibrium iterations is 30 • the smallest time step size is 1/64 of initial step size • line search is used • maximum displacement in each iteration is 5% of maximum model dimension 4 – Total load application with stabilization (TLA-S) scheme is enabled. In addition to TLA, the following stabilization is used: • matrix stabilization factor of 1.0e-10 • low speed dynamics • contact damping

Analysis Control

Name	Description
	Note: See sections below for parameters that may be specified for each of these schemes. Although the TLA and TLA-S schemes include the use of the automatic time stepping (ATS) scheme, the ATS set of parameters do not apply directly. Instead, the TLA set of parameters should be used to control the TLA or TLA-S schemes.
NPOSIT	Indicates whether analysis continues when the system matrix is not positive definite. (Integer; Default = 0) 0 – Analysis may stop 1 – Analysis continues Notes: If NPOSIT=0, analysis stops unless • AUTO > 0 is specified • contact analysis is being performed. It is not recommended to set NPOSIT=1 for a linear analysis.
MASSTYP	Selects the type of mass matrix to be used in dynamic analysis. (Integer; Default = 1) 0 – Lumped mass is used 1 – Consistent mass is used

Analysis Options

Name	Description
TINT	Integration order for the local t-direction (through thickness) of shell elements with shape memory alloy or elasto-plastic materials. By default, 5-point Newton-Cotes is used for single-layered shell and 3-point Newton-Cotes is used for multi-layered shell. Note that 2-point Gauss integration is always used for all shell elements with elastic materials. (Integer; Default = 0). 1 ≤ TINT ≤ 6 – Gauss integration method with integration order TINT -3, -5, -7 – Newton-Cotes integration with order -TINT

Analysis Options

Name	Description
ICMODE	Indicates whether incompatible modes are used for 4-node shell elements, plane strain and plane stress elements. (Integer; Default =1 for SOL 601 and 0 for SOL 701) 0 - Incompatible modes are not used 1 - Incompatible modes are used Note: In restarting from SOL 601 to 701 or vice versa, ICMODE needs to be specified such that both analyses use the same option.
MSTAB	Indicates whether the stiffness matrix stabilization feature is used. (Integer; Default = 0) 0 – Matrix stabilization is not used 1 – Matrix stabilization is used 2 – Matrix stabilization is automatically used if the ratio of maximum/minimum diagonal of factorized matrix is greater than 1.0E10.
MSFAC	Matrix stabilization factor. (Real; Default = 1.0E-10; Unitless)
DTDELAY	Element death time delay. (Real; Default = 0.0; Unit = Time) When an element is too deformed and becomes “dead”, its contribution to the overall stiffness of the structure is removed. By specifying DTDELAY > 0.0, the contribution from the element stiffness is gradually reduced to zero over time DTDELAY instead of being suddenly removed. This may help in the convergence of the solution.
SDOFANG	Angle used to determine whether a shell mid-surface node is assigned 5 or 6 degrees of freedom. (Real; Default = 5.0; Unit = Angle in degrees)
DRILLKF	On shell grids where the drilling stiffness is zero, this factor is multiplied by the maximum rotational stiffness at the grid and assigned as the drilling stiffness. (0.0 < Real < 1.0; Default = 1.0E-4; Unitless)
UPFORM	Indicates whether u/p formulation is used for elements. Note that u/p formulation is always used for hyperelastic elements and always not used for hyperfoam elements and elastic elements with Poisson’s ratio less than 0.48. It is also not used for gasket elements (Integer; Default = 0) 0 – u/p formulation is not used 1 – u/p formulation is used instead of displacement-based formulation

Analysis Options

Name	Description
ULFORM	Indicates which large strain formulation is used. (Integer; Default = 0) 0 – Updated Lagrangian-Hencky (ULH) formulation is used for SOL 601 and Updated Lagrangian-Jaumann (ULJ) formulation is used for SOL 701. 1 – Use ULH formulation 2 – Use ULJ formulation Note: In restarting from SOL 601 to 701 or vice versa, ULFORM needs to be specified such that both analyses use the same formulation.
DISPOPT	Indicates whether prescribed displacements are applied to the original configuration or the deformed configuration. This option is only applicable for a restart analysis or when a delay (or arrival) time is specified for the prescribed displacement. (Integer; Default = 0) 0 – Applied to original configuration 1 – Applied to deformed configuration
LOADOPT	Indicates whether prescribed loads (pressure and centrifugal) are deformation-dependent, i.e. the direction and magnitude of the load may change due to large deformation of the structure. This option is only applicable for large displacement analysis, i.e. PARAM,LGDISP,1 (Integer; Default = 1) 0 – Load is independent of structural deformation 1 – Load is affected by structural deformation
MAXDISP	Specifies a limit for the maximum incremental displacement that is allowed for any grid in any equilibrium iteration. This feature is generally useful for contact analysis where rigid body motion exists in a model. A value of 0.0 means there is no limit on incremental displacements for dynamic analysis or analysis without contact. For static analysis with contact, the maximum incremental displacement is 1% of the maximum model dimension if MAXDISP=0.0. (Real ≥ 0.0; Default = 0.0; Unit = Length)
BEAMALG	Selects the beam algorithm to use for elastic beam formulations. Note that the current algorithm is always used for elasto-plastic beams. (Integer; Default = 0) 0 – Use the current algorithm 1 – Use the old (v8.5) algorithm

Time Integration

Name	Description
TINTEG	Selects the time integration method to be used for nonlinear transient analysis. (Integer; Default = 0) 0 – Use the Newmark method 1 – Use the Bathe method
ALPHA	Alpha coefficient for the Newmark time integration method. (Real; Default = 0.25; Unitless)
DELTA	Delta coefficient for the Newmark method. (Real; Default = 0.5; Unitless)

SOL 701 Time Stepping

Name	Description
XSTEP	Selects time step method used in an explicit time integration analysis. (Integer; Default = 0) 0 - Time step size is calculated by the program based on the critical time step size. The data in the selected TSTEP bulk data entry is used to calculate the total solution time for the analysis. 1 - The number of time steps and the time step size as specified in the selected TSTEP bulk data entry is used.
XDTCAL	Calculation of the critical time step size may be computationally expensive. This parameter specifies that the critical time step size will be re-calculated every XDTCAL time steps. (Integer > 0, Default = 1)
XDTFAC	The critical time step size is calculated based on certain assumptions. It is often necessary, especially for nonlinear analysis, to use a time step size smaller than the calculated critical time step size. The factor multiplied by the calculated critical time step size gives the time step size used in the analysis. (4.0 > Real > 0.0; Default = 0.9; Unitless)
XMSCALE	Specifies the factor to scale the mass (densities) of the entire model (at the beginning of the analysis) to increase the critical time step size required for stability when the explicit time integration scheme is used. See warning in Remark 2. (Real = 1.0, Default = 1.0; Unitless)

SOL 701 Time Stepping

Name	Description
XDTMIN1	The minimum time step size used to determine if mass scaling will be applied to elements (at the beginning of the analysis) whose critical time step size is smaller than DTMIN1. The amount of mass scaling is calculated for each element so that the critical time step size is equal to DTMIN1. See Remark 2 and warning in Remark 3. (Real = 0.0, Default = 0.0; Unit = Time)
XDTMIN2	The minimum time step size used to determine whether an element will be removed in an explicit time integration analysis. In explicit time integration, the smaller an element size is, the smaller will the critical time step size be. If the critical time step size for an element is smaller than XDTMIN2, the element will be removed in the analysis. See Remark 2 and warning in Remark 3. (Real > 0.0, Default = 0.0; Unit = Time)

Iterative Solver

Name	Description
ITEMAX	Maximum number of iterations allowed for the multigrid or 3D-iterative solver to converge. (Integer > 0; Default = 1000 for multigrid solver and 200 for 3D-iterative solver)
EPSIA	Convergence tolerance EPSIA. (Real; Default = 1.0E-6; Unitless)
EPSIB	Convergence tolerance EPSIB. (Real; Default = 1.0E-4; Unitless)
EPSII	Convergence tolerance EPSII. (Real; Default = 1.0E-8; Unitless)

Equilibrium Iteration and Convergence

Name	Description
LSEARCH	Flag to indicate the use of line searches within the iteration scheme. (Integer; Default = 0) 0– Line search is not used 1 – Line search is used
LSLOWER	Lower bound for line search. (0.0 ≤ Real < 1.0; Default = 0.001; Unitless)

Equilibrium Iteration and Convergence

Name	Description
LSUPPER	Upper bound for line search. (1.0 ≤ Real; Default = 1.0 for contact analysis and 8.0 for analysis with no contact; Unitless)
PLASALG	Selects the algorithm used in plasticity. (Integer; Default = 1) 1 – Algorithm 1 is used 2 – Algorithm 2 is used Note: For a given time step size, if the iterations do not converge with algorithm 1 because the Jacobian determinant in the elements becomes non-positive, switching to algorithm 2 can sometimes enable convergence. When algorithm 2 is used, additional convergence check is performed. Between two successive iterations, if the state of stress changes (e.g., from elastic to plastic or from plastic to elastic) at any integration point of an element, convergence is prevented.
MAXITE	Maximum number of iterations within a time step. If the maximum number of iterations is reached without achieving convergence (see CONVCRI parameter), the program will stop unless the automatic time stepping (ATS) or load displacement control scheme is selected (see parameter AUTO). (1 ≤ Integer ≤ 999; Default = 15)
CONVCRI	Convergence Criteria. (Integer; Default = 0) 0 – Convergence based on energy 1 – Convergence based on energy and force 2 – Convergence based on energy and displacement 3 – Convergence based on force 4 – Convergence based on displacement
ETOL	Relative energy tolerance. (Real; Default = 0.001 if AUTO≠2; Default = 1.0e-6 if AUTO=2; Unitless)
RTOL	Relative force (and moment) tolerance (Real; Default = 0.01; Unitless)
RNORM	Reference force. (Real; Default = 0.0; Unit = Force). The reference force is automatically calculated by the program if RNORM=0.0.
RMNORM	Reference moment. (Real; Default = 0.0; Unit = Force * Length). The reference moment is automatically calculated by the program if RMNORM=0.0.
RCTOL	Relative contact force tolerance. (Real; Default = 0.05; Unitless)

Equilibrium Iteration and Convergence

Name	Description
DTOL	Relative displacement (translation and rotation) tolerance. (Real; Default = 0.01; Unitless)
DNORM	Reference translation. (Real; Default = 0.0; Unit = Length). The reference translation is automatically calculated by the program if DNORM=0.0.
DMNORM	Reference rotation. (Real; Default = 0.0; Unit = Angle in radians). The reference rotation is automatically calculated by the program if DMNORM=0.0.
STOL	Line search convergence tolerance. (Real; Default = 0.5; Unitless)
RCONSM	Reference contact force. (Real; Default = 0.01; Unit = Force)
ENLSTH	Line search energy threshold. (Real; Default = 0.0; Unit = Force * Length)

Automatic Time Stepping (ATS) Scheme

Name	Description
ATSSUBD	Number that limits the smallest time step size when the automatic time stepping (ATS) scheme is used. For a time step size of DT, the program will stop if convergence is not achieved and the next subdivided time step size is less than DT/ATSSUBD. (Integer ≥ 1; Default = 10)
ATSMXDT	Factor that limits the maximum time step size when the automatic time stepping (ATS) scheme is used. In a static analysis with ATSNEXT = 2, ATSLOWS = 0, only one time step block (i.e., TSTEP entry has only one line), and there is no temperature or temperature gradient loading, the ATS scheme may increase the time step size after convergence is achieved. In such a case, for a time step size of DT, the program will not use a time step size greater than ATSMXDT * DT. (Real; Default = 3.0; Unitless)
ATSMASS	Inertia factor used in low-speed dynamics analysis. (0.0 ≤ Real ≤ 1.0; Default = 1.0)

Automatic Time Stepping (ATS) Scheme

Name	Description
ATSNEXT	Flag controls what time step size to use once convergence is reached after an ATS subdivision. (Integer; Default = 0) 0 – Automatically set by program. For contact analysis, ATSNEXT = 2, otherwise ATSNEXT = 1. 1 – Use the time step size that gave convergence, i.e., the reduced time step that led to convergence is used again. 2 – Return to the original time step size, i.e., the original time step size before any subdivision took place is used. 3 – Use a time step size such that the solution time matches the original solution time specified by the user.
ATSDFAC	Division factor used calculate the sub-increment time step size. If current time step size is DT and convergence is not achieved, the next time step size will be DT/ATSDFAC. (Real > 1.0; Default = 2.0; Unitless)
ATSLOWS	Flag whether a low-speed dynamics analysis is performed instead of a static analysis. (Integer; Default = 0) 0 – Low-speed dynamics option is not activated 1 – Low-speed dynamics is performed
ATSDAMP	Damping factor used in low-speed dynamics analysis. (Real ≥ 0.0; Default = 1.0e-4; Unit = Time)

Load Displacement Control (LDC) Scheme

Name	Description
LDCGRID	Grid point id at which a displacement is prescribed for the first solution step. (Integer > 0)

Load Displacement Control (LDC) Scheme

Name	Description
LDCDOF	Degree of freedom in the displacement coordinate system for a prescribed displacement at grid point LDCGRID. The displacement coordinate system of a grid point is defined with the CD field on the GRID entry. (1≤ Integer ≤ 6) 1 – X translation 2 – Y translation 3 – Z translation 4 – X rotation 5 – Y rotation 6 – Z rotation
LDCDISP	Prescribed displacement at grid point LDCGRID for the first solution step. (Real; Unit = Length)
LDCIMAX	Displacement convergence factor used to limit the maximum incremental displacement during a solution step. If the incremental displacements exceed (100 * LDCIMAX * displacements in the first step), the current solution step will be repeated with a reduced load factor. (Real; Default = 3.0; Unitless)
LDCDMAX	Maximum (absolute magnitude) displacement (for the degree of freedom specified by LDCDOF) at the grid point LDCGRID allowed during the analysis. When the displacement reaches or exceeds LDCDMAX, the program will stop the analysis. See Section 6.2.4 in Advanced Nonlinear Theory and Modeling Guide for other criteria that determines when an LDC solution will stop. (Real; Unit = Length)
LDCCONT	Flag whether the solution is terminated when the first critical point on the equilibrium path is reached. (Integer; Default = 0) 0 – Solution stops 1 – Solution continues
LDCSUBD	Maximum number of arc length subdivisions allowed. (Integer ≥ 1; Default = 10)

Total Load Application (TLA) Scheme

Name	Description
TLANSTP	Number of time steps to use for the solution. The step size is automatically adjusted to obtain a total time of 10.0. (Integer > 0; Default = 50)
TLAMXIT	Maximum number of equilibrium iterations allowed to achieve convergence in any time step (subdivided or accelerated). (1 ≤ Integer ≤ 999; Default = 30)
TLAMXDF	Maximum displacement factor. The maximum incremental displacement allowed in any time step is equal to TLAMXDF * (maximum model dimension). TLAMXDF = 0.0 means there is no limit on the maximum incremental displacement. (Real ≥ 0.0; Default = 0.05; Unitless)
TLASTBF	Stiffness matrix stabilization factor. If TLASTBF = 0.0, then the stiffness matrix stabilization feature is not used. Applicable only to TLA-S (AUTO=4) scheme. (Real ≥ 0.0; Default = 1.0E-10; Unitless)
TLALSDF	Low-speed dynamics damping factor. If TLALSDF = 0.0, then the low-speed dynamics option is not used. Applicable only to TLA-S (AUTO=4) scheme. (Real ≥ 0.0; Default = 1.0E-4; Unit = Time)
TLALSMF	Low-speed dynamics inertia factor. Applicable only to TLA-S (AUTO=4) scheme. (0.0 ≤ Real ≤ 1.0; Default = 1.0; Unitless)
TLACTDF	Contact damping factor. The amount of contact damping used in the solution is equal to TLACTDF * (damping determined by the program). If TLACTDF = 0.0, then contact damping is not used. Applicable only to TLA-S (AUTO=4) scheme. (Real ≥ 0.0; Default = 0.001; Unitless)

Contact Control

Name	Description
IMPACT	Impact control scheme (Integer; Default = 0) 0 – No special treatment is applied for impact problems 1 – Post impact adjustment of velocities and accelerations is applied 2 – Modified parameters are used in Newmark time integration scheme

Contact Control

Name	Description
NSUPP	Number of iterations for pairing contactor node to target segment. If NSUPP > 0, during the first NSUPP iterations, the pairing target segment is recorded for each contactor node. From iteration NSUPP+1, if a target segment in the recorded list is repeated, it is “frozen” to be the pairing target segment for the remaining equilibrium iterations in that time step. Specifying NSUPP > 0 may help in the convergence for certain problems. (0 ≤ Integer ≤ 99; Default = 0)
CSTYPE	Selects the type of contact segment to use. (Integer; Default = 1) 0 – Use linear contact segment 1 – Use element-based contact segment which gives better contact traction results Note: CSTYPE = 1 is only applicable for contact algorithm TYPE = 0 or XTYPE = 0 or 1 in BCTPARA entry.
CTDISP	Selects the default displacement formulation used for contact analysis. A different formulation may be selected for each individual contact set via BCTPARA entry. (Integer; Default = 0 / 2) 0 or 2 – Use large displacement formulation (contact conditions are updated) (default) 1 – Use small displacement formulation (contact conditions are not updated) CTDISP is a global option since it applies to all contact definitions in the model. If you would like to prevent/allow a specific contact set from updating, the DISP option on the BCTPARA bulk entry can be used. Note: If CTDISP = 1 is selected, the search of target segments for the contactor nodes is performed only at the beginning of the analysis.
RTALG	Selects the rigid-target algorithm to use. (Integer; Default = 0) 0 – Use the current algorithm 1 – Use the old (v4) algorithm. RTALG=1 is retained for backward compatibility only and is not recommended.
FRICALG	Selects the friction algorithm to use. (Integer; Default = 0) 0 – Use the current algorithm 1 – Use the old (v4) algorithm. FRICALG=1 is retained for backward compatibility only and is not recommended.

Contact Control

Name	Description
TNSLCF	Indicates whether tensile consistent contact forces on quadratic 3D contact segments are allowed. (Integer; Default = 0) 0 – Not allowed 1 - Allowed
CTDAMP	Indicates whether stabilization damping is applied and how it is applied for contact analysis. This feature is generally useful when rigid body motion exists in a model. (Integer; Default = 0) 0 – No stabilization damping is applied 1 – Stabilization damping is applied at the first time step only. The specified damping coefficients are applied and ramped down to zero by the end of the first time step. 2 – The specified stabilization damping coefficients are applied at all time steps.
CTDAMPN	Specifies the normal stabilization damping coefficient. (Real ≥ 0.0, Default = 0.0; Unit = Force * Time/Length)
CTDAMPT	Specifies the tangential stabilization damping coefficient. (Real ≥ 0.0, Default = 0.0; Unit = Force * Time/Length)

Restart Options

Name	Description
MODEX	Indicates the mode of execution. (Integer; Default = 0) 0 – Normal analysis run, i.e. not a restart analysis 1 – Restart analysis 2 – Recover results from restart (.res) file without running analysis for any additional time steps. GPFORCE and SPCFORCES cannot be recovered. MODEX=2 is supported only for SOL 601 analysis. The restart (.res) file from a previous run must exist to do a restart analysis or result recovery. The filename and location of the restart file is determined by the “dbs” keyword. By default, dbs points to the current working directory with the prefix of the current job name. The IDs of grids, elements, properties, materials, contact sets, contact pairs and contact regions should be the same

Restart Options

Name	Description
	in both the original and the restart model. Note that keyword scratch=no must be used when running a restart analysis or result recovery.
TSTART	Solution starting time. If MODEX=1 or 2, TSTART must equal a solution time in which data was saved in a previous run. If TSTART = 0.0, the last time step in the restart file is used. (Real, Default = 0.0; Unit = Time)
IRINT	Frequency of saving the analysis results in the restart file. (Integer; Default = 0) 0 – IRINT is set to 1 when implicit time integration is used and set to the number of steps in the first time step block when explicit time integration is used. > 0 – Restart file is overwritten every IRINT time steps < 0 – Restart file is appended every IRINT time steps

Other Parameters

Name	Description
NSUBGRP	Number of sub-groups to divide large number of elements with same property ID into. Normally, elements with same type and property ID are placed into a group. If a group contains more than 1000 elements and NSUBGRP > 1, the elements are placed into NSUBGRP sub-groups for more efficient processing. If NSUBGRP=0, the program automatically divides an element group with more than 9999 elements into sub-groups. (Integer ≥ 0; Default=0)
XTCURVE	Indicates whether the table in TABLES1 entry is extended by linear extrapolation of the two last points. (Integer; Default = 1) 0 – Table is not extended. This option may be used to allow element rupture at the last specified strain value. 1 – Table is extended Note: XTCURVE is only applicable for the multilinear-plastic material model.

Other Parameters

Name	Description
CVSSVAL	Indicates whether the values in TABLES1 entry are converted from engineering stress-strain to true stress-strain. (Integer; Default = 0) 0 – No conversion of stress strain value 1 – Convert engineering stress and strain values to true stress and strain values. Note: CVSSVAL is only applicable for the multilinear-plastic material model.
ELRESCS	Indicates the coordinate system used for the stress output labeled "NONLINEAR STRESSES" on the elements CHEXA, CPENTA, CPYRAM, and CTETRA. (Integer; Default = 0). 0 – Stresses labeled "NONLINEAR STRESSES" are output in the element coordinate system. 1 – Stresses labeled "NONLINEAR STRESSES" are output in the material coordinate system.
BOLTSTP	Number of steps for applying the bolt pre-load force. BOLTSTP may be used to apply the bolt pre-load force incrementally if the solution fails to converge when the total pre-load force is applied in one step. (Integer > 0; Default = 1)

Translation Options

Name	Description
ELCV	Convert 8-node to 9-node quadrilateral (plane strain, axisymmetric and shell) elements and 20-node to 27-node brick elements. Note that this also converts 6-node to 7-node triangular (plane strain and axisymmetric) elements and 10-node to 11-node tetrahedral elements. (Integer; Default = 0) 0 – No conversion of elements 1 – Convert elements as described above; nodal coincidence is not checked against existing nodes and new generated nodes are always created. 2 – Convert elements as described above; nodal coincidence is checked against existing nodes and a new node will not be created at a location if a node already exist at that location. When ELCV=1 or 2, if the corner nodes of an element face include an SPC condition, the nodes internally generated by the software on these faces will also include the same internally generated SPC condition. SPCFORCES are not output on the internally generated grids. Although, the SUMMATION OF

Translation Options

Name	Description
	REACTIONS listed in the .f06 file does include forces from the internally generated grids. The SUMMATION OF REACTIONS should match your applied loads for a converged solution.
EQRBAR	Indicates how RBAR elements are handled. (Integer; Default = 0) 0 – RBAR is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis. 1 – RBAR is simulated using rigid option (i.e. simulated by rigid link or constraint equations as determined by program) 2 – RBAR is simulated using flexible option (i.e. simulated by spring or beam elements as determined by program) 3 – RBAR is simulated by spring elements See Section 2.7 of Advanced Nonlinear Theory and Modeling Guide for details on how RBAR elements are handled.
EQRBE2	Indicates how RBE2 elements are handled. (Integer; Default = 0) 0 – RBE2 is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis. 1 – RBE2 is simulated using rigid option (i.e. simulated by rigid links or constraint equations as determined by program) 2 – RBE2 is simulated using flexible option (i.e. simulated by spring or beam elements as determined by program) 3 – RBE2 is simulated by spring elements Note: When a flexible option is selected (EQRBE2=0, 2, or 3), the spring or beam elements created by the software have zero mass. See Section 2.7 of Advanced Nonlinear Theory and Modeling Guide for details on how RBE2 elements are handled.

Translation Options

Name	Description
SPRINGK	Stiffness of spring elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0; Unit = Force/Length) If SPRINGK = 0.0, program automatically sets SPRINGK according to the following calculations. SPRINGK = EMAX * LMODEL where EMAX = maximum Young's Modulus of materials in the model and LMODEL = largest dimension of the model. If no material is specified in the model, EMAX is set to 1.0E12.
BEAME	Young's Modulus of material assigned to beam elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0; Unit = Force/Length^2) If BEAME = 0.0, BEAME is set to EMAX * 100.0 where EMAX = maximum Young's Modulus of materials in the model. If no material is specified in the model, EMAX is set to 1.0E12.
BEAMA	Circular cross section area of beam elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0;Unit = Length^2) If BEAMA = 0.0, program automatically sets BEAMA according to the following calculation: BEAMA = (LMODEL * .01)^2 where LMODEL = largest dimension of the model
RBLCRIT	Critical length for determining how RBAR and RBE2 elements are simulated when the rigid or flexible option is used to simulate RBAR (see EQRBAR) and RBE2 (see EQRBE2). (Real, Default = 0.0; Unit = Length) When RBLCRIT = 0.0, a new value will be calculated by the software: If EQRBAR (or EQRBE2) = 1, RBLCRIT = LMODEL * 1.0E-6 If EQRBAR (or EQRBE2) = 2, RBLCRIT = LMODEL * 1.0E-3. See Table 2.7-1 in the Advanced Nonlinear Theory and Modeling Guide for information on how RBLCRIT affects the representation of RBAR or RBE2 elements.

Translation Options

Name	Description
GENMPC	Indicates whether generalized constraints are used for multipoint constraints (MPCs). The use of generalized constraint allows dependent degrees-of-freedom in one MPC to be assigned as independent by another MPC. (Integer; Default = 0) 0 – regular constraints are used for MPCs 1 – generalized constraints are used for MPCs

REMARKS:

1. The NXSTRAT bulk entry is not required. When it is not present, the default values are used. At least one parameter should be defined when a NXSTRAT entry exists.

2. XMSCALE, XDTMIN1 and XDTMIN2 may be used together. XDTMIN1 and DTMIN2 are applied after XMSCALE is applied. If XDTMIN1 and XDTMIN2 are both used, XDTMIN1 should be greater than XDTMIN2. If XDTMIN2 = XDTMIN1 is specified, XDTMIN1 will be ignored.

3. WARNING: Specifying XMSCALE > 1.0, XDTMIN1 > 0.0 or XDTMIN2 > 0.0 may change the model significantly. Hence, extra care should be exercised in examining the results when any of these parameters are used.

4. The following table summarizes the supported solutions for each NXSTRAT parameter. The table cells which include X^1 indicate that TRANOPT=1 must be defined on the TMCPARA bulk entry. The table cells which include X^o indicate that TRANOPT=0 or 2 must be defined. For example, AUTO=3 is supported by SOL 601,159 only when TRANOPT=1.

	SOL 601,*x*				701		SOL 601,*x*				701
	,106	,129	,153	,159			,106	,129	,153	,159	
SOLVER	X	X	X	X		RCONSM	X	X	X	X	
AUTO=1	X	X	X	X		ENLSTH	X	X	X	X	X
AUTO=2-4	X		X^1	X^1		ATSSUBD	X	X	X	X	
NPOSIT	X	X	X	X		ATSMXDT	X	X	X	X	
MASSTYP	X	X	X	X	X	ATSMASS	X		X^1	X^1	

TINT	X	X	X	X	X	ATSNEXT	X	X	X	X	
ICMODE	X	X	X	X	X	ATSDFAC	X	X	X	X	
MSTAB	X		X			ATSLOWS	X		X[1]	X[1]	
MSFAC	X		X			ATSDAMP	X		X[1]	X[1]	
DTDELAY	X	X	X	X	X	LDCGRID	X				
SDOFANG	X	X	X	X	X	LDCDOF	X				
DRILLKF	X	X	X	X		LDCDISP	X				
UPFORM	X	X	X	X	X	TLANSTP	X		X[1]	X[1]	
ULFORM	X	X	X	X	X	TLAMXIT	X		X[1]	X[1]	
DISPOPT	X	X	X	X	X	TLAMXDF	X		X[1]	X[1]	
LOADOPT	X	X	X	X	X	TLASTBF	X		X[1]	X[1]	
MAXDISP	X	X	X	X		TLALSDF	X		X[1]	X[1]	
TINTEG		X	X[o]	X[o]		TLALSMF	X		X[1]	X[1]	
ALPHA		X	X[o]	X[o]		TLACTDF	X		X[1]	X[1]	
DELTA		X	X[o]	X[o]		IMPACT		X	X[o]	X[o]	
XSTEP					X	NSUPP	X	X	X	X	
XDTCAL					X	CSTYPE	X	X	X	X	X
XDTFAC					X	CTDISP	X	X	X	X	X
XMSCALE					X	RTALG	X	X	X	X	
XDTMIN1					X	FRICALG	X	X	X	X	
XDTMIN2					X	TNSLCF	X	X	X	X	
ITEMAX	X	X	X	X		CTDAMP	X	X	X	X	
EPSIA	X	X	X	X		CTDAMPN	X	X	X	X	
EPSIB	X	X	X	X		CTDAMPT	X	X	X	X	
EPSII	X	X	X	X		MODEX	X	X	X	X	X
LSEARCH	X	X	X	X		TSTART	X	X	X	X	X
LSLOWER	X	X	X	X		IRINT	X	X	X	X	X
LSUPPER	X	X	X	X		NSUBGRP	X	X	X	X	X
PLASALG	X	X	X	X		XTCURVE	X	X	X	X	X
MAXITE	X	X	X	X		CVSSVAL	X	X	X	X	X
CONVCRI	X	X	X	X		ELRESCS	X	X	X	X	X
ETOL	X	X	X	X		BOLTSTP	X	X	X	X	
RTOL	X	X	X	X		ELCV	X	X	X	X	
RNORM	X	X	X	X		EQRBAR	X	X	X	X	

RMNORM	X	X	X	X		EQRBE2	X	X	X	X	
RCTOL	X	X	X	X		SPRINGK	X	X	X	X	
DTOL	X	X	X	X		BEAME	X	X	X	X	
DNORM	X	X	X	X		BEAMA	X	X	X	X	
DMNORM	X	X	X	X		RBLCRIT	X	X	X	X	
STOL	X	X	X	X		GENMPC	X	X	X	X	

Chapter 16: Bulk Data Entries O—P

OMIT

Omitted Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

FORMAT:

1	2	3	4	5	6	7	8	9	10
OMIT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

OMIT	16	2	23	3516			1	4	

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)

REMARKS:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. Up to 24 degrees-of-freedom may be specified on a single entry.

3. In many cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.

4. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

OMIT1

Omitted Degrees-of-Freedom, Alternate Form 1

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

FORMAT:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	-etc.-					

EXAMPLE:

OMIT1	3	2	1	3	10	9	6	5	
	7	8							

ALTERNATE FORMAT AND EXAMPLE:

OMIT1	C	G1	“THRU”	G2					
OMIT1	0	17	THRU	109					

FIELDS:

Field	Contents
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, G1 < G2.)

REMARKS:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. If the alternate format is used, not all points in the range G1 through G2 have to be defined. Undefined points will collectively produce a warning message but will otherwise be ignored.

3. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

PAABSF

Frequency-Dependent Absorbers Element Property

Defines the properties of a frequency-dependent acoustic absorber element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAABSF	PID	TZREID	TZIMID	S	A	B	K	RHOC	

EXAMPLE:

PAABSF	44	38	47						

FIELDS:

Field	Contents
PID	Property identification number that matches the identification number of the corresponding CAABSF entry. (Integer > 0)
TZREID	Identification number of a TABLEDi entry that defines the resistance as a function of frequency. The real part of the impedance. See Remark 1 . (Integer > 0)
TZIMID	Identification number of a TABLEDi entry that defines the reactance as a function of frequency. The imaginary part of the impedance. See Remark 1 . (Integer > 0)
S	Impedance scale factor. (Real; Default = 1.0)
A	Area factor when 1 or 2 grid points are specified on the CAABSF entry. (Real > 0.0; Default = 1.0)
B	Equivalent structural damping coefficient. (Real ≥ 0.0; Default = 0.0)
K	Equivalent structural stiffness coefficient. (Real ≥ 0.0; Default = 0.0)

Field	Contents
RHOC	Constant used in data recovery for calculating an absorption coefficient. RHO is the media density, and C is the speed of sound in the media. (Real; Default = 1.0)

REMARKS:

1. At least one of the four fields TZREID, TZIMID, B, or K must be specified.

2. If only one grid point is specified on the CAABSF entry, then the impedance $Z(f)=Z_R + iZ_I$ is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area. $Z_R(f)$=TZREID(f)+B and $Z_I(f)$=TZIMID(f)−K/(2πf).

3. The resistance represents a damper quantity B. The reactance represents a quantity of the type (ωM−K/ω). The impedance is defined as $Z = p/\dot{u}$ where p is the pressure and $\dot{u}$ is the velocity.

4. The impedance scale factor S is used in computing element stiffness and damping terms as:

$$k = \frac{\mathrm{A}}{\mathrm{S}} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

$$b = \frac{\mathrm{A}}{\mathrm{S}} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int \text{(of shape functions)}$$

5. The output for the element is specified by the STRESS Case Control command and consists of the resistance, reactance, and absorption coefficient. The absorption coefficient is defined as:

$$a = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

PACABS

Acoustic Absorber Property

Defines the properties of the acoustic absorber element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PACABS	PID	SYNTH	TID1	TID2	TID3	TESTAR	CUTFR	B	
	K	M							

EXAMPLE:

PACABS	12		1	2	3	3.5	500.0		

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
SYNTH	Request the calculation of B, K, and M from the tables TIDi below. (Character = “YES” or “NO”; Default = “YES”)
TID1	Identification of the TABLEDi entry that defines the resistance. See Remark 2 . (Integer > 0 or blank)
TID2	Identification of the TABLEDi entry that defines the reactance. See Remark 2 . (Integer > 0 or blank)
TID3	Identification of the TABLEDi entry that defines the weighting function. See Remark 2 . (Integer > 0 or blank)
TESTAR	Area of the test specimen. (Real > 0.0; Default = 1 .0)
CUTFR	Cutoff frequency for tables referenced above. (Real > 0.0)

Field	Contents
B, K, M	Equivalent damping, stiffness and mass values per unit area. (Real ≥ 0.0)

REMARKS:

1. PACABS is referenced by a CHACAB entry only.
2. If SYNTH = “YES”, then TID1 and TID2 must be supplied (TID3 is optional) and the equivalent structural model will be derived from tables TIDi. If TID3 is blank, then the weighting function defaults to 1.0.
3. If SYNTH = “NO”, then the equivalent structural model will be derived from one of B, K, or M.
4. The continuation entry is optional.
5. All data defined in tables TIDi must be a function of frequency in cycles/unit time.

PACBAR

Acoustic Barrier Property

Defines the properties of the acoustic barrier element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PACBAR	PID	MBACK	MSEPTM	FRESON	KRESON				

EXAMPLE:

PACBAR	12	1.0	0.01	400.0					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MBACK	Mass per unit area of the backing material. (Real > 0.0)
MSEPTM	Mass per unit area of the septum material. (Real > 0.0)
FRESON	Resonant frequency of the sandwich construction in hertz. (Real > 0.0 or blank)
KRESON	Resonant stiffness of the sandwich construction. (Real > 0.0 or blank)

REMARKS:

1. PACBAR is referenced by a CHACBR entry only.
2. Either FRESON or KRESON must be specified, but not both.

PAERO1

Aerodynamic Panel Property

Defines associated bodies for the panels in the Doublet-Lattice method.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAERO1	PID	B1	B2	B3	B4	B5	B6		

EXAMPLE:

PAERO1	1	3							

FIELDS:

Field	Contents
PID	Property identification number referenced by a CAERO1 entry. (Integer > 0)
Bi	Identification number of CAERO2 entries for associated bodies. (Integer ≥ 0 or blank)

REMARKS:

1. The associated bodies must be in the same aerodynamic group, as specified in the IGID field on CAERO2 entry.
2. If there are no bodies, the entry is still required (with Bi fields blank).
3. The Bi numbers above must appear on a CAERO2 entry to define these bodies completely.

PAERO2

Aerodynamic Body Properties

Defines the cross-sectional properties of aerodynamic bodies.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAERO2	PID	ORIENT	WIDTH	AR	LRSB	LRIB	LTH1	LTH2	
	THI1	THN1	THI2	THN2	THI3	THN3			

EXAMPLE:

PAERO2	2	Z	6.0	1.0	22	91	100		
	1	3							

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
ORIENT	Orientation flag. Type of motion allowed for bodies. Refers to the aerodynamic coordinate system of ACSID. See AERO entry. (Character = “Z”, “Y”, or “ZY”)
WIDTH	Reference half-width of body and the width of the constant width interference tube. (Real > 0.0)
AR	Aspect ratio of the interference tube (height/width). (Real > 0.0)
LRSB	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the slender body elements. If blank, the value of WIDTH will be used. (Integer > 0 or blank)

Field	Contents
LRIB	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the interference elements. If blank, the value of WIDTH will be used. (Integer > 0 or blank)
LTH1, LTH2	Identification number of AEFACT entries for defining θ arrays for interference calculations. (Integer ≥ 0)
THIi, THNi	The first and last interference element of a body to use the θ_1 array; the others use the θ_2 array. (Integer ≥ 0)

REMARKS:

1. The half-widths (given on AEFACT entries referenced in fields 6 and 7) are specified at division points. The number of entries on an AEFACT entry used to specify half-widths must be one greater than the number of elements.
2. The half-width at the first point (i.e., the nose) on a slender body is usually 0.0; thus, it is recommended (but not required) that the LRSB data is supplied with a zero first value.
3. THIi and THNi are interference element numbers on a body. The first element is one for each body.
4. A body is represented by a slender body surrounded by an interference tube. The slender body creates the downwash due to the motion of the body, while the interference tube represents the effects upon panels and other bodies.

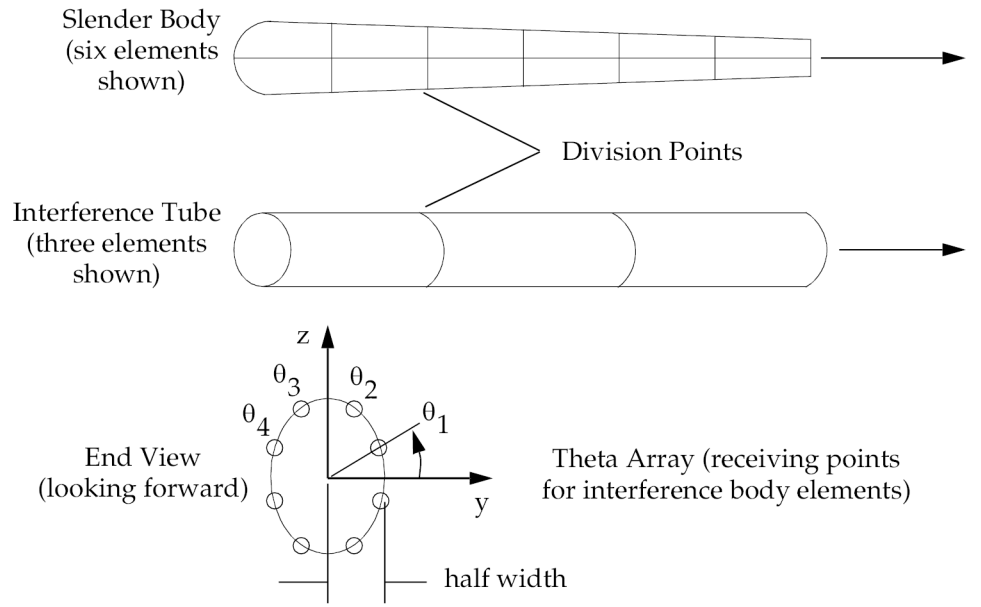


Figure 16-1. Idealization of Aerodynamic Body

5. The angles θ_1 and θ_2 are input in degrees using the aerodynamic element coordinate system as the reference coordinate system for defining the theta points.

6. Distribution of the theta points need not be uniform. A theta point must be placed a finite distance from any aerodynamic box edge; preferably the box edge would be equidistant from any two theta points. This aerodynamic coordinate system is defined on the AERO Bulk Data entry.

7. For half models, the theta arrays LTH1 and LTH2 should encompass a full 360 degree range.

PAERO3

Aerodynamic Panel Property

Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAERO3	PID	NBOX	NCTRL		X5	Y5	X6	Y6	
	X7	Y7	X8	Y8	X9	Y9	X10	Y10	
	X11	Y11	X12	Y12					

EXAMPLE:

PAERO3	2001	15	1		0.	65.			
	78.	65.	108.	65.	82.	97.5	112.	97.5	
	86.	130.	116.	130.					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
NBOX	Number of Mach boxes in the flow direction. (0 < Integer < 50)
NCTRL	Number of control surfaces. (Integer 0, 1, or 2)
X5 through Y12	Locations of points 5 through 12, which are in the aerodynamic coordinate system, to define the cranks and control surface geometry. (Real)

REMARKS:

1. For an illustration of the geometry, see the CAERO3 entry description.
2. If Y5 ≤ 0.0, there is no leading edge crank. Also, if Y6 ≤ 0.0, there is no trailing edge crank.
3. If NCTRL = 0, no continuations are required. If NCTRL = 1 or 2, then NCTRL continuations are required.
4. Y7 ≥ Y8, Y9 ≥ Y10, and Y11 ≥ Y12.
5. The number of Mach boxes in the spanwise direction (NSB) may be found from the following formula:

$$NSB = \mathrm{INT}\left[\frac{\beta \cdot y_{max}}{\left(\frac{x_{max}}{\mathrm{NBOX} + 0.5}\right)} + 0.5\right]$$

where

$\beta = \sqrt{M^2 - 1}$

x_{max} = maximum chordwise direction

y_{max} = maximum spanwise direction

NBOX = initial number of boxes specified in field 3

The number of Mach boxes in the streamwise direction may then be computed from:

$$\mathrm{NBOX} = \mathrm{INT}\left[\frac{x_{max}}{\left(\frac{\beta \cdot y_{max}}{NSB - 0.5}\right)} + 0.999\right]$$

The number of chordwise boxes specified by the user (NBOX ≥ 50) will be replaced by a floating point number (usually slightly higher than NBOX). The method contracts the mesh equally in both dimensions until a box edge lies on the surface tip. This mesh size is then used to compute the number of chordwise boxes.

Note

A minimum of seven Mach boxes in the flow direction (NBOX) is recommended.

PAERO4

Aerodynamic Strip Properties

Defines properties of each strip element for Strip theory.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAERO4	PID	CLA	LCLA	CIRC	LCIRC	DOC1	CAOC1	GAPOC1	
	DOC2	CAOC2	GAPOC2	DOC3	CAOC3	GAPOC3	-etc.-		

EXAMPLE:

PAERO4	6001	1	501	0	0	0.0	0.0	0.0	
	0.50	0.25	0.02	0.53	0.24	0.0			

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
CLA	Select Prandtl-Glauert correction. (Integer = -1, 0, 1; Default = 0)
	-1 Compressibility correction made to lift curve slope data for a reference Mach number.
	0 No correction and no list needed. (Default)
	+1 No correction and lift curve slope provided by a list as a function of strip location and Mach number.
LCLA	Identification number of the AEFACT entry that lists the lift curve slope on all strips for each Mach number on the MKAEROi entry. See Remark 2 below. (Integer = 0 if CLA = 0, > 0 if CLA ≠ 0)

Field	Contents
CIRC	Select Theodorsen's function $C(k)$ or the number of exponential coefficients used to approximate $C(k)$. (Integer = 0, 1, 2, 3; Default = 0. Must be zero if CLA ≠ 0.) 0 Theodorsen function. 1, 2, 3 Approximate function with b_0, b_1, β_1, ..., b_n, β_n n = 1, 2, 3.
LCIRC	Identification number of the AEFACT entry that lists the b, β values for each Mach number. See Remark 3, 4, and 5 below; variable b's and β's for each mi on the MKAEROi entry. (Integer = 0 if CIRC = 0, > 0 if CIRC ≠ 0)
DOCi	d/c = distance of the control surface hinge aft of the quarter-chord divided by the strip chord (Real ≥ 0.0)
CAOCi	c_a/c = control surface chord divided by the strip chord. (Real ≥ 0.0)
GAPOCi	g/c = control surface gap divided by the strip chord. (Real ≥ 0.0)

REMARKS:

1. PAERO4 is required for Strip theory with three fields (DOCi, CAOCi, GAPOCi) specified per strip.
2. If CLA = -1, lift curve slope data at one Mach number are needed on the AEFACT entry.
3. If CAOCi = 0.0, there is no control surface.
4. If GAPOCi = 0.0, there is no slot flow.
5. If GAPOCi < 0.01, then 0.01 is used.
6. Embedded blank fields are not permitted.
7. The following table lists the lift curve slope or lag function selection and the AEFACT entry formats used for Strip theory:

Table 16-1. Strip Theory Function Selections and AERACT Entry Formats

Theodorsen Function	Data Type Input	Parameter Combinations				Number of Words	Entry Format Index
		CLA	LCLA	CIRC	LCIRC		
Exact	Lift Curve Slope $c_{l_{\alpha i}} = 2\pi$	0	0	0	0	No AEFACT entry required	
	$c_{l_{\alpha i}}$ Input, Uses Prandtl-Glauert Correction	-1	ID	0	0	(NSTRIP+1)	a.
	$c_{l_{\alpha i}}$ Input, for All m's on MKAERO Entry	1	ID	0	0	(NSTRIP+1)* NMACH	b.
Approximate Coefficients	b_{0i}, b_{1i}, β_{1i}, etc.	0	0	1	ID	4*NMACH	c.
		0	0	2	ID	6*NMACH	d.
		0	0	3	ID	8*NMACH	e.

Entry Format:

a.

$$\text{AEFACT, ID, } m_1, c_{l\alpha_1}, c_{l\alpha_2}, \ldots, c_{l\alpha_{\text{NSTRIP}}}$$

b.

$$\text{AEFACT, ID, } m_1, c_{l\alpha_{11}}, c_{l\alpha_{21}}, \ldots, c_{l\alpha_{\text{NSTRIP1}}},$$

$m_2, c_{l\alpha_{11}}, c_{l\alpha_{12}}, c_{l\alpha_{21}} c_{l\alpha_{22}}, \ldots, c_{l\alpha_{\text{NSTRIP1}}}, c_{l\alpha_{\text{NSTRIP2}}}$, for all m on MKAEROi data entry

c. AEFACT, ID, m_1, b_{01}, b_{11}, β_{11}, m_2, b_{02}, b_{12}, P_{12}, m_3, etc.

d. AEFACT, ID, m_1, b_{01}, b_{11}, β_{11}, b_{21}, β_{21}, m_2, etc.

e. AEFACT, ID, m_1, b_{01}, b_{11}, β_{11}, b_{21}, β_{21}, b_{31}, β_{31}, m_2, etc.

8. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *NX Nastran Aeroelastic Analysis User's Guide*.

PAERO5

Aerodynamic Panel Property

Defines properties of each strip element for Piston theory.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PAERO5	PID	NALPHA	LALPHA	NXIS	LXIS	NTAUS	LTAUS		
	CAOC1	CAOC2	CAOC3	CAOC4	CAOC5				

EXAMPLE:

PAERO5	7001	1	702	1	701	1	700		
	0.0	0.0	5.25	3.99375	0.0				

FIELDS:

Field	Contents
PID	Property identification number. (Unique Integer > 0)
NALPHA	Number of angle of attack (α) values to be input for each Mach number (mi) on the MKAERO1 or MKAERO2 entry. (Integer > 0)

NALPHA	Meaning
1	α is the same value for all strips; enter one value, in units of degrees, on the AEFACT entry for each Mach number.
Number of Strips	α is different for each strip; enter α's, in units of degrees, in the following order: m_1, α_1, α_2, ..., m_2, α_1, α_2, ..., etc.

LALPHA	ID number of the AEFACT entry that lists the α's for the strips at each Mach number in the MKAERO1 or MKAERO2 entry. (Integer > 0)

NXIS Number of dimensionless chord coordinates ξ to be input. (Integer ≥ 0, Default = 0)

NXIS	Meaning
0	No ξ's are required. (Default)
1	ξ's are the same for all strips; enter values for one strip on the AEFACT entry (ξ_h if NTHICK > 0, or ξ_m and ξ_h if NTHICK = 0)
Number of Strips	ξ's have to be input for each strip (ξ_{h1}, ξ_{h2}, ..., ξ_{hNSPAN}, if NTHICK> 0, or ξ_{m1}, ξ_{h1}, ξ_{m2}, ξ_{h2}, ..., ξ_{mNSPAN}, ξ_{hNSPAN}, β_{hNSPAN} if NTHICK = 0)

LXIS Identification number of AEFACT entry that lists the ξ values for the strip in order indicated by values of NXIS and NTHICK. (Integer = 0 if c_a = 0 and NTHICK > 0 or LXIS > 0 if c_a = 0 and/or NTHICK = 0)

NTAUS Parameter used to select the number of thickness ratio (τ) values to be input. (Integer ≥ 0, Default = 0)

NTAUS	Meaning
0	No τ's are required. (Default)
1	τ's are the same for all strips; enter (τ_1, τ_{h1}, τ_{t1}) values for one strip on AEFACT entry.
Number of Strips	τ's must to be input for each strip on an AEFACT entry in the following order: (τ_1, τ_{h1}, τ_{t1}, τ_2, τ_{h2}, τ_{t2}, ..., τ_{NSPAN}, τ_{hNSPAN}, τ_{tNSPAN})

LTAUS Identification number of AEFACT entry that lists the τ values for the strips. (Integer = 0 or blank if NTAUS = 0, LTAUS > 0 if NTAUS > 0)

CAOCi c_a/c = control surface chord divided by the strip chord. (Real ≥ 0.0)

REMARKS:

1. The continuation entry is required for Piston theory with one entry (CAOCi) per strip.
2. Embedded blank fields are not allowed on the continuation entry.
3. If CAOCi = 0.0, there is no control surface.
4. Table 16-2 lists the thickness data input and AEFACT entry format used for Piston theory.

Table 16-2. Thickness Data Input and AEFACT Entry Format for Piston Theory

Type of Input	Parameter Combinations						Number of Words	Entry Format Index
	CAOC	NGHICK	NXIS	LXIS	NTAUS	LTAUS		
No control surfaces, Integrals input are same for all strips	0.0	ID[a]	0	0	0	0	6	a.
With control surfaces, Integrals input, same hinge on all strips	≠ 0.0	ID[b]	1	ID[c]	0	0	121	b.c.
With control surfaces, Integrals input, variable hinge	≠ 0.0	ID[b]	NSTRIP	ID[d]	0	0	12NSTRIP	b.d.
No control surfaces, thickness inputs are same for all strips	0.0	0	1	ID[f]	1	ID[e]	32	e.f.
With control surfaces, thickness inputs are same for all strips	≠ 0.0	0	1	ID[f]	1	ID[e]	32	e.f.
With control surfaces, thickness inputs vary for strips	≠ 0.0	0	NSTRIP	ID[h]	NSTRIP	ID[g]	3*NSTRIP 2*NSTRIP	g.h.

Entry Format

a. AEFACT, ID, I_1, I_2, I_3, I_4, I_5, I_6

b. AEFACT, ID, I_1, ..., I_6, J_1, ..., $J_6 I_1$, I_2, I_3, I_4, I_5, I_6

c. AEFACT, ID, ξ_h

d. AEFACT, ID, ξ_{h1}, ξ_{h2}, ξ_{h3}, ..., $\xi_{h\mathrm{NSTRIP}}$

e. AEFACT, ID, τ_m, τ_h, τ_t

f. AEFACT, ID, ξ_m, ξ_h

g. AEFACT, ID, τ_{m1}, τ_{h1}, τ_{t1}, τ_{m2}, τ_{h2}, τ_{t2}, ..., $\tau_{m\mathrm{NSTRIP}}$, $\tau_{h\mathrm{NSTRIP}}$, $\tau_{t\mathrm{NSTRIP}}$

τ_{m1}, τ_{h1}, τ_{t1}, τ_{m2}, τ_{h2}, τ_{t2}, ..., $\tau_{m\mathrm{NSTRIP}}$, $\tau_{h\mathrm{NSTRIP}}$, $\tau_{t\mathrm{NSTRIP}}$

h. AEFACT, ID, ξ_{m1}, ξ_{h1}, ξ_{m2}, ξ_{h2}, ..., $\xi_{m\mathrm{NSTRIP}}$, $\xi_{h\mathrm{NSTRIP}}$

5. The following table lists the angle-of-attack distribution and AEFACT entry formats used for Piston theory.

Type of Distribution	Parameter Combinations		Number of Words	Entry Format Index
	NALPHA	LALPHA		
Equal angle of attack on all strips	1	ID	2*NMACH	a.
Unequal angle of attack	NSTRIP	ID	(1 + NSTRIP) * NMACH	b.

Entry Format

a. AEFACT, ID, m_1, α_1, m_2, α_2, ...,

b. AEFACT, ID, m_1, α_{11}, α_{21}, α_{31}, ..., $\alpha_{\mathrm{NSTRIP}1}$, m_2, α_{12}, α_{22}, ..., $\alpha_{\mathrm{NSTRIP}2}$, m_2, etc., for all m on MKAEROi entry.

c. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *NX Nastran Aeroelastic Analysis User's Guide*.

PANEL

Panel Definition for Coupled Fluid-Structural Analysis

Selects sets of structural grid points, elements, or physical properties that define one or more panels.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PANEL	NAME1	SID1	NAME2	SID2	NAME3	SID3	NAME4	SID4	

EXAMPLE:

PANEL	BKDOOR	103							

FIELDS:

Field	Contents
NAMEi	Panel label. (Character. See Bulk Data Syntax Rules.)
SIDi	Identification number of a SET1 or SET3 bulk entry that lists the structural grid points, elements, or physical properties of the panel. (Integer > 0)

REMARKS:

1. Panels are groups of structural grid points.
   - If a set of grid points is referenced, the set must include only structural grid points. The panel will consist of all the grid points in the referenced set. SET1 and SET3 bulk entries are used to define sets of grid points.
   - If a set of elements is referenced, the set must include only structural elements. The panel will consist of all the grid points that are connection

points for these elements. SET3 bulk entries are used to define sets of elements.

- If a set of physical property identifiers is referenced, the physical properties must be referenced by structural elements. The panel will consist of all grid points that are connection points for the structural elements referencing the physical properties included in the set. SET3 bulk entries are used to define sets of physical property identifiers.

2. If the referenced SET1 or SET3 bulk entries include structural grid points, the sets must include at least four grid points for quadrilateral faces and three grid points for triangular faces.

3. It is recommended that all of the connection points for a given element belong to the same panel.

4. NAMEi is used only for labeling the output of the panel modal participation factors (refer to the MODCON and PANCON case control commands). See Defining Panels in the *NX Nastran Acoustics User's Guide*.

5. When used with the ERP case control command, a PANEL must be defined with the SET3 bulk entry.

PARAM

Parameter

Specifies values for parameters used in solution sequences or user-written DMAP programs.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1	V2						

EXAMPLE:

PARAM	IRES	1							

FIELDS:

Field	Contents
N	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single-precision	Real	Blank
Character	Character	Blank
Real, double-precision	Double-precision real	Blank
Complex, single-precision	Real or blank	Real or blank
Complex, double-precision	Double-precision real	Double-precision real

REMARKS:

1. See "Parameters and Parameter Applicability Tables" for a list of parameters used in solution sequences that may be set by the user on PARAM entries.

2. If the large field entry format is used, the second physical entry must be present, even though fields 6 through 9 are blank.

PBAR

Simple Beam Property

Defines the properties of a simple beam element (CBAR entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

EXAMPLE:

PBAR	39	6	2.9		5.97				
			2.0	4.0					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2 and 3. (Integer > 0)
A	Area of bar cross section. (Real; Default = 0.0)
I1, I2, I12	Area moments of inertia. See Figure 16-2. (Real; I1 ≥ 0.0, I2 ≥ 0.0, I1*I2 ≥ 112^2; Default = 0.0)
J	Torsional constant. See Figure 16-2. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)
Ci, Di, Ei, Fi	Stress recovery locations. See Remark 6. (Real; Default = 0.0)
K1, K2	Area factor for shear. See Remark 5 . (Real; Default = 0.0)

REMARKS:

1. Both continuation entries may be omitted.

2. For structural problems, MID must reference a MAT1 material entry.

3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.

4. See the CBAR entry description for a discussion of bar element geometry.

5. The transverse shear stiffnesses per unit length in planes 1 and 2 are K1*A*G and K2*A*G, respectively, where G is the shear modulus. When K1/K2 are set to Blank or 0.0 (Default), the transverse shear flexibilities are set equal to zero (which is equivalent to K1/K2 being infinite). K1 and K2 are ignored if I12 ≠ 0. K1 and K2 must be blank if A = 0.0.

6. C1 and C2, etc., are the y and z coordinates in the bar element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the bar.

7. For response spectra analysis on stress recovery coefficients, the CBEAM element entry should be used because bar element results will be inaccurate.

8. Figure 16-2 describes the PBAR element coordinate system.

   Note:

I1	=	I_{zzelem}
I2	=	I_{yyelem}
I12	=	I_{zyelem}
J	=	I_{xxelem}

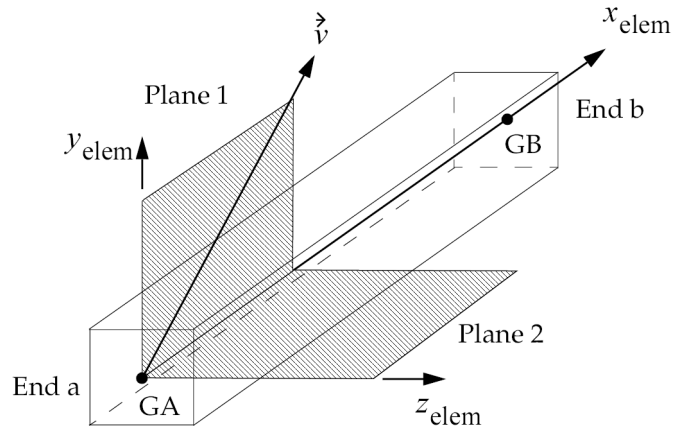


Figure 16-2. PBAR Element Coordinate System

9. By definition, the shear center and neutral axes coincide.

REMARKS RELATED TO SOLS 601 AND 701:

1. NSM, C1, C2, D1, D2, E1, E2, F1, F2, and I12 are ignored.
2. PBAR cannot be used for elasto-plastic beam elements. For elasto-plastic beam elements, PBARL with circular (TYPE = ”ROD” or “TUBE”) or rectangular (TYPE=”BAR”) cross sections must be used.
3. For structural problems, MID must reference a MAT1 or MATPLCY material entry.
4. For heat transfer problems, MID must reference a MAT4 material entry.

PBARL

Simple Beam Cross-Section Property

Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID	GROUP	TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

EXAMPLE:

PBARL	39	6		I					
	14.0	6.0	0.5	0.5	0.5				

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. See Remarks 6 and 9. (Character; Default = “MSCBML0”)
TYPE	Cross-section type. See Remarks 6 and 9 and Figure 3. (Character: “ROD”, “TUBE”, “I”, “CHAN”, “T”, “BOX”, “BAR”, “CROSS”, “H”, “T1”, “I1”, “CHAN1”, “Z”, “CHAN2”, “T2”, “BOX1”, “HEXA”, “HAT”, “HAT1” for GROUP = “MSCBML0”)
DIMi	Cross-sectional dimensions. (Real > 0.0 for GROUP = “MSCBMLO”)

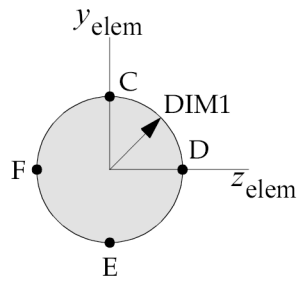
Field	Contents
NSM	Nonstructural mass per unit length. NSM is specified after the last DIMi. (Real ≥ 0; Default = 0.0)

REMARKS:

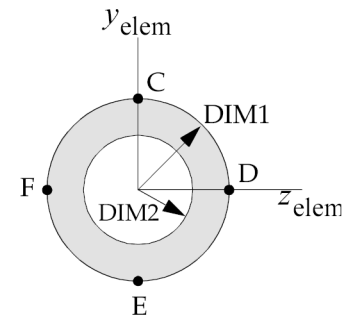
1. For structural problems, PBARL entries must reference a MAT1 material entry.
2. PID must be unique with respect to all other PBAR and PBARL property identification numbers.
3. See CBAR entry for a discussion of bar element geometry.
4. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
5. For response spectra analysis on stress recovery coefficients, the CBEAM element should be used because results for the CBAR element will not be accurate.
6. The GROUP is associated with an FMS CONNECT statement that specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require one or more subroutines to convert DIMi information to geometric property information contained on a PBAR entry and optimization information.
7. An equivalent PBAR entry is created from the PBARL entry. Any sorted echo request will also cause printout and/or punch of the derived PBAR.
8. The cross sectional properties on the equivalent PBAR entry are calculated using hard-coded formulas for each section type. To use the mesh-based Pilkey method to calculate the cross sectional properties, specify PARAM,PBRPROP,YES.
9. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are computed using the TYPE and DIMi as shown in Figure 16-3. The origin of element coordinate system is centered at the shear center of the cross-section oriented as shown. The PBARL does not account for offsets between the neutral axis and the shear center. Therefore, the CHAN, CHAN1 and CHAN2 cross-sections may produce incorrect results. The PBEAML is recommended.
10. C, D, E, and F in Figure 16-3 are the locations of stress recovery on a cross-section.

REMARKS RELATED TO SOLS 601 AND 701:

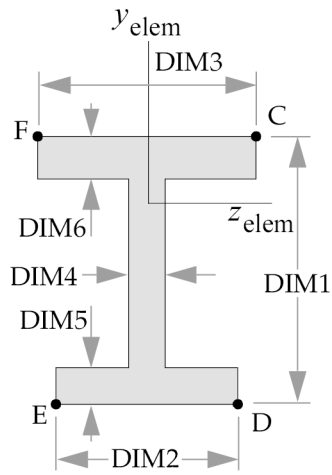
1. GROUP and NSM are ignored.
2. TYPE= “Z”, “BOX1”, “CROSS”, “HEXA”, “HAT”, and “HAT1” are not supported.
3. No shear stiffness correction is used.
4. For structural problems, MID must reference a MAT1 or MATPLCY material entry.
5. For heat transfer problems, MID must reference a MAT4 material entry.
6. For elasto-plastic beam elements, circular (TYPE = “ROD” or “TUBE”) or rectangular (TYPE= “BAR”) cross sections must be used.



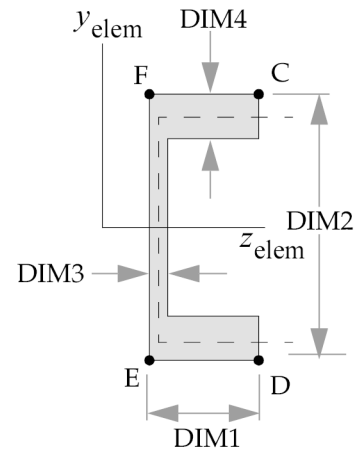
TYPE="ROD"



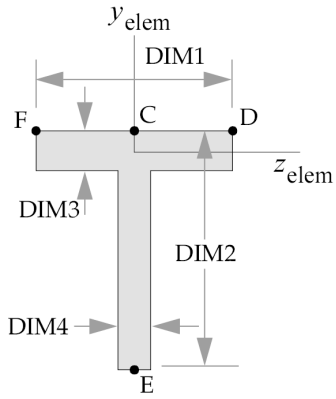
TYPE="TUBE"



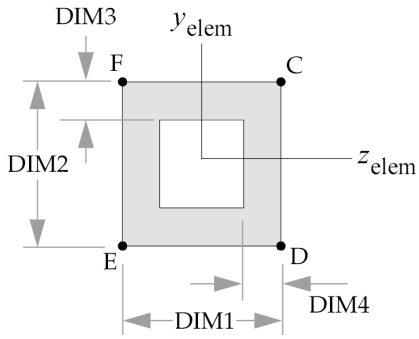
TYPE="I"



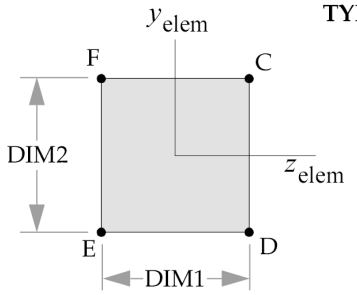
TYPE="CHAN"



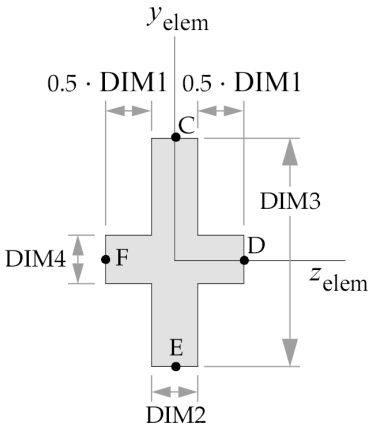
TYPE="T"



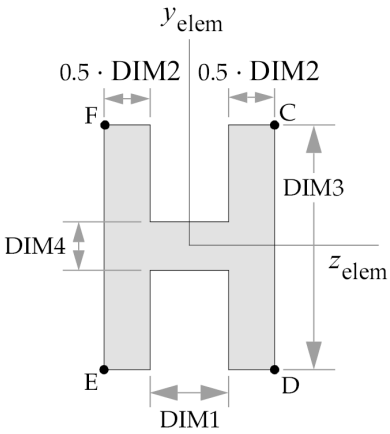
TYPE="BOX"



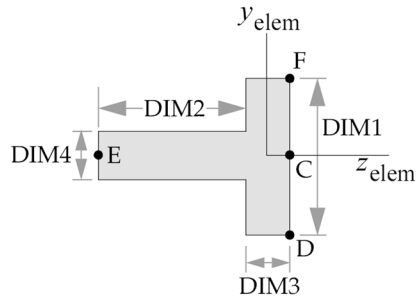
TYPE="BAR"



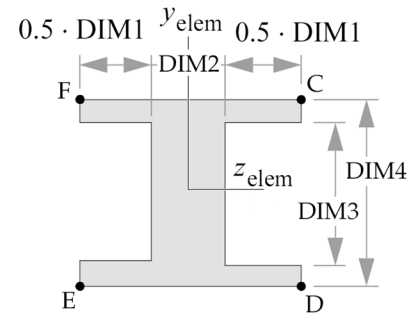
TYPE="CROSS"



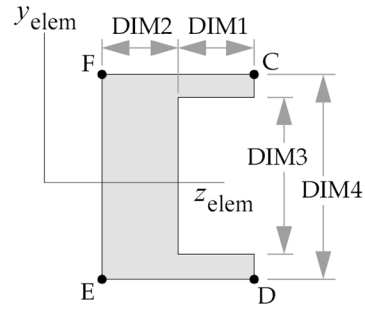
TYPE="H"



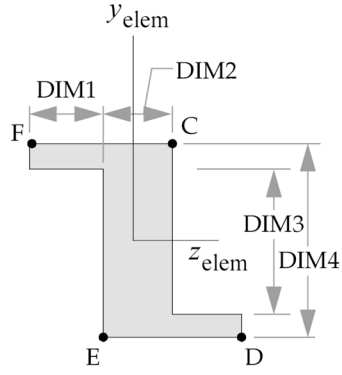
TYPE="T1"



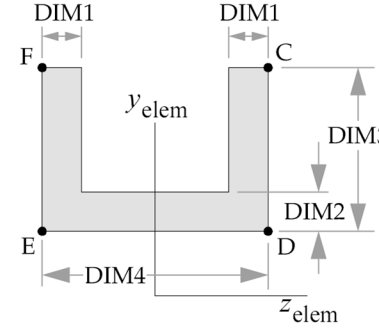
TYPE="I1"



TYPE="CHAN1"



TYPE="Z"



TYPE="CHAN2"

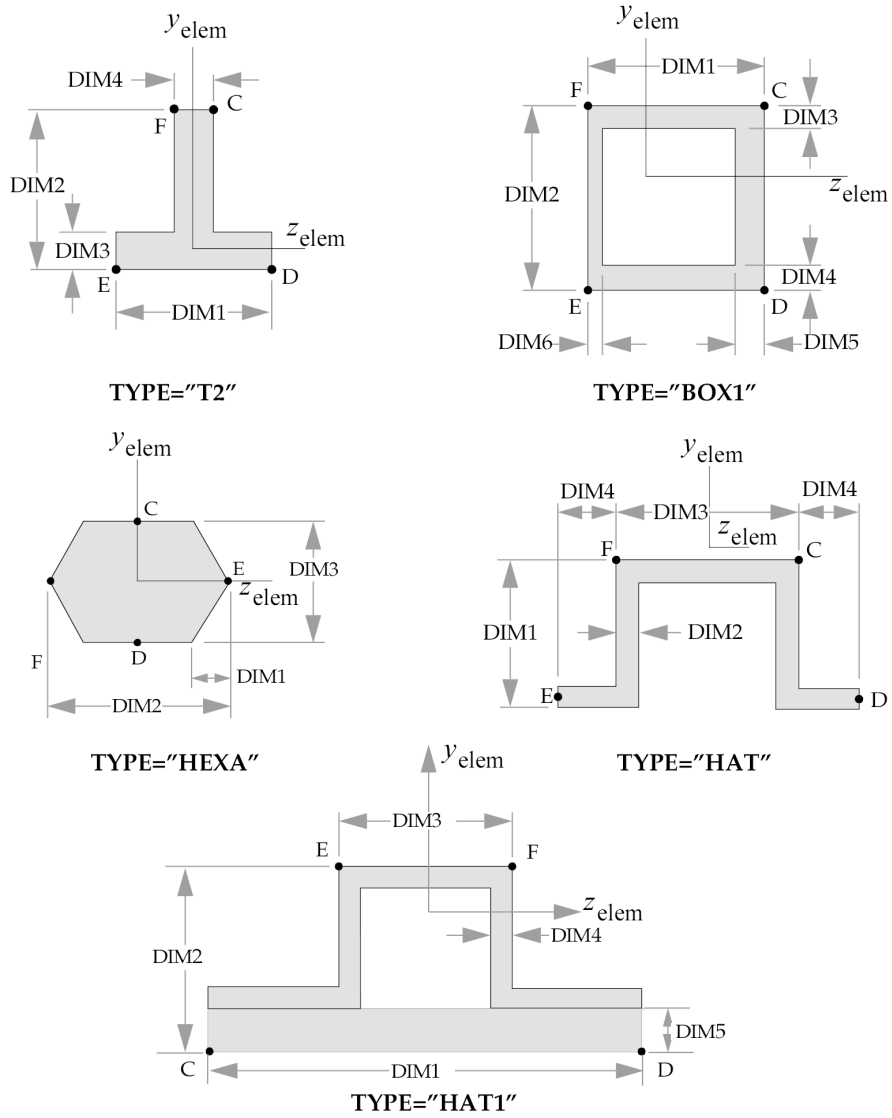


Figure 16-3. Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

PBCOMP

Beam Property (Alternate Form of PBEAM)

Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. This entry is also used to specify lumped areas of the beam cross section for nonlinear analysis and/or composite analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBCOMP	PID	MID	A	I1	I2	I12	J	NSM	
	K1	K2	M1	M2	N1	N2	SYMOPT		
	Y1	Z1	C1	MID1					
	Y2	Z2	C2	MID2					
	-etc.-								

EXAMPLE:

PBCOMP	39	6	2.9						
							1		
	-0.5	1.2	0.1	18					
	0.2	0.9	0.15						

FIELDS:

Field	Contents
PID	Property identification number. See Remark 1 . (Integer > 0)
MID	Material identification number. See Remarks 2 and 5. (Integer > 0)
A	Area of beam cross section. (Real > 0.0)
I1	Area moment of inertia in plane 1 about the neutral axis. See Remark 6 . (Real > 0.0)

Field	Contents
I2	Area moment of inertia in plane 2 about the neutral axis. See Remark 6 . (Real > 0.0)
I12	Area product of inertia. See Remark 6 . (Real; Default = 0.0, but $I1 \cdot I2 - (I12)^2 > 0.0$)
J	Torsional stiffness parameter. See Remark 6 . (Real > 0.0; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real > 0.0; Default = 0.0)
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. See Remark 4 . (Real > 0.0; Default = 1.0)
M1, M2	The (y,z) coordinates of center of gravity of nonstructural mass. See the figure in the CBEAM entry description. (Real; Default = 0.0)
N1, N2	The (y,z) coordinates of neutral axis. See the figure in the CBEAM entry description. (Real; Default = 0.0)
SYMOPT	Symmetry option to input lumped areas for the beam cross section. See Figure 16-5 and Remark 7 . (0 ≤ Integer ≤ 5; Default = 0)
Yi, Zi	The (y,z) coordinates of the lumped areas in the element coordinate system. See Remark 1 . (Real)
Ci	Fraction of the total area for the i-th lumped area. (Real > 0.0; Default = 0.0)
MIDi	Material identification number for the i-th integration point. See Remark 5 . (Integer > 0)

REMARKS:

1. The PID number must be unique with respect to other PBCOMP entries as well as PBEAM entries. The second continuation entry may be repeated 18 more times. If SECTION = 5 a maximum of 21 continuation entries is allowed; i.e., a maximum of 20 lumped areas may be input. If SECTION = 1 through 4, the total number of areas input plus the total number generated by symmetry must not exceed 20. If these are not specified, the program defaults, as usual, to the elliptically distributed eight nonlinear rods. See Figure 16-4.

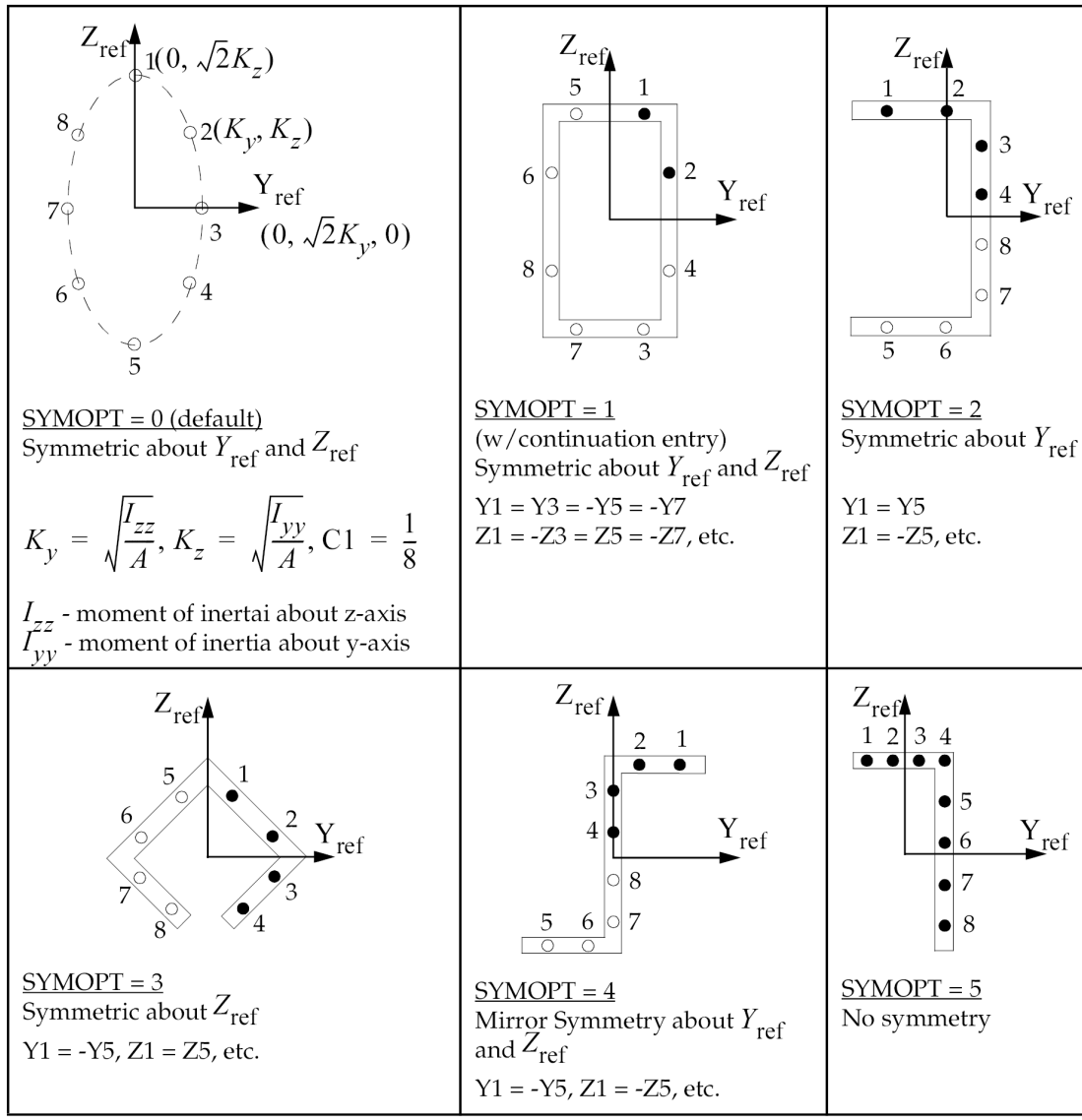


Figure 16-4. PBCOMP Entry SYMOPT Type Examples with 8 Lumped Areas

Figure Example Notes:

Integration points (lumped area) are numbered 1 through 8.

User-specified points are denoted by ● and the program default point is denoted by ○.

2. For structural problems, MID and MIDi must reference a MAT1 material entry. For material nonlinear analysis, the material should be perfectly plastic since the plastic hinge formulation is not valid for strain hardening. For heat transfer problems, MID and MIDi must reference a MAT4 or MAT5 material entry.

3. For the case where the user specifies I1, I2 and I12 on the parent entry, the stress-output location may also be specified on continuation entries. The (y,z) coordinates specified on these entries will serve as stress output locations with the corresponding Ci's set to 0. Stress output is provided at the first four lumped area locations only. If one of the symmetry options is used and fewer than four lumped areas are input explicitly, the sequence of output locations in the imaged quadrants is shown in Figure 16-4. For one specific example in the model shown in Remark 7 (Figure 16-5), output can be obtained at points 1 and 2 and in the image points 3 and 4.

4. Blank fields for K1 and K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear stiffness becomes rigid and the transverse shear flexibilities are set to 0.0.

5. The values E_0 and G_0 are computed based on the value of MID on the parent entry. MID is will follow the same symmetry rules as Ci depending on the value of SECTION. If the MIDi field on a continuation entry is blank, the value will be that of MID on the parent entry. MIDi values may be input on continuations without the corresponding Yi, Zi, and Ci values to allow different stress-strain laws.

6. If the lumped cross-sectional areas are specified, fields I1, I2, and I12 will be ignored. These and other modified values will be calculated based on the input data (Yi, Zi, Ci, MIDi) as follows:

$$y_{NA} = \frac{\sum_{i=1}^{n} \mathrm{Yi\ Ci\ Ei}}{\sum_{i=1}^{n} \mathrm{Ci\ Ei}}$$

$$z_{NA} = \frac{\sum_{i=1}^{n} \mathrm{Zi\ Ci\ Ei}}{\sum_{i=1}^{n} \mathrm{Ci\ Ei}}$$

$$\bar{A} = A \sum_{i=1}^{n} \frac{\mathrm{Ci\ Ei}}{\mathrm{E}_0}$$

$$\bar{I}_1 = A \sum_{i=1}^{n} \frac{\mathrm{Ci\ Ei}\ (\mathrm{Yi} - y_{\mathrm{NA}})^2}{\mathrm{E}_o}$$

$$\bar{I}_2 = A \sum_{i=1}^{n} \frac{\mathrm{Ci\ Ei}\ (\mathrm{Yi} - z_{\mathrm{NA}})^2}{\mathrm{E}_o}$$

$$\overline{I_{12}} = A \sum_{i=1}^{n} \frac{\mathrm{Ci\ Ei}(\mathrm{Yi} - y_{NA})(\mathrm{Zi} - z_{NA})}{\mathrm{E}_o}$$

$$\bar{J} = \mathrm{J} \sum_{i=1}^{n} \frac{\mathrm{Ci\ Gi}}{\mathrm{G}_o}$$

where n is the number of lumped cross-sectional areas specified.

7. As can be seen from Figure 16-4, if the user chooses to leave the SECTION field blank, the program defaults to the elliptically distributed eight nonlinear rods, similar to the PBEAM entry. For this particular case it is illegal to supply Ci and MIDi values. For a doubly symmetric section (SECTION = 1), if the lumped areas are specified on either axis, the symmetry option will double the areas. For example, for the section shown in Figure 16-5, points 2 and 4 are coincident and

so are points 6 and 8. In such cases, it is recommended that users input the value of area as half of the actual value at point 2 to obtain the desired effect.

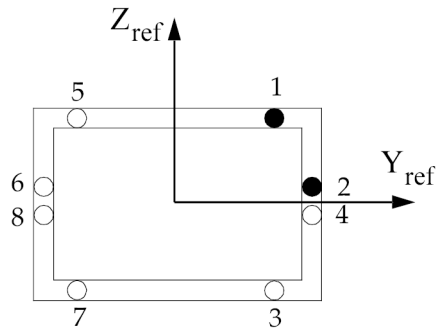


Figure 16-5. Doubly Symmetric PBCOMP Section

8. For SECTION = 5, at least one Yi and one Zi must be nonzero.

REMARKS RELATED TO SOLS 601 AND 701:

1. Only MID, A, I1, I2, J, K1, and K2 are used to define the properties of a simple beam. All other fields are ignored.

2. The neutral axis and shear center are assumed to be coincident with the element x-axis.

3. The behavior of a CBEAM element is the same as a CBAR element in SOLs 601 and 701.

4. For structural problems, MID must reference a MAT1 or MATPLCY material entry.

5. For heat transfer problems, MID must reference a MAT4 material entry.

PBEAM

Beam Property

Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark 5. and SO and X/XB must be specified.

	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

EXAMPLE: TAPERED BEAM WITH A=2.9 AT END A AND A=5.3 AT END B.

PBEAM	39	6	2.9	3.5	5.97				
			2.0	-4.0					
	YES	1.0	5.3	56.2	78.6				
			2.5	-5.0					
			1.1		2.1		0.21		
					0.5		0.0		

FIELDS:

Field	Contents	Default Values
PID	Property identification number. (Integer > 0)	Required
MID	Material identification number. See Remarks 1 and 2. (Integer > 0)	Required
A(A)	Area of the beam cross section at end A. (Real > 0.0)	Required
I1(A)	Area moment of inertia at end A for bending in plane 1 about the neutral axis. See Remarks 10 and 11. (Real > 0.0)	Required
I2(A)	Area moment of inertia at end A for bending in plane 2 about the neutral axis. See Remarks 10 and 11. (Real > 0.0)	Required
I12(A)	Cross product of inertia at end A. See Remarks 10 and 11. (Real ≥ 0.0)	0.0
J(A)	Torsional stiffness parameter at end A. See Remark 10. (Real ≥ 0.0 but > 0.0 if warping is present)	0.0
NSM(A)	Nonstructural mass per unit length at end A. (Real)	0.0
Ci(A), Di(A) Ei(A), Fi(A)	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery. (Real)	y = z = 0.0
SO	Stress output request option. See Remark 9. (Character) “YES” Stresses recovered at points Ci, Di, Ei,and Fi on the next continuation. “YESA” Stresses recovered at points with the same y and z location as end A. “NO” No stresses or forces are recovered.	Required*

Field	Contents	Default Values
X/XB	Distance from end A in the element coordinate system divided by the length of the element. See Figure 16-6 in Remark 10. (Real > 0.0)	Required* See Remark 5 .
A, I1, I2, I12, J, NSM	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x. (Real; J > 0.0 if warping is present.)	See Remark 1 .
Ci, Di, Ei, Fi	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see Figure 16-6 in Remark 10) for the cross section located at X/XB. The values are fiber locations for stress data recovery. (Real)	
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. See Remark 13. (Real)	1.0, 1.0
S1, S2	Shear relief coefficient due to taper for plane1 and plane 2. (Real)	0.0, 0.0
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at ends A and B. See Figure 16-6. (Real)	0.0, same as end A
CW(A), CW(B)	Warping coefficient for end A and end B. See Remark 12. (Real)	0.0, same as end A
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B. See Figure 16-6. (Real)	0.0 (no offset from shear center), same values as end A
N1(A), N2(A),N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B. See Figure 16-6. (Real)	0.0 (no offset from shear center), same values as end A

REMARKS:

1. For structural analysis, MID must reference a MAT1 material entry. For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic using only H = 0.0. If a MATS1 is referenced which is not elastic-perfectly plastic with H = 0.0, a message is produced stating the material will be treated as elastic. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.

2. For heat transfer analysis, MID must reference a MAT4 or MAT5 material entry.

3. If no stress data at end A is to be recovered and a continuation with the SO field is specified, then the first continuation entry, which contains the fields C1(A) through F2(A), may be omitted.

4. If SO is “YESA” or “NO”, the third continuation entry, which contains the fields C1 through F2, must be omitted. If SO is “YES”, the continuation for Ci, Di, Ei, and Fi must be the next entry.

5. The rules for the continuations entries are:

   - The second and third continuation entries, which contain fields SO through F2, may be repeated nine more times for intermediate X/XB values for linear beam elements. The order of these continuation pairs is independent of the X/XB value; however, one value of X/XB must be 1.0, corresponding to end B. The intermediate stress output requests will be ignored in the nonlinear solution sequences (SOLs 106 and 129).

   - The fourth and fifth continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.

   - if X/XB is not specified, then output for end A only will be generated.

6. If any fields 4 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and NSM are set to the values given for end A. For the continuations that have intermediate values of X/XB between 0.0 and 1.0 and use the default option (any of the fields 4 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing section properties.

7. If SO is “YES”, blank fields are defaulted to 0.0.

8. Blank fields for K1, K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear flexibilities are set to 0.0 and field G on the MAT1 entry selected by MID must be nonzero.

9. If end B forces are desired, put “YESA” in the SO field even when no end A stress points are input.

10. Figure 16-6 describes the PBEAM element coordinate system.

Note:

$I1 = I_{(zz)_{na}}$	$N1(A) = y_{na}$	$N1(B) = y_{nb}$
$I2 = I_{(yy)_{na}}$	$N2(A) = z_{na}$	$N2(B) = z_{nb}$
$I12 = I_{(zy)_{na}}$	$M1(A) = y_{ma}$	$M1(B) = y_{mb}$
$J = I_{(xx)_{na}}$	$M2(A) = z_{ma}$	$M2(B) = z_{mb}$

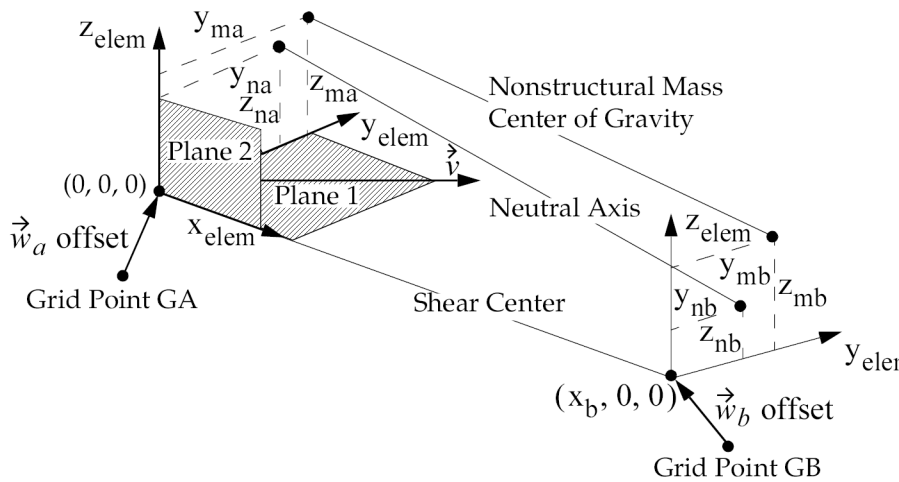


Figure 16-6. PBEAM Element Coordinate System

11. The product of I1 and I2 must be greater than I12 squared ($I1 \times I2 > I12^2$), otherwise a fatal message is issued.

12. The warping coefficient CW is represented in the following differential equation for the torsion of a beam about the axis of the shear centers:

$$G\frac{d}{dx}\left(J\frac{d\theta}{dx}\right) - E\frac{d^2}{dx^2}\left(CW\frac{d^2\theta}{dx^2}\right) = m$$

where

G = shear modulus

J = torsional stiffness

E = Young's modulus

θ = angle of rotation at any cross-section

m = applied torsional moment per unit length

Note: CW has units of $(\text{length})^6$.

13. The shear stiffness factors K_1 and K_2 adjust the effective transverse shear cross-section area according to the Timoshenko beam theory. Their default values of 1.0 approximate the effects of shear deformation. To neglect shear deformation (i.e., to obtain the Bernoulli-Euler beam theory), the values of K_1 and K_2 should be set to 0.0.

REMARKS RELATED TO SOLS 601 AND 701:

1. Tapered beams are not supported. Hence, only the properties at end A are used and a constant cross section is assumed, i.e., only A(A), I1(A), I2(A), J(A), K1, and K2 are used for the properties of the beam element.

2. I12(A), NSM(A), S1, S2, and all fields for intermediate stations and end B are ignored.

3. The neutral axis and shear center are assumed to be coincident with the element x-axis.

4. The behavior of a CBEAM element is the same as a CBAR element in SOLs 601 and 701.

5. For structural problems, MID must reference a MAT1 or MATPLCY material entry.

6. For heat transfer problems, MID must reference a MAT4 material entry.

7. PBEAM cannot be used for elasto-plastic beam elements. For elasto-plastic beam elements, PBEAML with circular (TYPE = "ROD" or "TUBE") or rectangular (TYPE="BAR") cross sections must be used.

PBEAML

Beam Cross-Section Property

Defines the properties of a beam element by cross-sectional dimensions.

FORMAT: (NOTE: N = NUMBER OF DIMENSIONS AND M = NUMBER OF INTERMEDIATE STATIONS)

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	GROUP	TYPE					
	DIM1(A)	DIM2(A)	-etc.-	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	DIM1(1)	
	DIM2(1)	-etc.-	DIMn(1)	NSM(1)	SO(2)	X(2)/XB	DIM1(2)	DIM2(2)	
	-etc.-	DIMn(2)	-etc.-	NSM(m)	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	
		NSM(B)							

EXAMPLE:

PBEAML	99	21		T					
	12.0	14.8	2.5	2.6		NO	0.4	6.0	
	7.0	1.2	2.6		YES	0.6	6.0	7.8	
	5.6	2.3		YES					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Character; Default = “MSCBML0”)

Field	Contents
TYPE	Cross-section shape. See Remark 4. (Character: “ROD”, “TUBE”, “L”, “I”, “CHAN”, “T”, “BOX”, “BAR”, “CROSS”, “H”, “T1”, “I1”, “CHAN1”, “Z”, “CHAN2”, “T2”, “BOX1”, “HEXA”, “HAT”, “HAT1” for GROUP = “MSCBML0”)
DIMi(A), DIMi(B)	Cross-section dimensions at end A and B. (Real > 0.0 for GROUP = “MSCBML0”)
NSM(A), NSM(B)	Nonstructural mass per unit length. (Real ≥ 0.0; Default = 0.0)
SO(j),SO(B)	Stress output request option for intermediate station j and end B. (Character; Default = “YES”) YES: Stresses recovered at all points on next continuation and shown in Figure 16-7 as C, D, E, and F. NO: No stresses or forces are recovered.
X(j)/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (Real > 0.0; Default = 1.0)
NSM(j)	Nonstructural mass per unit length at intermediate station j. (Real ≥ 0.0; Default = 0.0)
DIMi(j)	Cross-section dimensions at intermediate station j. (Real > 0.0 for GROUP = “MSCBML0”)

REMARKS:

1. For structural problems, PBEAML entries must reference a MAT1 material entry. For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic using only H = 0.0. If a MATS1 is referenced which is not elastic-perfectly plastic with H=0.0, a message is produced stating the material will be treated as elastic. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.

2. PID must be unique with respect to all other PBEAM and PBEAML property identification numbers.

3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.

4. See the PBEAM entry description for a discussion of beam-element geometry.

5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j)/XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.

6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is “MSCBML0”. Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry.

7. For GROUP = “MSCBML0”, the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in Figure 16-7. The element coordinate system is located at the shear center.

8. An equivalent PBEAM entry is created from the PBEAML entry. Any sorted echo request will also cause printout and/or punch of the derived PBEAM. The equivalent PBEAM does not include warping coefficients.

9. The cross sectional properties on the equivalent PBEAM entry are calculated using hard-coded formulas for each section type. To use the mesh-based Pilkey method to calculate the cross sectional properties, specify PARAM,PBRPROP,YES.

10. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words.

    None of these limits are exceeded with the NX Nastran beam library, but a user defined beam library could.

    There is a further limit that the NSM values input on the PBARL and PBEAML must be ≥ 0. No other property types have this limit.

11. C, D, E, and F in Figure 16-7 are the locations of stress recovery on a cross-section.

REMARKS RELATED TO SOLS 601 AND 701:

1. Tapered beams are not supported. Hence, only the dimensions at end A are used and a constant cross section is assumed, i.e., only DIM1(A), DIM2(A), to DIMn(A) are used for the properties of the beam element.

2. GROUP, NSM(A), and all fields for intermediate stations and end B are ignored.

3. TYPE = “L”, “Z”, “BOX1”, “CROSS”, “HEXA”, “HAT”, and “HAT1” are not supported.

4. No shear stiffness correction is used.

5. The neutral axis and shear center are assumed to be coincident with the element x-axis.

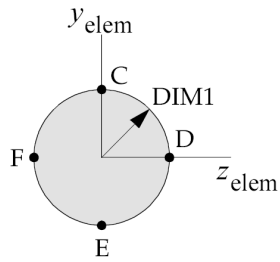
6. The behavior of a CBEAM element is the same as a CBAR element in SOLs 601 and 701.

7. For structural problems, MID must reference a MAT1 or MATPLCY material entry.

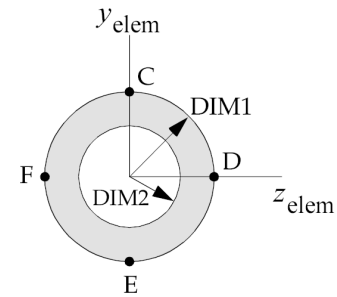
8. For heat transfer problems, MID must reference a MAT4 material entry.

9. For elasto-plastic beam elements, circular (TYPE = “ROD” or “TUBE”) or rectangular (TYPE= “BAR”) cross sections must be used.

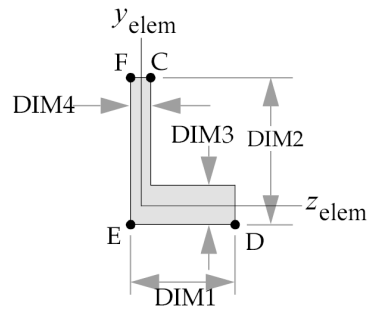
10. Offsets defined between the neutral axis and the shear center are ignored.



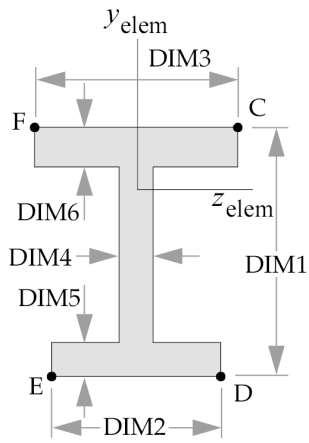
TYPE="ROD"



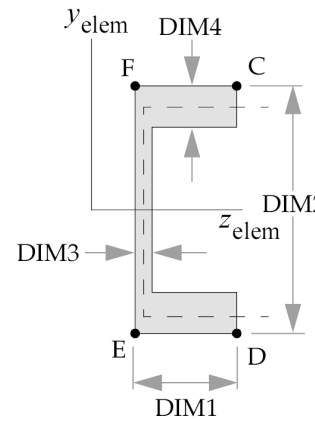
TYPE="TUBE"



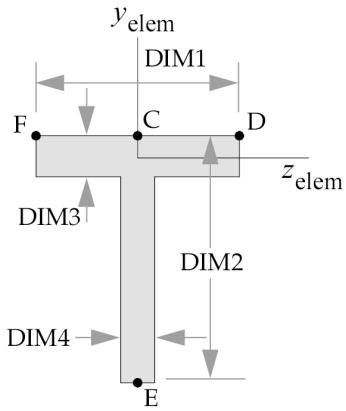
TYPE="L"



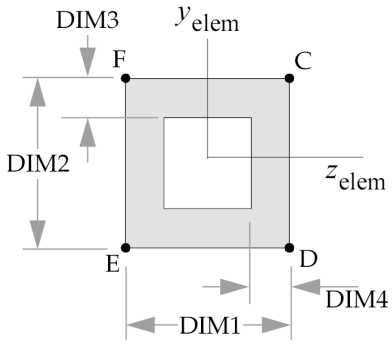
TYPE="I"



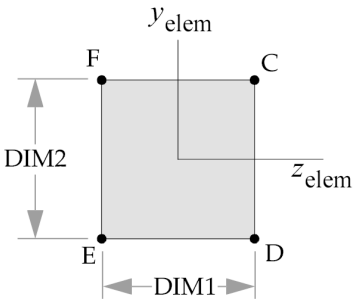
TYPE="CHAN"



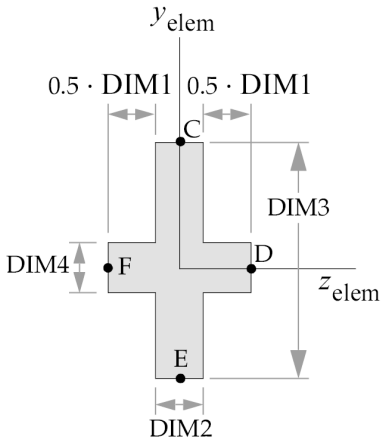
TYPE="T"



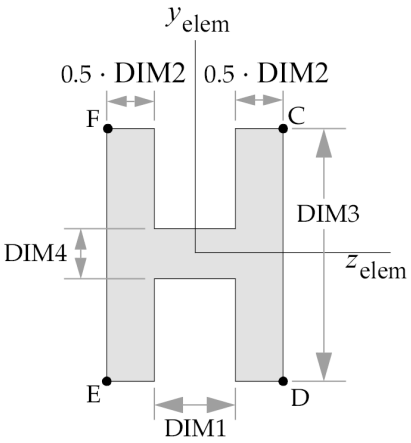
TYPE="BOX"



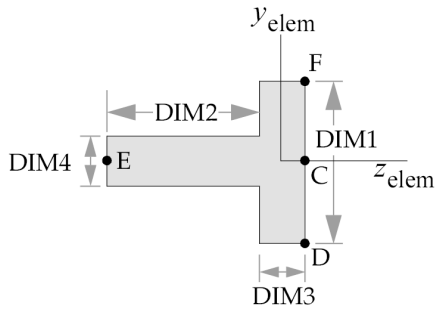
TYPE="BAR"



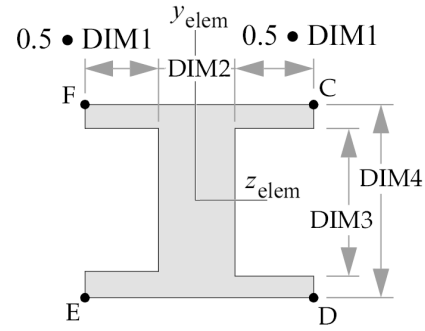
TYPE="CROSS"



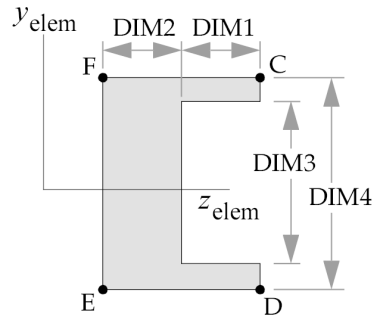
TYPE="H"



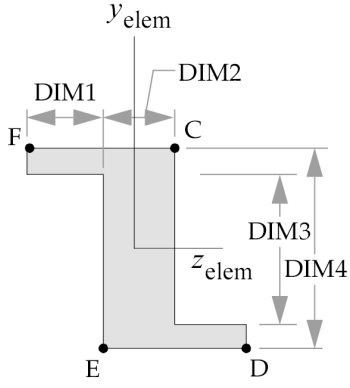
TYPE="T1"



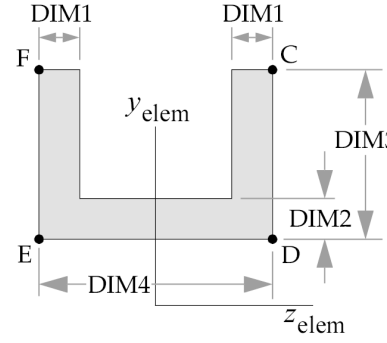
TYPE="I1"



TYPE="CHAN1"



TYPE="Z"



TYPE="CHAN2"

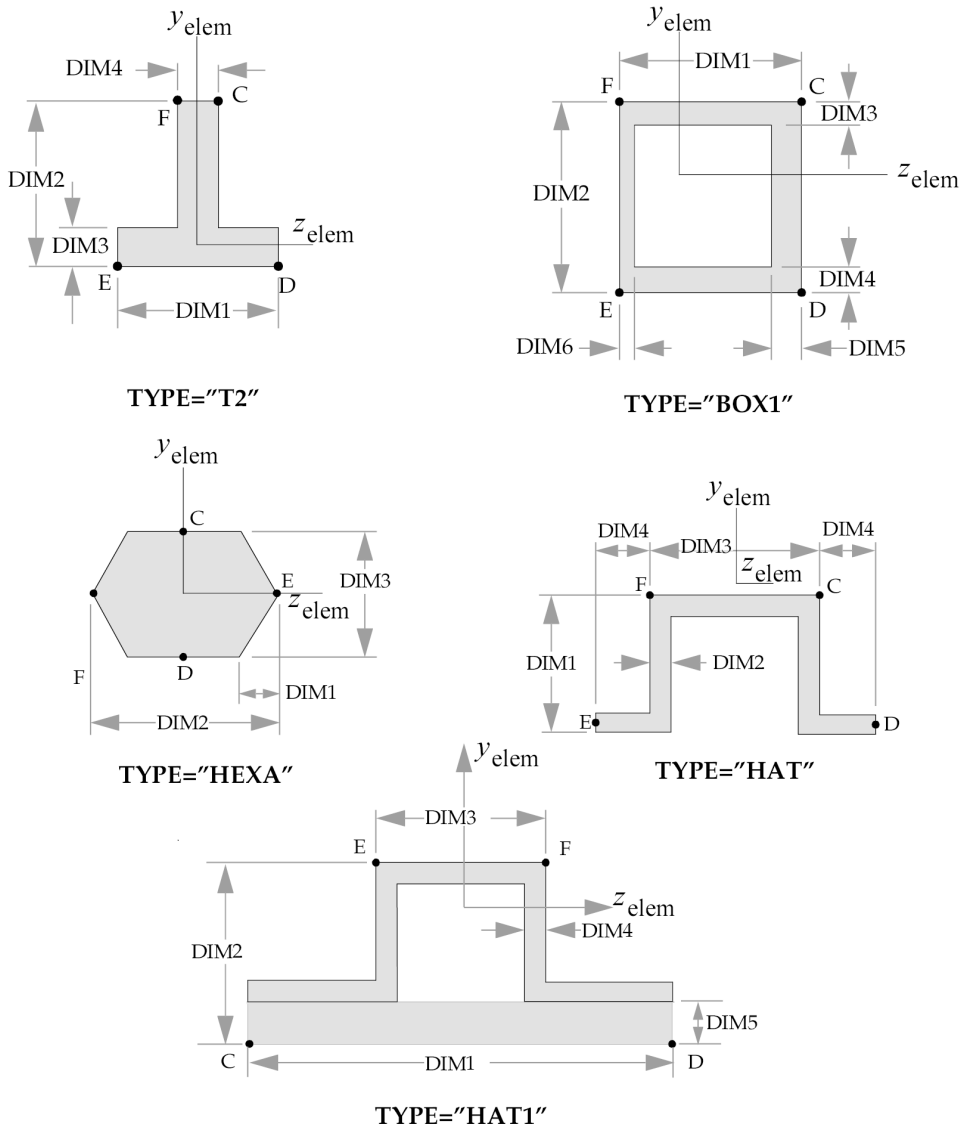


Figure 16-7. Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

PBEAR

Bearing Property Definition

Defines stiffness and viscous damping matrices for bearing connection. Applicable to all rotor dynamics solution types (SOLs 101, 107, 108, 109, 110, 111, 112).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBEAR	PID	TYPE	TXX	TXY	TYX	TYY		NOMVAL1	

CONTINUATION LINE FORMATS:

1	2	3	4	5	6	7	8	9	10
		“COM”	C1	C1Z	D1O				
		TYPEZ	TXZ	TYZ	TZX	TZY	TZZ	NOMVAL2	
		“COMZ”	C2	C2Z	D2O				
		TYPER	TRXRX	TRXRY	TRYRX	TRYRY		NOMVAL3	
		“COMR”	C3	C3Z	D3O				

The ordering of continuation lines is arbitrary.

EXAMPLES:

PBEAR	5	K	1001	1002	25.0	27.5			
		B	2001	2002	2003	2004			

PBEAR	5	KD	1001	1002	25.0	27.5		1.0E-1	
		BD	4001	4002	4002	4001			
		COM	1.0	0.0	1.0E-3				
		KDZ	2001	2002	2001	2002	2003	2.0E-1	
		BDZ	5001	5002	5001	5002	5003		
		COMZ	0.0	1.0	0.0				
		KDR	3001	3002	3002	1.0E5		1.0E-2	

		BDR	6001	6002	6002	1.0E2			
		COMR	1.0	0.0	0.0				

FIELDS:

Field	Contents
PID	Property identification number which is referenced by a CBEAR entry. (Integer > 0)
TYPE	Type of data in the TXX, TXY, TYX, and TYY fields on the same line. (Character: “K”, “B”, “KD”, “KF”, “BD”, “BF”)

- If TYPE = “K”, specifies constant stiffness or stiffness that is a function of rotor speed.
- If TYPE = “B”, specifies constant viscous damping or viscous damping that is a function of rotor speed.
- If TYPE = “KD”, specifies constant stiffness or stiffness that is a function of rotor speed and relative displacement.
- If TYPE = “KF”, specifies constant stiffness or stiffness that is a function of rotor speed and relative force.
- If TYPE = “BD”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative displacement.
- If TYPE = “BF”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative force.

If TYPE = “KD”, “KF”, “BD”, “BF”, see Remark 1.

Field	Contents
TXX, TXY, TYX, TYY	Stiffness or viscous damping matrix entry. See Remark 3. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 4) If TYPE = “K”, “B”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds. • If integer entry, identification number of a TABLEDi bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed. See Remark 5. If TYPE = “KD”, “KF”, “BD”, “BF”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds and relative displacements or rotor speeds and relative forces. • If integer entry, identification number of a TABLEST bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed and relative displacement or rotor speed and relative force. The TABLEST bulk entry references a series of TABLEDi bulk entries. The TABLEDi bulk entries contain tabular data of stiffness or viscous damping vs. rotor speed at constant values of relative displacement or relative force. See Remark 5 and Remark 8.
NOMVAL1	Valid if TYPE = “KD”, “KF”, “BD”, “BF”. Field is ignored if TYPE = “K”, “B”. See Remark 6. (Real ≥ 0.0; Default = 0.0) • For SOLs 101, 108, 109, 111, and 112, defines the relative displacement or relative force that is used to either directly compute the bearing stiffness or viscous damping, or initiate iteration for the bearing stiffness or viscous damping. See Remark 7. • For SOLs 107, 110, defines the relative displacement or relative force that is used to directly compute the bearing stiffness or viscous damping.
“COM”	COM flag. Indicates that coefficients for composite relative displacement radial equation or composite relative force radial equation follow. (Character)

Field	Contents
C1, C1Z, D1O	Coefficients for composite relative displacement radial equation or composite relative force radial equation. See Remark 9 and Remark 10. (Real; Defaults are C1 = 1.0, C1Z = 0.0, D1O = 0.0)
TYPEZ	Type of data in the TXZ, TYZ, TZX, TZY, and TZZ fields on the same line. (Character: “KZ”, “BZ”, “KDZ”, “KFZ”, “BDZ”, “BFZ”) • If TYPE = “KZ”, specifies constant stiffness or stiffness that is a function of rotor speed. • If TYPE = “BZ”, specifies constant viscous damping or viscous damping that is a function of rotor speed. • If TYPE = “KDZ”, specifies constant stiffness or stiffness that is a function of rotor speed and relative displacement. • If TYPE = “KFZ”, specifies constant stiffness or stiffness that is a function of rotor speed and relative force. • If TYPE = “BDZ”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative displacement. • If TYPE = “BFZ”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative force. If TYPE = “KDZ”, “KFZ”, “BDZ”, “BFZ”, see Remark 1.
TXZ, TYZ, TZX, TZY, TZZ	Stiffness or viscous damping matrix entry. See Remark 3. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 4) If TYPE = “KZ”, “BZ”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds. • If integer entry, identification number of a TABLEDi bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed. See Remark 5. If TYPE = “KDZ”, “KFZ”, “BDZ”, “BFZ”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds and relative displacements or rotor speeds and relative forces.

Field	Contents
	• If integer entry, identification number of a TABLEST bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed and relative displacement or rotor speed and relative force. The TABLEST bulk entry references a series of TABLEDi bulk entries. The TABLEDi bulk entries contain tabular data of stiffness or viscous damping vs. rotor speed at constant values of relative displacement or relative force. See Remark 5 and Remark 8.
NOMVAL2	Valid if TYPE = “KDZ”, “KFZ”, “BDZ”, “BFZ”. Field is ignored if TYPE = “KZ”, “BZ”. See Remark 6. (Real ≥ 0.0; Default = 0.0) • For SOLs 101, 108, 109, 111, and 112, defines the relative displacement or relative force that is used to either directly compute the bearing stiffness or viscous damping, or initiate iteration for the bearing stiffness or viscous damping. See Remark 7. • For SOLs 107, 110, defines the relative displacement or relative force that is used to directly compute the bearing stiffness or viscous damping.
“COMZ”	COMZ flag. Indicates that coefficients for composite relative displacement axial equation or composite relative force axial equation follow. (Character)
C2, C2Z, D2O	Coefficients for composite relative displacement axial equation or composite relative force axial equation. See Remark 9 and Remark 10. (Real; Defaults are C2 = 0.0, C2Z = 1.0, D2O = 0.0)
TYPER	Type of data in the TRXRX, TRXRY, TRYRX, and TRYRY fields on the same line. (Character: “KR”, “BR”, “KDR”, “KFR”, “BDR”, “BFR”) • If TYPE = “KR”, specifies constant stiffness or stiffness that is a function of rotor speed. • If TYPE = “BR”, specifies constant viscous damping or viscous damping that is a function of rotor speed. • If TYPE = “KDR”, specifies constant stiffness or stiffness that is a function of rotor speed and relative displacement. • If TYPE = “KFR”, specifies constant stiffness or stiffness that is a function of rotor speed and relative force.

Field	Contents
	• If TYPE = “BDR”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative displacement. • If TYPE = “BFR”, specifies constant viscous damping or viscous damping that is a function of rotor speed and relative force. If TYPE = “KDR”, “KFR”, “BDR”, “BFR”, see Remark 1.
TRXRX, TRXRY, TRYRX, TRYRY	Stiffness or viscous damping matrix entry. See Remark 3. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 4) If TYPE = “KR”, “BR”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds. • If integer entry, identification number of a TABLEDi bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed. See Remark 5. If TYPE = “KDR”, “KFR”, “BDR”, “BFR”: • If real entry, value of stiffness or viscous damping matrix entry used for all rotor speeds and relative displacements or rotor speeds and relative forces. • If integer entry, identification number of a TABLEST bulk entry that defines the stiffness or viscous damping matrix entry as a function of rotor speed and relative displacement or rotor speed and relative force. The TABLEST bulk entry references a series of TABLEDi bulk entries. The TABLEDi bulk entries contain tabular data of stiffness or viscous damping vs. rotor speed at constant values of relative displacement or relative force. See Remark 8.

Field	Contents
NOMVAL3	Valid if TYPE = “KDR”, “KFR”, “BDR”, “BFR”. Field is ignored if TYPE = “KR”, “BR”. See Remark 6. (Real ≥ 0.0; Default = 0.0) • For SOLs 101, 108, 109, 111, and 112, defines the relative displacement or relative force that is used to either directly compute the bearing stiffness or viscous damping, or initiate iteration for the bearing stiffness or viscous damping. See Remark 7. • For SOLs 107, 110, defines the relative displacement or relative force that is used to directly compute the bearing stiffness or viscous damping.
“COMR”	COMR flag. Indicates that coefficients for composite relative displacement rotational equation or composite relative force rotational equation follow. (Character)
C3, C3Z, D3O	Coefficients for composite relative displacement rotational equation or composite relative force rotational equation. See Remark 9 and Remark 10. (Real; Defaults are C3 = 1.0, C3Z = 0.0, D3O = 0.0)

REMARKS:

1. *Relative displacement* is the displacement between the two coincident grids used to define the CBEAR connection. *Relative force* is the force carried through the CBEAR connection. The relative force is calculated from the relative displacement.

2. Define speed-dependent bearing properties in the same units for speed that are specified in the RUNIT field of the ROTORD bulk entry.

3. “X”, “Y”, and “Z”, in TXX, TXY, TYX, TYY, TXZ, TYZ, TZX, TZY, and TZZ, refer to the X, Y, and Z-axes of the coordinate system referenced in the RCORDi field of the ROTORD bulk entry. “RX” and “RY” in TRXRX, TRXRY, TRYRX, and TRYRY refer to rotation about the X-axis and rotation about the Y-axis of the coordinate system referenced in the RCORDi field of the ROTORD bulk entry. In a rotor dynamic analysis, a rotor’s axis of rotation must be aligned with the Z-axis of the coordinate system referenced in the RCORDi field of the ROTORD bulk entry.

4. If any of TXX, TXY, TYX, TYY, TXZ, TYZ, TZX, TZY, TZZ, TRXRX, TRXRY, TRYRX, or TRYRY fields are blank or zero (either integer zero or real zero), the software uses real zero as the value for the corresponding field.

5. In TABLEDi bulk entries, enter the rotor speed data in the units that are specified in the RUNIT field of the ROTORD bulk entry.

6. If a NOMVALi is defined in both a stiffness and damping row, the value in the stiffness row is used.

7. When TYPE = “KD”, “KF”, “BD”, “BF”, “KDZ”, “KFZ”, “BDZ”, “BFZ”, “KDR”, “KFR”, “BDR”, or “BFR” for SOL 101, the method used to determine the stiffness and viscous damping for CBEAR elements depends on the value of MAXITER.

   - If MAXITER = 0, the value specified in the NOMVALi field is used to directly compute the bearing stiffness and viscous damping.

   - If MAXITER ≠ 0, the value specified in the NOMVALi field is used to initiate iteration over relative displacement or relative force. Consecutive iterations use the relative displacement and relative force from the previous iteration to update the bearing stiffness and viscous damping. The relative displacement or relative force magnitude on the CBEAR element at each iteration is compared with the value from the previous iteration to evaluate convergence. When iterating over relative displacement, convergence is reached when:

     |(Disp (old) - Disp (new)) / Disp (old)| < THRSHOLD

     When iterating over relative force, convergence is reached when:

     |(Force (old) - Force (new)) / Force (old)| < THRSHOLD

     Iterating stops when either convergence is met, or the number of iterations exceeds MAXITER.

     MAXITER and THRSHOLD are specified on the ROTORD bulk entry.

8. On a TABLEST bulk entry referenced by a PBEAR bulk entry, the values for relative displacement or relative force that correspond to the TABLEDi bulk entries must be consistently defined for all TABLEDi bulk entries. For example, a valid entry is as follows:

```
PBEAR   789     KD      6891    6892    6893    6894
$
$ TABLEST tables for PBEAR
$
TABLEST 6891
        0.0     7891    0.1     8891    0.3     9891    ENDT
TABLEST 6892
        0.0     7892    0.1     8892    0.3     9892    ENDT
TABLEST 6893
        0.0     7893    0.1     8893    0.3     9893    ENDT
TABLEST 6894
        0.0     7894    0.1     8894    0.3     9894    ENDT
```

The following entry is invalid:

```
PBEAR   789     KD      6891    6892    6893    6894
$
$ TABLEST tables for PBEAR
$
TABLEST 6891
        0.0     7891    0.1     8891    0.3     9891    ENDT
TABLEST 6892
        0.0     7892    0.1     8892    0.4     9892    ENDT
TABLEST 6893
        0.0     7893    0.1     8893    0.3     9893    ENDT
TABLEST 6894
        0.0     7894    0.1     8894    0.3     9894    ENDT
```

The following entry is also invalid:

```
PBEAR   789     KD      6891    6892    6893    6894
$
$ TABLEST tables for PBEAR
$
TABLEST 6891
        0.0     7891    0.1     8891    0.3     9891    ENDT
TABLEST 6892
        0.0     7892    0.1     8892    0.3     9892    ENDT
TABLEST 6893
        0.0     7893    0.1     8893    ENDT
TABLEST 6894
        0.0     7894    0.1     8894    0.3     9894    ENDT
```


However, the tabular data entered on the TABLEDi bulk entries that are referenced by a TABLEST bulk entry do not need to have the same range.

9. *Composite relative displacements* are linear combinations of radial and axial relative displacements. The software calculates the composite relative displacements from the following equations:

Δ_1 = C1 Δ_r + C1Z Δ_z + D1O
Δ_2 = C2 Δ_r + C2Z Δ_z + D2O
Δ_3 = C3 Δ_r + C3Z Δ_z + D3O

where Δ_1 is the composite radial relative displacement, Δ_2 is the composite axial relative displacement, Δ_3 is the composite rotational relative displacement, Δ_r is the radial relative displacement, and Δ_z is the axial relative displacement. The D1O, D2O, and D3O coefficients represent preload displacements.

The software calculates Δ_r as follows:

$\Delta_r = (\Delta_x^2 + \Delta_y^2)^{1/2}$

where Δ_x is the relative displacement in the x-direction and Δ_y is the relative displacement in the y-direction.

The software calculates Δ_z as follows:

$\Delta_z = \Delta_{GB} - \Delta_{GA}$

where Δ_{GA} and Δ_{GB} are the axial displacements of the grids listed on the CBEAR bulk entry.

For bearing properties that are speed and displacement-dependent, the software uses the composite relative displacements to look up values for bearing stiffness and bearing viscous damping. The matrix for bearing stiffness or bearing viscous damping is as follows:

$$\begin{Bmatrix} f_x \\ f_y \\ f_z \\ f_{rx} \\ f_{ry} \end{Bmatrix} = \begin{bmatrix} T_{xx}(\Delta_1,\Omega) & T_{xy}(\Delta_1,\Omega) & T_{xz}(\Delta_2,\Omega) & 0 & 0 \\ T_{yx}(\Delta_1,\Omega) & T_{yy}(\Delta_1,\Omega) & T_{yz}(\Delta_2,\Omega) & 0 & 0 \\ T_{zx}(\Delta_2,\Omega) & T_{zy}(\Delta_2,\Omega) & T_{zz}(\Delta_2,\Omega) & 0 & 0 \\ 0 & 0 & 0 & T_{rxrx}(\Delta_3,\Omega) & T_{rxry}(\Delta_3,\Omega) \\ 0 & 0 & 0 & T_{ryrx}(\Delta_3,\Omega) & T_{ryry}(\Delta_3,\Omega) \end{bmatrix} \begin{Bmatrix} \Delta_x \\ \Delta_y \\ \Delta_z \\ \Delta_{rx} \\ \Delta_{ry} \end{Bmatrix}$$

where $T_{ij}(\Delta_i, \Omega)$ are stiffness or viscous damping matrix entries as a function of composite relative displacement and angular speed of the rotor, Ω. If the matrix entries represent viscous damping, Δ_x, Δ_y, and Δ_z represent translational velocities, and Δ_{rx} and Δ_{ry} represent angular velocities.

If the calculated relative displacement is less than zero, the software automatically resets it to zero when it looks up values for bearing stiffness and bearing viscous damping.

10. *Composite relative forces* are linear combinations of radial and axial relative forces. The software calculates the composite relative forces from the following equations:

F_1 = C1 F_r + C1Z F_z + D1O
F_2 = C2 F_r + C2Z F_z + D2O
F_3 = C3 F_r + C3Z F_z + D3O

where F_1 is the composite radial relative force, F_2 is the composite axial relative force, F_3 is the composite rotational relative force, F_r is the radial relative force, and F_z is the axial relative force. The D1O, D2O, and D3O coefficients represent preload forces.

The software calculates F_r as follows:

$F_r = (F_x^2 + F_y^2)^{1/2}$

where F_x is the relative force in the x-direction and F_y is the relative force in the y-direction.

For bearing properties that are speed and force-dependent, the software uses the composite relative forces to look up values for bearing stiffness and bearing

viscous damping. The matrix for bearing stiffness or bearing viscous damping is as follows:

$$\begin{Bmatrix} f_x \\ f_y \\ f_z \\ f_{rx} \\ f_{ry} \end{Bmatrix} = \begin{bmatrix} T_{xx}(F_1,\Omega) & T_{xy}(F_1,\Omega) & T_{xz}(F_2,\Omega) & 0 & 0 \\ T_{yx}(F_1,\Omega) & T_{yy}(F_1,\Omega) & T_{yz}(F_2,\Omega) & 0 & 0 \\ T_{zx}(F_2,\Omega) & T_{zy}(F_2,\Omega) & T_{zz}(F_2,\Omega) & 0 & 0 \\ 0 & 0 & 0 & T_{rxrx}(F_3,\Omega) & T_{rxry}(F_3,\Omega) \\ 0 & 0 & 0 & T_{ryrx}(F_3,\Omega) & T_{ryry}(F_3,\Omega) \end{bmatrix} \begin{Bmatrix} \Delta_x \\ \Delta_y \\ \Delta_z \\ \Delta_{rx} \\ \Delta_{ry} \end{Bmatrix}$$

where $T_{ij}(F_i, \Omega)$ are stiffness or viscous damping matrix entries as a function of composite relative force and angular speed of the rotor, Ω. If the matrix entries represent viscous damping, Δ_x, Δ_y, and Δ_z represent translational velocities, and Δ_{rx} and Δ_{ry} represent angular velocities.

If the calculated relative force is less than zero, the software automatically resets it to zero when it looks up values for bearing stiffness and bearing viscous damping.

PBEND

Curved Beam or Pipe Element Property

Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	A	I1	I2	J	RB	THETAB	
	C1	C2	DI	D2	E1	E2	F1	F2	
	K1	K2	NSM	RC	ZC	DELTAN			

EXAMPLE:

PBEND	39	1	0.8	0.07	0.04	0.04	10.		
	0.5	0.4	-0.5	0.4					
	0.6	0.6				0.1			

ALTERNATE FORMAT AND EXAMPLE FOR ELBOWS AND CURVED PIPES:

PBEND	PID	MID	FSI	RM	T	P	RB	THETAB	
	SACL	ALPHA	NSM	RC	ZC	FLANGE			
	KX	KY	KZ		SY	SZ			

PBEND	39	1	1	0.5	0.02	10.	10.		
					0.1				

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 1 and 2. (Integer > 0)
A	Area of the beam cross section. (Real > 0.0)
I1, I2	Area moments of inertia in planes 1 and 2. (Real > 0.0)
J	Torsional stiffness. (Real > 0.0)
FSI	Flag selecting the flexibility and stress intensification factors. See“Flexibility and stress intensification factors” in the *NX Nastran Element Library*. (Integer = 1-6)
RM	Mean cross-sectional radius of the curved pipe. (Real > 0.0)
T	Wall thickness of the curved pipe. (Real ≥ 0.0; RM + T/2 < RB)
P	Internal pressure. (Real)
RB	Bend radius of the line of centroids. (Real. Optional, see CBEND entry.)
THETAB	Arc angle of element. (Real, in degrees. Optional, see CBEND entry.)
Ci, Di, Ei, Fi	The r,z locations from the geometric centroid for stress data recovery. See Remark 7. (Real)
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. (Real)
NSM	Nonstructural mass per unit length. (Real)
RC	Radial offset of the geometric centroid from points GA and GB. See Figure 16-8. (Real)
ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. See Figure 16-8. See Remark 8. (Real)
DELTAN	Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature. See Figure 16-8. See Remark 8 . (Real; Default is described in Remark 4.)

Field	Contents
SACL	Miter spacing at center line. See Figure 16-11 and Remark 9 . (Real > 0.0)
ALPHA	One-half angle between the adjacent miter axes (Degrees). (Real) Required for FSI=5 with miter bend. See Figure 16-11 and Remark 9.
FLANGE	For FSI=5, defines the number of flanges attached. See Remark 10. (Integer; Default=0)
KX	For FSI=6, the user defined flexibility factor for the torsional moment. (Real ≥ 1.0) See Remark 11.
KY	For FSI=6, the user defined flexibility factor for the out-of-plane bending moment. (Real ≥ 1.0) See Remark 11.
KZ	For FSI=6, the user defined flexbility factor for the in-plane bending moment. (Real ≥ 1.0) See Remark 11.
SY	For FSI=6, the user defined stress intensificatation factor for the out-of-plane bending. (Real ≥ 1.0)
SZ	For FSI=6, the user defined stress intensification factor for the in-plane bending. (Real ≥ 1.0)

REMARKS:

1. For structural problems, MID must reference a MAT1 material entry.
2. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
3. The transverse shear stiffness in planes 1 and 2 are K1*A*G and K2*A*G, respectively. The default values for K1 and K2 on the first format are zero, which means the transverse shear flexibilities (1/Ki*A*G) are set equal to zero. Transverse shear stiffness for the alternate format are automatically calculated for the curved pipe.
4. The neutral axis radial offset from the geometric centroid is default to the

$$\Delta N = \frac{\mathrm{I1}}{\mathrm{A} \cdot \mathrm{RB}}$$

It is recommended that the default be used whenever

$$\frac{(\mathrm{RB})^2 \mathrm{A}}{\mathrm{I1}} < 15$$

in which case the default value of ΔN is within 5% of the exact expression for circular or rectangular cross sections. For the alternate format, the neutral axis offset is calculated from an analytical expression for a hollow or solid circular cross section.

The user may compute an exact value for N as follows:

$$\Delta N = \frac{\mathrm{RB}}{1 + \frac{(\mathrm{RB})^2 \mathrm{A}}{Z}}$$

where

$$Z = \int \frac{r^2 \mathrm{dA}}{1 + \frac{r}{\mathrm{RB}}}$$

The integration is carried out over the cross section of the element.

5. If T is zero, a solid circular cross section of radius RM is assumed and FSI must be 1.

6. If the first format is used, third-order moments are neglected for the consistent mass matrix. These moments are zero whenever the cross section of the beam is symmetric about both the r and z axes.

7. If the circular cross-sectional property entry format is used, the stress points are automatically located at the points indicated in Figure 16-8.

8. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.

   - Offset vectors are not affected by thermal loads.

   - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200 and also nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)

9. For FSI=5, an elbow bend will be assumed if ALPHA is not defined (i.e. blank). If ALPHA is defined and SACL is not defined, then the widely spaced miter bend

factors will be used. When both SACL and ALPHA are defined, the closely spaced miter bend factors will be used if the spacing at center line is less than RM*(1+tan(ALPHA) where RM is the mean cross-sectional radius of the cuved pipe.

10. The FLANGE option is only supported when FSI=5 for elbow and closely spaced miter bend.

11. If a value less than 1 is entered for KX, KY, or KZ, its value will be reset to 1.0, a warning will be issued and the solve will continue.

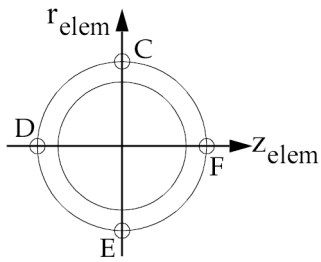


Figure 16-8. PBEND Circular Cross Section

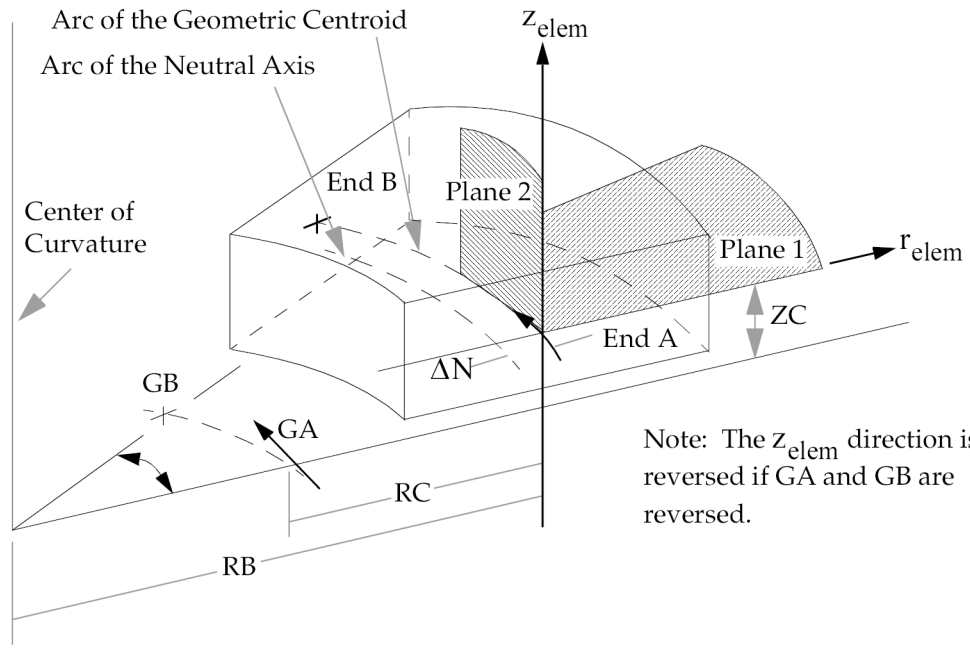


Figure 16-9. PBEND Element Coordinate System

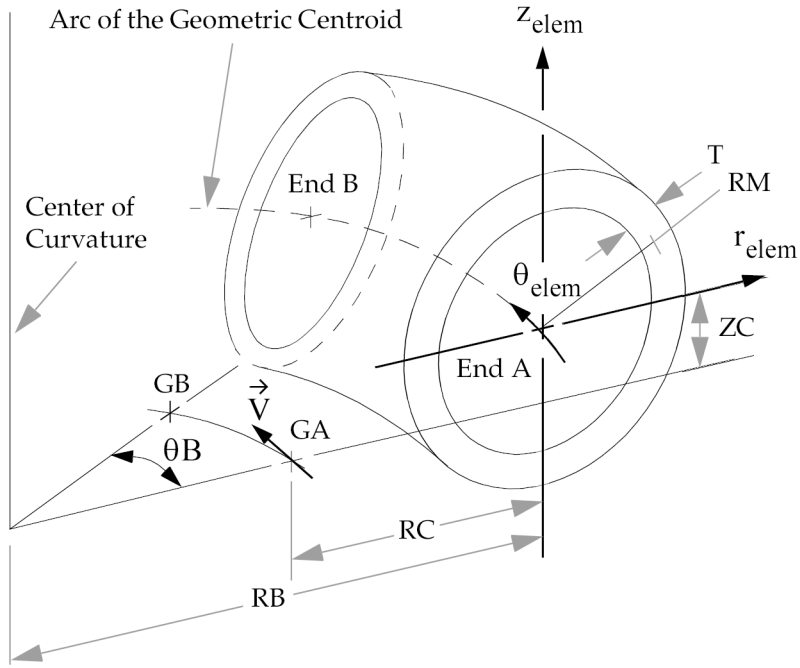


Figure 16-10. PBEND Circular Cross Section Element Coordinate System

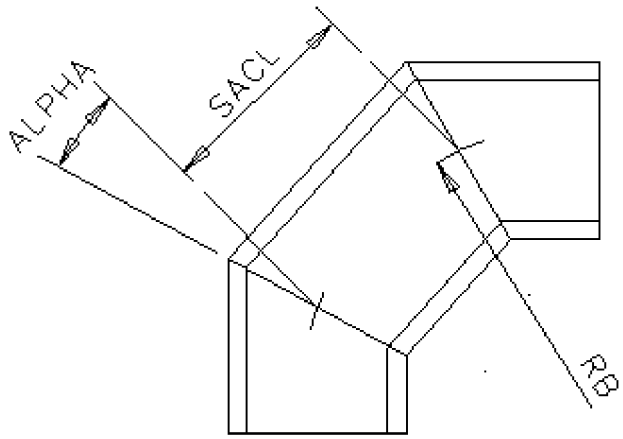


Figure 16-11. Definition of SACL and ALPHA

Where:

for a closely spaced miter:

$$R_B = \frac{(SACL)\cot(ALPHA)}{2}$$

for a widely spaced miter:

$$R_B = \frac{R_M(1 + \cot(ALPHA))}{2}$$

PBUSH

Generalized Spring-and-Damper Property

Defines the nominal property values for a generalized spring-and-damper structural element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
		“B”	B1	B2	B3	B4	B5	B6	
		“GE”	GE1	GE2	GE3	GE4	GE5	GE6	
		“RCV”	SA	ST	EA	ET			

EXAMPLE 1: STIFFNESS AND STRUCTURAL DAMPING ARE SPECIFIED.

PBUSH	35	K	4.35	2.4				3.1	
		GE	.06						
		RCV	7.3	3.3					

EXAMPLE 2: DAMPING FORCE PER UNIT VELOCITY IS SPECIFIED.

PBUSH	35	B	2.3						

EXAMPLE 3: ALL DAMPING TYPES ARE SPECIFIED.

PBUSH	2	K	1000.0		1000.0				
		B	0.02		0.02				
		GE	0.01		0.02				
		RCV	1.0	1.0					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks 2, 3 and 10. (Real; Default = 0.0)
“B”	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficient in units of force per unit velocity. (Real; Default = 0.0)
“GE”	Flag indicating that the next fields, 1–6, are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1–6. See Remarks 2 and 3. (Real; Default = 0.0)
“RCV”	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. (Character)
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)

REMARKS:

1. Ki, Bi, or GEi may be made frequency dependent for both direct and modal frequency response by use of the PBUSHT entry.

2. The nominal values are used for all analysis types except frequency response. For modal frequency response, the normal modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.

3. If PARAM,W4 is not specified, GEi is ignored in transient analysis.

4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces.

5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements.

6. The “K”, “B”, “GE”, or “RCV” entries may be specified in any order.

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

8. Applicable fields refer to directions in the element’s coordinate system.

9. For upward computability, if ONLY GE1 is specified on ALL PBUSH entries and GEi, i=2 – 6 are blank on ALL PBUSH entries, then a single structural damping for each PBUSH applied to all defined Ki for each PBUSH is assumed. If ANY PBUSH entry has a GEi, i=2 – 6 specified, then the GEi fields are considered variable on ALL PBUSH entries.

10. Rotational stiffness should be specified as moment per radian.

REMARK RELATED TO SOL 601:

1. The “GE” and “RCV” entries are not supported.

PBUSH1D

Rod Type Spring-and-Damper Property

Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBUSH1D	PID	K	C	M		SA	SE		
	“SHOCKA”	TYPE	CVT	CVC	EXPVT	EXPVC	IDTS		
			IDETS	IDECS	IDETSD	IDECSD			
	“SPRING”	TYPE	IDT	IDC	IDTDU	IDCDU			
	“DAMPER”	TYPE	IDT	IDC	IDTDV	IDCDV			
	“GENER”		IDT	IDC	IDTDU	IDCDU	IDTDV	IDCDV	

EXAMPLE:

PBUSH1D	35	3000.	200.	300.					
	SHOCKA	TABLE	2.2	1.2	1.		200		

The continuation entries are optional. The four options, SHOCKA, SPRING, DAMPER, and GENER can appear in any order

FIELDS:

Field	Contents	Default
PID	Property identification number. (Integer > 0).	Required
K	Stiffness. (Real ≥ 0).	See Remark 1 .
C	Viscous damping. (Real ≥ 0).	See Remarks 1 and 2.
M	Total mass of the element. (Real ≥ 0).	Blank

Field	Contents	Default
SA	Stress recovery coefficient [1/area]. (Real ≥ 0).	Blank
SE	Strain recovery coefficient [1/length]. (Real ≥ 0).	Blank
SHOCKA	Character string specifying that the next 10 fieldsare coefficients of the following force versus velocity/displacement relationship. (Character). $$F(u, v) = C_v \cdot S(u) \cdot \mathrm{sign}(v) \cdot \lvert v \rvert^{\mathrm{EXPV}}$$ The force F, the displacement u, and the velocity v, are in the axial direction of the damper. The axis of the damper is defined by the two connecting grid points GA and GB on the CBUSH1D Bulk Data entry. The displacement u and the velocity v, are the relative displacement and the relative velocity with respect to the grid point GA. The scale factor S(u) must be defined with a table or with an equation.	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT, the fields IDETS, IDECS, IDETSD, and IDECSD are identification numbers of DEQATN entries. For TYPE = TABLE the field IDTS is an identification number of a TABLEDi entry. If no character string is provided (blanks), TYPE = TABLE is set.	TABLE
CVT	Viscous damping coefficient C_V for tension v > 0, force per unit velocity. (Real).	Required for SHOCKA
CVC	Viscous damping coefficient C_V for compression v > 0, force per unit velocity. (Real).	CVT
EXPVT	Exponent of velocity EXPV for tension v > 0.(Real).	1.
EXPVC	Exponent of velocity EXPV for compression v < 0. (Real).	EXPVT
IDTS	Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE. The TABLEDi entry defines the scale factor S, versus displacement u.	Required for SHOCKA and TYPE=TABLE

Field	Contents	Default
IDETS	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for tension u > 0.	Required for SHOCKA and TYPE=EQUAT
IDECS	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for compression u < 0.	IDETS
IDETSD	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for tension u>0.	Required for SHOCKA and TYPE=EQUAT
IDECSD	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for compression u < 0.	IDETSD
SPRING	Character string specifying that the next 5 fields define a nonlinear elastic spring element in terms of a force versus displacement relationship. (Character). $F(u) = F_T(u)$ Tension is u > 0 and compression is u < 0.	
DAMPER	Character string specifying that the next 5 fields define a nonlinear viscous element in terms of a force versus velocity relationship. (Character). $F(v) = F_T(u)$ Tension is v > 0 and compression is v < 0.	

Field	Contents	Default
GENER	Character string specifying that the next 7 fields define a general nonlinear elastic spring and viscous damper element in terms of a force versus displacement and velocity relationship. (Character). For this element, the relationship can only be defined with TYPE = EQUAT. $$F(u, v) = F_T(u, v)$$ Tension is u > 0 and compression is u < 0.For SPRING, DAMPER, and GENER, the remaining fields are	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT the following fields are identification numbers of DEQATN entries. For TYPE = TABLE the following field is an identification number of a TABLEDi entry. TYPE is ignored for GENER.	Required for SPRING or DAMPER
IDT	Identification number of a DEQATN entry for tension if TYPE = EQUAT. Identification numberof a TABLEDi entry for tension and compressionif TYPE = TABLE.	Required for SPRING, DAMPER, and GENER
IDC	Identification number of a DEQATN entry for compression if TYPE = EQUAT. Is ignored for TYPE = TABLE.	IDT
IDTDU	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for tension u > 0. For SPRING and GENER only.	Required if TYPE=EQUAT
IDCDU	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for compression u < 0. For SPRING and GENER only.	IDTDU

Field	Contents	Default
IDTDV	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for tension v > 0. For DAMPER and GENER only.	Required if TYPE=EQUAT
IDCDV	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for compression v < 0. For DAMPER and GENER only.	IDCDT

REMARKS:

1. Either the stiffness K or the damping C must be specified.
2. The damping C and mass M are ignored in static solution sequences.
3. The parameters defined on the continuation entries are used in nonlinear solution sequences only.
4. The linear parameters K and C are used in all solution sequences unless parameters on continuation entries are defined and a nonlinear solution sequence is used. Then, the parameters K and C are used for initial values in the first iteration of the first load step and the parameters from continuation entries overwrite the linear parameters thereafter. When SHOCKA, SPRING or GENER are specified, K is overwritten. When SHOCKA, DAMPER or GENER is specified, C is overwritten.
5. PBUSH1D may only be referenced by CBUSH1D elements in the residual structure which do not attach to omitted degrees-of-freedom.
6. The continuation entries SHOCKA, SPRING, DAMPER and GENER may be specified in any order. If more than one continuation entry is defined, then the forces of SHOCKA, SPRING, etc. are added. Multiple continuation entries of the same kind are not allowed, for example, multiple SPRING continuation entries.
7. For TYPE = TABLE, values on the TABLEDi entry are for tension and compression. If table values $f(u)$ are provided only for positive values u > 0, then it is assumed that $f(-u) = -f(u)$.

8. For TYPE = EQUAT, the equations for tension and compression can be different. If the identification numbers for compression are left blank, it is assumed that the equation for tension is also valid for compression.

REMARKS RELATED TO SOLS 601 AND 701

1. Only “SPRING” continuation entry line is supported. Warning is issued for other continuation entry lines.

2. Only the following fields are supported: PID, K, C, M, SA, TYPE and IDT in “SPRING”. Hence, the spring stiffness may be nonlinear but the damping is linear.

3. Only TYPE=TABLE (or blank) is supported. Otherwise, a fatal error is issued.

PBUSHT

Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		“KN”	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	

EXAMPLE:

PBUSHT	2	K	100	101					
		B	102	103					
		GE	104	105					

FIELDS:

Field	Contents
PID	Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)
TKIDi	Identification number of a TABLEDi entry that defines the stiffness vs. frequency relationship. See Remark 10. (Integer ≥ 0; Default = 0)
“B”	Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)

Field	Contents
TBIDi	Identification number of a TABLEDi entry that defines the force per unit velocity damping vs. frequency relationship. (Integer ≥ 0; Default = 0)
“GE”	Flag indicating that the next fields 1–6 are structural damping frequency table identification numbers. (Character)
TGEIDi	Identification number of a TABLEDi entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer ≥ 0; Default = 0)
“KN”	Flag indicating that the next 1 to 6 entries are nonlinear force-deflection table identification numbers. (Character)
TKNIDi	Identification number of a TABLEDi entry that defines the force vs. deflection relationship. (Integer ≥ 0; Default = 0)

REMARKS:

1. The “K”, “B”, and “GE” entries are associated with same entries on the PBUSH entry.
2. Only CBUSH elements in the residual structure that do not attach to any omitted degrees-of-freedom can reference a PID identifying both a PBUSH entry and a PBUSHT entry.
3. The nominal values defined on the PBUSH entry are used for all analysis types except frequency response and nonlinear analyses.
4. When frequency dependent stiffness is included in a modal frequency response, the modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. See the parameter SDAMPUP for more options.
5. The “K”, “B”, “GE” or “KN” entries may be specified in any order.
6. The PBUSHT entry is ignored in all solution sequences except frequency response and nonlinear analyses.
7. For nonlinear analysis, only the “KN” field is used.
8. For frequency responses, only the “K”, “B” and/or “GE” fields are used.

9. For upward computability, if ONLY TGEID1 is specified on ALL PBUSHT entries and TGEIDi, i=2 – 6 are blank on ALL PBUSHT entries, then a single structural damping table for each PBUSHT applied to all defined Ki for each PBUSH is assumed. If ANY PBUSH entry has a TGEIDi, i=2 – 6 specified, then the GEi fields on the PBUSH and the TGEIDi fields on the PBUSHT are considered variable on ALL PBUSH and PBUSHT entries.

10. Rotational stiffness should be specified as moment per radian.

11. With designed frequency dependent properties in SOL 200, currently only TABLED1 can be used, and the use of TABLED1 for this case is limited to the LINEAR, LINEAR default options for XAXIS and YAXIS. For frequency dependent properties not associated with design variables, other options and other TABLEDi can be used.

REMARKS RELATED TO SOL 601:

1. Only the “KN” entry is supported.

PCHOCK

Defines Properties for 2D Chocking Elements

Defines properties for CCHOCKi elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCHOCK	PID	MID	NGAP	GAPT					

EXAMPLE:

PCHOCK	100	1	6	1.0					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number for a MAT1 or MAT11 bulk entry. (Integer > 0; No default)
NGAP	Number of gaps. (Integer > 0; No default)
GAPT	Initial gap opening. See Remarks 1, 2, and 3. (Real; Default = 0.0)

REMARKS:

1. The value for GAPT is used for all grid points Gi in the connectivity of chocking elements that reference the PCHOCK entry, except for those grid points Gi where

GAPi on the chocking element bulk entry is specified. For those grid points, the GAPi value takes precedence.

2. If the value for GAPT or, if applicable, the value for GAPi on the corresponding CCHOCKi entry is negative, the software uses the value to calculate an initial compressive hoop strain.

3. Figure 16-12 shows a possible application for chocking elements and how the initial gap opening GAPT is defined.

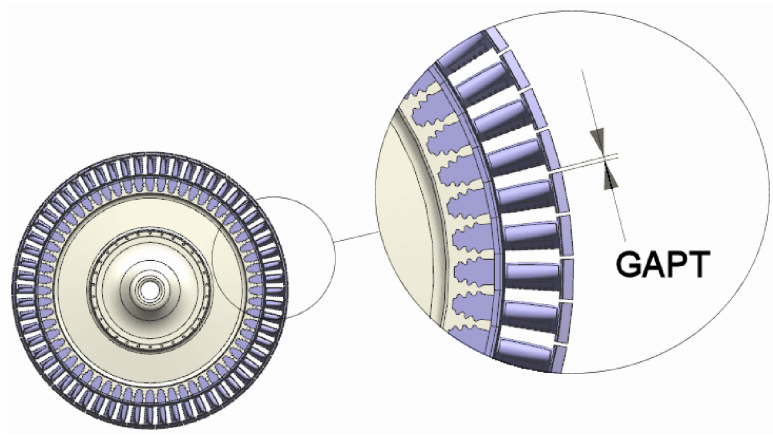


Figure 16-12. Initial Gap Opening

PCOMP

Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	SOUT3	-etc.-				

EXAMPLE:

PCOMP	181	-0.224	7.45	10000.0	HOFF				
	171	0.056	0.0	YES			45.0		
			-45.0				90.0		

FIELDS:

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark 14. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0) See Remark 12.

Field	Contents
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed): “HILL” for the Hill theory. “HOFF” for the Hoffman theory. “TSAI” for the Tsai-Wu theory. “STRN” for the Maximum Strain theory. See the Laminates chapter in the *NX Nastran User's Guide* for a detailed explanation of each theory.
TREF	Reference temperature. See Remark 5. (Real; Default = 0.0)
GE	Damping coefficient. See Remark 6 and Remark 16. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remark 17. “Blank” All plies must be specified and all stiffness terms are developed. “SYM” Only plies on one side of the laminate centerline are specified. The plies are numbered starting with 1 for the bottom ply. If the laminate contains an odd number of plies, then model the center ply as half the thickness of the actual center ply. “MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed. “BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed. “SMEAR” All plies must be specified, stacking sequence is ignored, MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero.

Field	Contents
	"SMCORE" Face plies on one side of the laminate and the core are specified to define a laminate that is symmetric about the midplane of the core. The core is specified last. When calculating face sheet stiffness, stacking sequence of the face sheets is ignored.
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs can refer to MAT1, MAT2, MAT8, MATSMA (SOL 601 only) or MATVE (SOL 601 only) bulk entries. See Remark 2 and SOL 601 Remark 4. (0 < Integer < 99999999 or blank, except MID1 must be specified.)
Ti	Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUTi	Controls individual ply stress and strain print or punch output. See Remark 7 and Remark 8. (Character: "YES" or "NO"; Default = "NO")

REMARKS:

1. PID must be unique with respect to all PCOMP, PCOMPG, and PSHELL entries.

2. The default for MIDi+1, ..., MIDn is the last defined MIDi. In the example above, MID(PLY1) is the default for MID(PLY2), MID(PLY3), and MID(PLY4). The same logic applies to Ti.

3. Composite shell elements do not support nonlinear elastic materials defined with the MATS1 bulk entry.

4. At least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.

5. A temperature dependent material defined with the combined MATi and MATTi entries can be referenced for a ply material (MIDi field on the PCOMP entry). For linear solutions, when computing the equivalent PSHELL and MAT2 entries from the PCOMP definition, the software uses TREF defined on the PCOMP entry to evaluate any temperature dependent material properties for the plies. TREF defaults to 0.0 if undefined. The TEMPERATURE(INIT) case control command is not used in this phase of the solution, although it must be defined, otherwise the software will ignore the temperature dependent material properties and use the properties on the referenced MATi entry. After the software creates the equivalent PSHELL and MAT2 entries, if a thermal load was defined with the TEMPERATURE(LOAD) case control command, the software will use the TEMPERATURE(INIT) command to compute thermal strains as described in the remarks on the TEMPERATURE case control command.

   By default, SOL 106 behaves as described above. Although, if PARAM, COMPMATT, YES is defined, SOL 106 will use the temperatures selected with the TEMPERATURE(LOAD) command to evaluate temperature dependent material properties for the plies when computing the equivalent PSHELL and MAT2 entries. A unique TEMPERATURE(LOAD) command in each subcase will result in the recomputing of the equivalent PSHELL and MAT2 entries. As described above for the linear solutions, the TEMPERATURE(INIT) case control command is also required in SOL 106 in order for the software to use the temperature dependent material properties when computing the equivalent PSHELL and MAT2 entries.

6. GE given on the PCOMP entry will be used for the element and the values supplied on material entries for individual plies are ignored. You are responsible for supplying the equivalent damping value on the PCOMP entry. GE is ignored in a transient analysis if PARAM,W4 is not specified. See the parameter W4.

7. The parameter NOCOMPS determines if stress and/or strain recovery is at the composite ply layers (default), on the equivalent PSHELL, or both. See the parameter NOCOMPS. The STRESS and/or STRAIN case control commands are required for any of these recovery options. When ply results are requested, stress and/or strain are computed at the middle of each ply. To print the ply stress and/or strain results, the case control command request must include the “PRINT” option (default). To punch these results, the case control command request must include the “PUNCH” option. SOUTi=YES should then be defined on any ply definitions in which you would like print or punch output. The SOUTi entry is not used in the computing or printing of failure indices. See Remark 9.

8. Stress and strain output for individual plies are available in all superelement static and normal modes analysis and requested by the STRESS and STRAIN case control commands.

9. To compute STRESS failure index, the following must be present:

   a. STRESS case control command.

b. The parameter NOCOMPS set to 1 (default) or 0.

c. SB and FT (= to HILL, HOFF or TSAI) on the PCOMP bulk entry.

d. Stress allowables Xt, Xc, Yt, Yc, and S on all referenced MAT8 bulk entries.

e. Stress allowables ST, SC, and SS on all referenced MAT1 bulk entries.

To compute STRAIN failure index, the following must be present:

a. STRESS case control command.

b. The parameter NOCOMPS set to 1 (default) or 0.

c. SB and FT (= STRN) on the PCOMP bulk entry.

d. Strain allowables Xt, Xc, Yt, Yc, S, and STRN=1.0 on all referenced MAT8 bulk entries.

By default, failure index output prints in the f06 file even when using the PLOT or PUNCH describers on the STRESS and STRAIN case control commands. The parameter entry PARAM,NOFISR,1 can be used to turn off the printing of the failure index output. See the parameter NOFISR.

10. To output strength ratio, the failure index output conditions listed in Remark 9 must exist, and the parameter SRCOMPS must equal “YES”. See the parameter SRCOMPS.

11. Stress resultant output can be requested with the FORCE case control command.

12. The failure index of the bonding material is calculated by:
$FI_{bonding} = ((\tau_{1z}, \tau_{2z})/$ allowable bonding stress).
The allowable bonding stress is defined on the SB field. The strength ratio for the bonding material is:
$SR_{bonding} = (1 / FI_{bonding})$.

13. The software automatically creates equivalent PSHELL and MATi entries from a PCOMP definition. You can optionally include a sorted echo request to print the derived PSHELL and MATi entries in User Information Message 4379, or to the punch file. The parameter NOCOMPS controls if stress and strain are computed for the composite elements, the equivalent homogeneous element, or both. See the parameter NOCOMPS. The software designates the equivalent homogeneous elements with a MID1 or MID2 ID greater than or equal to 10^8 on the PSHELL entry. Homogenous stresses are based upon a smeared representation of the laminate’s properties and in general will be significantly different than the more accurate lamina stresses available from PCOMP-based elements.

14. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses

are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.

15. An unsymmetrical layup or the use of Z0 to specify an unsymmetrical layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetrical layup.

16. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

17. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide the following special purpose stiffness calculations: MEM option only considers membrane effects, BEND option only considers bending effects, SMEAR ignores stacking sequence and is intended for cases where the sequence is not yet known, SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.

18. Element output for the SMEAR and SMCORE options are produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.

19. When the PCOMP or PCOMPG bulk entries are included in a distributed parallel method, the gpart keyword used for selecting the partitioning method must be gpart=1.

20. PCOMP is supported in all solutions except SOL 153 or 159 heat transfer analysis, and 701.

21. For elements referencing a PCOMP, stress and strain output for the individual lamina is supported in solutions 101, 103, 105, 106, 108, 109, 111, 112, 114, 129, 144, 200, and 601. In other solutions, stress and strain can only be recovered for the equivalent laminate. That is, output on the equivalent PSHELL created by the software.

REMARKS RELATED TO SOL 601:

1. Z0, NSM, SB, FT, TREF, GE, LAM and SOUTi are ignored.

2. When the STRESS and/or STRAIN case control commands are defined, results at the composite ply layers are computed. Stress and strain components are computed at the center of each ply. Inter-laminar results, failure indices, and strength ratios are not computed. Stress resultant output is not supported.

3. Large strain formulation is not available for multi-layered shell elements.

4. Elasto-plastic material model is supported, but not nonlinear elastic material model. That is, a MATS1 entry with TYPE=PLASTIC is supported, but not TYPE=NLELAST.

PCOMPG

Layered Composite Element Property with global ply IDs

Defines the properties of an n-ply composite material laminate which includes global ply IDs.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYIDi	MIDi	Ti	THETAi	SOUTi				

EXAMPLE:

PCOMPG	73	–2.E-4	0.0	8.E+7	TSAI				
	101	1	1.E-4	0.	YES				
	102		1.E-4	0.	YES				
	103		1.E-4	0.	YES				
	104		1.E-4	0.	YES				

FIELDS:

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark 14. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0) See Remark 12.

Field	Contents
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed): “HILL” for the Hill theory “HOFF” for the Hoffman theory “TSAI” for the Tsai-Wu theory “STRN” for the Maximum Strain theory See the Laminates chapter in the *NX Nastran User's Guide* for a detailed explanation of each theory.
TREF	Reference temperature. See Remark 5. (Real; Default = 0.0)
GE	Damping coefficient. See Remarks 7 and 16. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remark 17. “Blank” All plies must be specified and all stiffness terms are developed. “MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed. “BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed. “SMEAR” All plies must be specified, stacking sequence is ignored, MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero. “SMCORE” Face plies on one side of the laminate and the core are specified to define a laminate that is symmetric about the midplane of the core. The core is specified last. When calculating face sheet stiffness, stacking sequence of the face sheets is ignored.
GPLYIDi	Global ply IDs. See Remark 2. (Integer > 0)

Field	Contents
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, MAT8, MATSMA (SOL 601 only) or MATVE (SOL 601 only) bulk entries. See Remark 4. (0 < Integer < 99999999 or blank, except MID1 must be specified.)
Ti	Thicknesses of the various plies. See Remark 4. (Real or blank, except T1 must be specified.)
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with the first listed at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUTi	Controls individual ply stress and strain print or punch output. See Remarks 8 and 9. (Character: “YES” or “NO”; Default = “NO”)

REMARKS:

1. PID must be unique with respect to all PCOMP, PCOMPG, and PSHELL entries.

2. Each global ply identification number GPLYIDi in a single PCOMPG entry should be unique.

   The global ply identification numbers (GPLYIDi) are reused across different PCOMPG bulk entires in order to post-process relative ply layers with common GPLYIDi.

3. Composite shell elements do not support nonlinear elastic materials defined with the MATS1 bulk entry.

4. The default for MIDi+1, ..., MIDn is the last defined MIDi. In the example above, MID(PLY1) is the default for MID(PLY2), MID(PLY3), and MID(PLY4). The same logic applies to Ti.

5. A temperature dependent material defined with the combined MATi and MATTi entries can be referenced for a ply material (MIDi field on the PCOMPG entry). For linear solutions, when computing the equivalent PSHELL and MAT2 entries from the PCOMPG definition, the software uses TREF defined on the PCOMPG

entry to evaluate any temperature dependent material properties for the plies. TREF defaults to 0.0 if undefined. The TEMPERATURE(INIT) case control command is not used in this phase of the solution, although it must be defined, otherwise the software will ignore the temperature dependent material properties and use the properties on the referenced MATi entry. After the software creates the equivalent PSHELL and MAT2 entries, if a thermal load was defined with the TEMPERATURE(LOAD) case control command, the software will use the TEMPERATURE(INIT) command to compute thermal strains as described in the remarks on the TEMPERATURE case control command.

By default, SOL 106 behaves as described above. Although, if PARAM, COMPMATT, YES is defined, SOL 106 will use the temperatures selected with the TEMPERATURE(LOAD) command to evaluate temperature dependent material properties for the plies when computing the equivalent PSHELL and MAT2 entries. A unique TEMPERATURE(LOAD) command in each subcase will result in the recomputing of the equivalent PSHELL and MAT2 entries. As described above for the linear solutions, the TEMPERATURE(INIT) case control command is also required in SOL 106 in order for the software to use the temperature dependent material properties when computing the equivalent PSHELL and MAT2 entries.

6. At least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.

7. GE given on the PCOMPG entry will be used for the element and the values supplied on material entries for individual plies are ignored. You are responsible for supplying the equivalent damping value on the PCOMPG entry. GE is ignored in a transient analysis if PARAM,W4 is not specified. See the parameter W4.

8. The parameter NOCOMPS determines if stress and/or strain recovery is at the composite ply layers (default), on the equivalent PSHELL, or both. See the parameter NOCOMPS. The STRESS and/or STRAIN case control commands are required for any of these recovery options. When ply results are requested, stress and/or strain are computed at the middle of each ply. To print the ply stress and/or strain results, the case control command request must include the “PRINT” option (default). To punch these results, the case control command request must include the “PUNCH” option. SOUTi=YES should then be defined on any ply definitions in which you would like print or punch output. The SOUTi entry is not used in the computing or printing of failure indices. See Remark 10.

9. Stress and strain output for individual plies are available in all superelement static and normal modes analysis and requested by the STRESS and STRAIN case control commands.

10. To output STRESS failure index, the following must be present:

    a. STRESS case control command.

    b. The parameter NOCOMPS set to 1 (default) or 0.

c. SB and FT (= to HILL, HOFF or TSAI) on the PCOMPG Bulk Data entry.

d. Stress allowables Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries.

e. Stress allowables ST, SC, and SS on all referenced MAT1 Bulk Data entries.

To output STRAIN failure index, the following must be present:

a. STRESS case control command.

b. The parameter NOCOMPS set to 1 (default) or 0.

c. SB and FT (= STRN) on the PCOMPG Bulk Data entry.

d. Strain allowables Xt, Xc, Yt, Yc, S, and STRN=1.0 on all referenced MAT8 Bulk Data entries.

By default, failure index output prints in the f06 file even when using the PLOT or PUNCH describers on the STRESS and STRAIN case control commands. The parameter entry PARAM,NOFISR,1 can be used to turn off the printing of the failure index output. See the parameter NOFISR.

11. To output strength ratio, the failure index output conditions listed in Remark 10 must exist, and the parameter SRCOMPS must equal “YES”. See the parameter SRCOMPS.

12. The failure index of the bonding material is calculated by:
$FI_{bonding} = ((\tau_{1z}, \tau_{2z})/$ allowable bonding stress).
The allowable bonding stress is defined on the SB field. The strength ratio for the bonding material is:
$SR_{bonding} = (1 / FI_{bonding})$.

13. The software automatically creates equivalent PSHELL and MATi entries from a PCOMPG definition. You can optionally include a sorted echo request to print the derived PSHELL and MATi entries in User Information Message 4379, or to the punch file. The parameter NOCOMPS controls if stress and strain are computed for the composite elements, the equivalent homogeneous element, or both. See the parameter NOCOMPS. The software designates the equivalent homogeneous elements with a MID1 or MID2 ID greater than or equal to 10^8 on the PSHELL entry. Homogenous stresses are based upon a smeared representation of the laminate’s properties and in general will be significantly different than the more accurate lamina stresses available from PCOMP-based elements.

14. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.

15. An unsymmetrical layup or the use of Z0 to specify an unsymmetrical layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetrical layup.

16. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

17. The MEM, BEND, SMEAR and SMCORE options provide the following special purpose stiffness calculations: MEM option only considers membrane effects, BEND option only considers bending effects, SMEAR ignores stacking sequence and is intended for cases where the sequence is not yet known, SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.

18. Element output for the SMEAR and SMCORE options are produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.

19. When the PCOMP or PCOMPG bulk entries are included in a distributed parallel method (DMP), the gpart keyword used for selecting the partitioning method must be gpart=1.

20. PCOMPG is supported in all solutions except SOL 153 or 159 heat transfer analysis, 601 and 701.

21. For elements referencing a PCOMPG, stress and strain output for the individual lamina is supported in solutions 101, 103, 105, 106, 108, 109, 111, 112, 114, 129, 144, and 200. In other solutions, stress and strain can only be recovered for the equivalent laminate. That is, output on the equivalent PSHELL created by the software.

REMARKS RELATED TO SOL 601:

1. Z0, NSM, SB, FT, TREF, GE, LAM, and SOUTi are ignored.

2. When the STRESS and/or STRAIN case control commands are defined, results at the composite ply layers are computed. Stress and strain components are computed at the center of each ply. Inter-laminar results, failure indices, and strength ratios are not computed. Stress resultant output is not supported.

3. Large strain formulation is not available for multi-layered shell elements.

4. Elasto-plastic material model is supported, but not the nonlinear elastic material model. That is, a MATS1 entry with TYPE=PLASTIC is supported, but not TYPE=NLELAST.

PCOMPS

Layered Composite Element Property for Solid Elements

Defines the properties of an n-ply composite material laminate for CHEXA and CPENTA solid elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCOMPS	PID	CORDM	PSDIR	SB	NB	TREF	GE		
	GPLYIDi	MIDi	TRi	THETAi	FTi	ILFTi	SOUTi		

EXAMPLE:

PCOMPS	20	2	13	10000.					
	1	1	0.02	45.	TSAI	NB	YES		
	2	1	0.03	–45.	HILL	SB	YES		
	3	2	0.05	0.	TSAI	SB	YES		
	4	1	0.03	–45.	HILL	SB	YES		
	5	1	0.02	45.	TSAI	NB	YES		

FIELDS:

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
CORDM	Identification number of the coordinate system that the software uses to determine the orientation of the material coordinate system. See Remark 1. Enter “0” or leave blank to use the basic coordinate system. (Integer; Default = 0)

Field	Contents
PSDIR	Ply and stack directions. Enter the X-, Y-, and Z-axes of CORDM as 1, 2, and 3, respectively. See Remark 1. (Integer; 12,13,21,23,31,32; Default = 13)
SB	Allowable inter-laminar shear stress of the bonding material. (Real > 0.0; Default = Failure index not calculated)
NB	Allowable inter-laminar normal stress of the bonding material. (Real > 0.0; Default = Failure index not calculated)
TREF	Reference temperature. (Real; Default = 0.0)
GE	Damping coefficient. (Real; Default = 0.0)
GPLYIDi	Global ply IDs. For information on stacking order, see Remark 2. (Integer > 0; No default)
MIDi	Material ID of the various plies. The MIDs must refer to MAT1, MAT9, or MAT11 bulk entries. For progressive ply failure, the MIDs must refer to MAT11 bulk entries. (Integer > 0; For default behavior, see Remark 3)
TRi	Ply thickness. See Remark 4. (Real > 0.0; No default)
THETAi	Ply orientation angle. (Real; Default = 0.0)
FTi	Ply failure theory. See Remark 5. Allowable entries are: “HILL” for the Hill failure theory. “HOFF” for the Hoffman failure theory. “TSAI” for the Tsai-Wu failure theory. “STRN” for the Maximum Strain failure theory. “STRS” for the Maximum Stress failure theory. “TS” for the Maximum Transverse Shear Stress failure theory. “PFA” for progressive ply failure. See Remark 6. (Character; Default = No failure theory)

Field	Contents
ILFTi	Inter-laminar failure theory. Allowable entries are: “SB” for transverse shear stress failure index. “NB” for normal stress failure index. (Character; Default = No failure index)
SOUTi	Controls individual ply stress, strain, and damage output. See Remark 9. Allowable entries are: “NO” for do not compute. “YES” for compute. (Character; Default = “NO”)

REMARKS:

1. The software uses the CORDM, PSDIR, and THETAi specifications and the element coordinate system to define the orientation of the material coordinates. For CHEXA elements, the procedure is as follows:

   a. To determine the Z_m–axis (the stacking direction), the software uses the absolute value of direction cosines to determine which element coordinate direction is most closely aligned with the CORDM axis specified by the second number in the PSDIR field. The most closely aligned element coordinate direction is used as the Z_m–axis.

   b. To determine the X_m–axis for the ith ply, the software projects the CORDM axis specified by the first number in the PSDIR field onto the plane normal to the Z_m–axis. The software rotates the projected axis about the Z_m–axis through angle THETAi to obtain the X_m–axis for the ith ply. The positive sense of rotation for THETAi is about the positive Z_m–axis.

   c. For each ply, the software selects the Y_m–axis so that it is orthogonal to the X_m–axis for the ply, and the Z_m–axis, and yields a right-hand coordinate system.

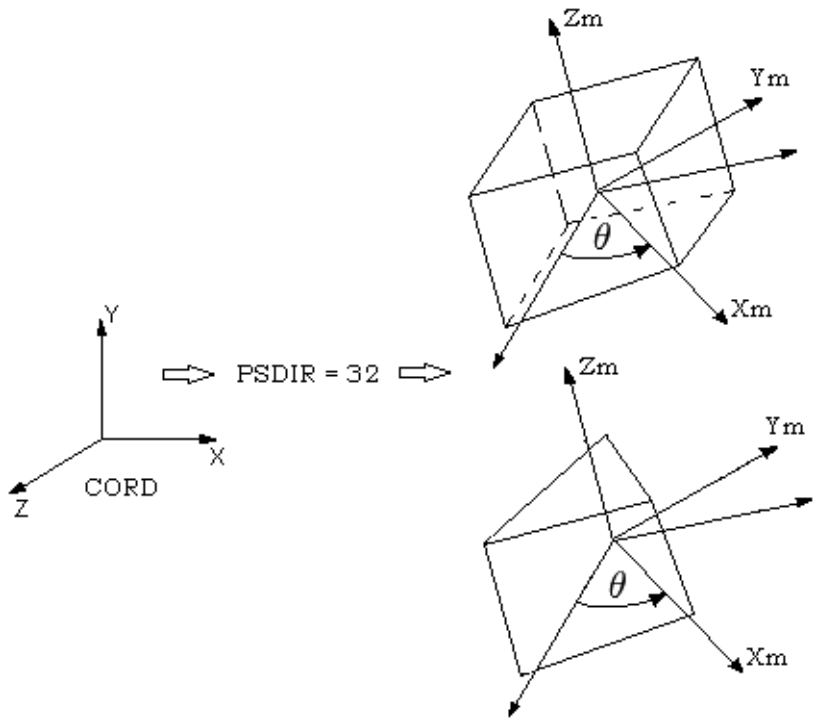
   For CPENTA elements, the procedure is as follows:

a. To determine the Z_m–axis (the stacking direction), the software selects the element coordinate direction that is normal to the triangular faces of the element.

b. To determine the X_m–axis for the ith ply, the software projects the CORDM axis specified by the first number in the PSDIR field onto the plane normal to the Z_m–axis. The software rotates the projected axis about the Z_m–axis through angle THETAi to obtain the X_m–axis for the ith ply. The positive sense of rotation for THETAi is about the positive Z_m–axis.

c. For each ply, the software selects the Y_m–axis so that it is orthogonal to the X_m–axis for the ply, and the Z_m–axis, and yields a right-hand coordinate system.

The figure shows the material coordinates for the ith ply in a CHEXA and a CPENTA element when PSDIR=32.



2. The ply stacking order is the same as the order of the continuation lines. The ply defined in the first continuation line is ply 1. The ply defined in the second continuation line is ply 2, and so on. Thus, the stacking order is independent of the GPLYIDi values.

Because the orientation of an element coordinate system depends on element connectivity, the positive sense for a pair of parallel Z_m–axes can differ. To allow for a single PCOMPS definition to be applied across many elements, the software uses the positive sense of the CORDM axis that is used to determine the Z_m–axis for each element as the direction of increasing ply number. That is, ply 2 is further out the positive axis than ply 1.

3. MID1 must be defined. If all other MIDi are blank, they default to the value in the MID1 field. If multiple, but not all, MIDi fields are defined, the blank fields default to the last defined MIDi. For example, if a laminate consists of six plies and MID1 and MID4 are defined, MID1 will be used for the first three plies, and MID4 will be used for the last three plies.

4. The laminate thickness is adjusted at the corners to coincide with the distance between grid points. The thickness of each ply in the laminate is adjusted proportionally.

5. FTi is failure theory for the ith ply. The material properties for all failure theories except progressive ply failure are specified by MATFT bulk entries. The material properties for progressive ply failure are specified by MATDMG bulk entries. For the ith ply, the software references the MATFT or MATDMG bulk entry that has the same MID as that used for the MIDi field.

6. To considers in-plane progressive damage to a ply, use progressive ply failure. To consider inter-laminar failure, use the SB or NB options in the ILFTi field. To consider inter-laminar progressive failure and delamination, leave the ILFTi field blank and use cohesive elements to model the inter-laminar behavior.

7. To compute a ply and/or bonding failure index, the STRESS case control command must be present, SOUTi on the PCOMPS bulk entry must be set to “YES”, and the following must be defined.

   For a stress or strain ply failure index:

   a. FTi on the PCOMPS bulk entry.

   b. The stress or strain allowables on the referenced MATFT bulk entry.

   For a stress bonding failure index:

   a. ILFTi on the PCOMPS bulk entry.

   b. The stress allowables SB or NB on the PCOMPS bulk entry.

   By default, failure index output prints in the f06 file even when using the PLOT or PUNCH describers on the STRESS and STRAIN case control commands. The parameter entry PARAM,NOFISR,1 can be used to turn off the printing of the failure index output. See the parameter NOFISR.

8. Ply stress and strain results are always computed in the ply coordinate system.

9. To request that ply stress and/or strain be computed, the STRESS and/or STRAIN case control command must be defined with the appropriate PRINT, PUNCH, or PLOT output option, and the SOUTi field must equal “YES”. The STRESS and STRAIN commands also include the CPLYMID, CPLYBT, and CPLYBMT describers to specify stress or strain recovery at the bottom, middle, or top of the plies. See the remarks on the STRESS and STRAIN case control commands.

10. GPSTRESS or GPSTRAIN output is not supported.

11. Glue or contact definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the glue/contact definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BCRESULTS and BGRESULTS case control commands.

12. PCOMPS is supported in solutions 101, 103, 105, 108, 109, 111, 112, and 401.

13. For SOLs 101, 103, 105, 108, 109, 111, and 112, the software uses TREF on the PCOMPS entry as the reference temperature for all plies. The software ignores TREF on the MATi entries referenced by the plies. See the remarks for linear solutions on the TEMPERATURE case control command for information on how the reference temperature is used to compute thermal strain in these solutions.

    For SOL 401, the software uses TREF defined on the MATi entry referenced by a ply for the ply reference temperature. The software ignores TREF on the PCOMPS entry. See "Thermal Loads" in the *Multi-Step Nonlinear User's Guide* for information on how the reference temperature is used to compute thermal strain in SOL 401.

PCONV

Convection Property Definition

Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF					

EXAMPLE:

PCONV	3	2	0	.25					

FIELDS:

Field	Contents
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for free convection. (Integer 0, 1, 10, 11, 20, or 21; Default = 0)
EXPF	Free convection exponent as implemented within the context of the particular form that is chosen. See Remark 3. (Real ≥ 0.0; Default = 0.0)

REMARKS:

1. Every surface to which free convection is to be applied must reference a PCONV entry. PCONV is referenced on the CONV Bulk Data entry.

2. MID must reference a MAT4 entry that specifies the convection heat transfer coefficient (H).

3. EXPF is the free convection temperature exponent.

   - If FORM = 0, 10, or 20, EXPF is an exponent of (T – TAMB), where the convective heat transfer is represented as

$$q = H \cdot u_{\mathrm{CNTRLND}} \cdot (T - \mathrm{TAMB})^{\mathrm{EXPF}} \cdot (T - \mathrm{TAMB}) \ .$$

   - If FORM = 1, 11, or 21,

$$q = H \cdot u_{\mathrm{CNTRLND}} \cdot (T^{\mathrm{EXPF}} - \mathrm{TAMB}^{\mathrm{EXPF}})$$

     where T represents the elemental grid point temperatures and TAMB is the associated ambient temperature.

4. FORM specifies the formula type and the reference temperature location used in calculating the convection film coefficient if FLMND = 0.

   - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperatures (average).

   - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).

   - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperatures).

REMARKS RELATED TO SOL 601:

1. FORM and EXPF are ignored.

2. The convective heat transfer is represented as:

   - $q = H \cdot (T - \mathrm{TAMB})$, CNTRLND = 0

   - $q = (H \cdot u_{\mathrm{CNTRLND}})(T - \mathrm{TAMB})$, CNTRLND ≠ 0

3. In calculating the convection film coefficient, SOL 601 uses the surface temperatures at the integration points of the element (closest to FORM=10).

PCONVM

Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PCONVM	PCONID	MID	FORM	FLAG	COEF	EXPR	EXPPI	EXPPO	

EXAMPLE:

PCONVM	3	2	1	1	.023	0.80	0.40	0.30	

FIELDS:

Field	Contents
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)
FLAG	Flag for mass flow convection. (Integer = 0 or 1; Default = 0)
COEF	Constant coefficient used for forced convection. (Real > 0.0)
EXPR	Reynolds number convection exponent. (Real ≥ 0.0; Default = 0.0)
EXPPI	Prandtl number convection exponent for heat transfer into the working fluid. (Real ≥ 0.0; Default = 0.0)
EXPPO	Prandtl number convection exponent for heat transfer out of the working fluid. (Real ≥ 0.0; Default = 0.0)

REMARKS:

1. Every surface to which forced convection is applied must reference a PCONVM entry. PCONVM is referenced on the CONVM entry.

2. MID references a MAT4 entry that specifies material properties of the working fluid at the temperature of the point FLMND. FLMND is specified on the CONVM entry.

3. The material properties are used in conjunction with the average diameter and mass flow rate (mdot). MID references the material properties and supplies the fluid conductivity (k), heat capacity (cp), and viscosity (μ) needed to compute the Reynolds (Re) and Prandtl (Pr) numbers as follows:

$$\mathrm{Re} = (4 \cdot |\mathrm{mdot}|) / (\pi \cdot \mathrm{diameter} \cdot \mu)$$
$$\mathrm{Pr} = \mathrm{cp} \cdot \mu / k$$

4. FORM controls the type of formula used in determination of the forced convection film coefficient h. There are two cases:

   - If FORM = 0, 10, or 20 than $h = \mathrm{coef} \cdot \mathrm{Re}^{\mathrm{EXPR}} \cdot \mathrm{Pr}^{\mathrm{EXPP}}$.

   - If FORM = 1, 11, or 21 then the above h is multiplied by k and divided by the average hydraulic diameter.

   - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.

   - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).

   - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).

   - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).

5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.

6. FLAG controls the convective heat transfer into the downstream point (the second point as identified on the CHBDYi statement is downstream if mdot is positive).

   - FLAG = 0, no convective flow (stationary fluid).

   - FLAG = 1, convective energy flow that is consistent with the Streamwise Upwind Petrov Galerkin (SUPG) element formulation.

7. No phase change or internal heat generation capabilities exist for this element.

PDAMP

Scalar Damper Property

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

EXAMPLE:

PDAMP	14	2.3	2	6.1					

FIELDS:

Field	Contents
PIDi	Property identification number. (Integer > 0)
Bi	Force per unit velocity. (Real)

REMARKS:

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library.*

PDAMP5

Scalar Damper Property for CDAMP5

Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PDAMP5	PID	MID	B						

EXAMPLE:

PDAMP5	2	3	4.0						

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT4 or MAT5 entry. (Integer > 0)
B	Damping multiplier. (Real > 0.0)

REMARKS:

1. B is the mass that multiplies the heat capacity CP on the MAT4 or MAT5 entry.

PDAMPT

Frequency-Dependent Damper Property

Defines the frequency-dependent properties for a PDAMP Bulk Data entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PDAMPT	PID1	TBID1							

EXAMPLE:

PDAMPT	12	34							

FIELDS:

Field	Contents
PID	Property identification number that matches the identification number on a PDAMP entry. (Integer > 0)
TBID1	Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship. (Integer ≥ 0; Default = 0)

REMARKS:

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements in the residual structure, which do not attach to any omitted degrees-of-freedom.
2. The PDAMPT entry is ignored in all solution sequences except frequency response analysis (including within SOL 200).
3. With designed frequency dependent properties in SOL 200, currently only TABLED1 can be used, and the use of TABLED1 for this case is limited to the

LINEAR, LINEAR default options for XAXIS and YAXIS. For frequency dependent properties not associated with design variables, other options and other TABLEDi can be used.

PDUMi

Dummy Element Property

Defines the properties of a dummy element (1 ≤ i ≤ 9). Referenced by the CDUMi entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PDUMi	PID	MID	A1	A2	A3	A4	A5	A6	
	A7	-etc.-							

EXAMPLE:

PDUM3	108	2	2.4	9.6	1.E4	15.		3.5	
	5		2						

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
Aj	Additional fields. (Real or Integer)

REMARKS:

1. The additional fields are defined in the user-written element subroutines.

PELAS

Scalar Elastic Property

Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

EXAMPLE:

PELAS	7	4.29	0.06	7.92	27	2.17	0.0032		

FIELDS:

Field	Contents
PIDi	Property identification number. (Integer > 0)
Ki	Elastic property value. See Remarks 8, 9 and 10. (Real)
GEi	Damping coefficient, g_e. See Remarks 6, 7 and 8. (Real)
Si	Stress coefficient. (Real)

REMARKS:

1. Be careful using negative spring values.
2. Spring values are defined directly on the CELAS2 and CELAS4 entries, and therefore do not require a PELAS entry.

3. One or two elastic spring properties may be defined on a single entry.

4. The element stress is computed by multiplying the stress coefficient S with the recovered element force.

5. For a discussion of scalar elements, see *"Overview of 0D (Scalar) Elements"* in the *NX Nastran Element Library*.

6. If PARAM,W4 is not specified, GEi is ignored in transient analysis. See "Parameters and Parameter Applicability Tables".

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

8. If PELAST is used in conjunction with PELAS, the nominal values here should be Ki > 0 and/or GEi > 0, as applicable, as otherwise the corresponding PELAST entries will be ignored.

9. Rotational stiffness should be specified as moment per radian.

10. In solutions 153 and 159, the software calculates heat conduction for CELAS1 and CELAS3 elements as

    $q = K \times \Delta T$

    where K is the value entered on the PELAS entry.

REMARKS RELATED TO SOLS 601 AND 701:

GEi and Si are ignored.

PELAST

Frequency Dependent Elastic Property

Defines the frequency dependent properties for a PELAS Bulk Data entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PELAST	PID	TKID	TGEID	TKNID					

EXAMPLE:

PELAST	44	38							

FIELDS:

Field	Contents
PID	Property identification number that matches the identification number on a PELAS entry. (Integer > 0)
TKID	Identification number of a TABLEDi entry that defines the force per unit displacement vs. frequency relationship. (Integer > 0; Default = 0)
TGEID	Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0)
TKNID	Identification number of a TABELDi entry that defines the nonlinear force vs. displacement relationship. (Integer > 0; Default = 0)

REMARKS:

1. The PELAST entry may only be referenced by CELAS1 or CELAS3 elements in the residual structure which do not attach to any omitted degrees-of-freedom.

2. The nominal values defined on the PELAS entry are used for all analysis types except frequency response and nonlinear analyses.

3. When frequency dependent stiffness is included in a modal frequency response, the modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. See the parameter SDAMPUP for more options.

4. The following table summarizes the usage PELAST entry in various solution sequences.

Field	Frequency Response	Nonlinear	Linear (Non-Frequency Response)
TKID	Used	Ignored	Ignored
TGEID	Used	Ignored	Ignored
TKNID	Ignored	Used	Ignored

5. With designed frequency dependent properties in SOL 200, currently only TABLED1 can be used, and the use of TABLED1 for this case is limited to the LINEAR, LINEAR default options for XAXIS and YAXIS. For frequency dependent properties not associated with design variables, other options and other TABLEDi can be used.

REMARKS RELATED TO SOLS 601 AND 701:

1. TKID and TGEID are ignored.

PFAST

Define Properties for CFAST Connector

Defines properties for CFAST connector.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE					

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
PFAST	44	.85	11		130000	54000	11500		

FIELDS:

Field	Contents
PID	Property identification number of a PFAST bulk data entry (Integer > 0).
D	Diameter of the virtual connector. Used to locate the virtual grids on the shell element patch. (Real > 0)
MCID	Specifies a material coordinate system in which KTi and KRi are applied. (Integer >= -1 or blank. Default = -1) Remark 1.
MFLAG	Determines MCID is treated as absolute or relative. (Integer 0 or 1, Default = 0) When MFLAG = 0, MCID is a relative coordinate system. When MFLAG = 1, MCID is an absolute coordinate system. Remark 1
KTi	X,Y,Z translational stiffness values. (Real, Default = 0.0)

Field	Contents
KRi	X,Y,Z rotational stiffness values. (Real, Default = 0.0)
MASS	Mass of connector. (Real, Default = 0.0)
GE	Structural damping. (Real, Default = 0.0)

REMARKS:

1. The CFAST element coordinates and stiffness directions depend on how MCID and MFLAG are defined:

   A. When MCID = –1 (default), MFLAG is ignored. The element x-axis is defined from GA to GB. If zero length (GA and GB are coincident), the element x-axis is defined by the normal from shell A. Similar to the CWELD element, the element y-axis is perpendicular to the element x-axis, and is oriented relative to the closest basic coordinate axis. If two basic coordinate axis are equally close, the first of x, y, then z is taken. The element z-axis is the cross product of the element x- and y-axis. KT1, KT2, and KT3 are applied in the element x-, y-, and z- axis, respectively.

   B. When MCID > 0 and MFLAG=0, the element x-axis is defined as described in “A”. The cross product of the element x-axis into the MCID y-axis determines the element z-axis. The element y-axis is the cross product of the element z- and x-axis. KT1, KT2, and KT3 are applied in the element x-, y-, and z- axis, respectively.

   C. When MCID ≥ 0 and MFLAG = 1 (absolute), the element stiffness values KT1, KT2, and KT3 are applied in the material coordinates, although, the element forces are computed in the coordinates as described in “A”.

PGAP

Gap Element Property

Defines the properties of the gap element (CGAP entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1	MU2	
	TMAX	MAR	TRMIN						

EXAMPLE:

PGAP	2	.025	2.5	1.E6		1.E6	0.25	0.25	

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
U0	Initial gap opening. See Figure 16-14. (Real; Default = 0.0)
F0	Preload. See Figure 16-14. (Real ≥ 0.0; Default = 0.0)
KA	Axial stiffness for the closed gap; i.e., $U_a - U_b > U0$. See Figure 16-14. (Real ≥ 0.0)
KB	Axial stiffness for the open gap; i.e., $U_a - U_b < U0$. See Figure 16-14. See Remark 2 . (Real ≥ 0.0; Default = 1.0E-14 · KA)
KT	Transverse stiffness when the gap is closed. See Figure 16-15. It is recommended that KT ≥ (0.1 · KA). (Real ≥ 0.0; Default = MUI · KA)

Field	Contents
MU1	Coefficient of static friction (μ_s) for the adaptive gap element or coefficient of friction in the y transverse direction (μ_y) for the nonadaptive gap element. See Remark 3 and Figure 16-15. (Real ≥ 0.0; Default = 0.0)
MU2	Coefficient of kinetic friction (μ_k) for the adaptive gap element or coefficient of friction in the z transverse direction (μ_z) for the nonadaptive gap element. See Remark 3 and Figure 16-15. (Real ≥ 0.0 for the adaptive gap element, MU2 ≤ MU1; Default = MU1)
TMAX	Maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment. See Remark 4 (Real; Default = 0.0).
MAR	Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See Remark 5 (1.0 < Real < 10^6; Default = 100.0).
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 6 . (0.0 ≤ Real ≤ 1.0; Default = 0.001)

REMARKS:

1. Figure 16-13, Figure 16-14, and Figure 16-15 show the gap element and the force-displacement curves used in the stiffness and force computations for the element.

2. For most contact problems, KA (penalty value) should be chosen to be three orders of magnitude higher than the stiffness of the neighboring grid points. A much larger KA value may slow convergence or cause divergence, while a much smaller KA value may result in inaccurate results. The value is adjusted as necessary if TMAX > 0.0.

3. When the gap is open, there is no transverse stiffness. When the gap is closed and there is friction, the gap has the elastic stiffness (KT) in the transverse direction until the friction force is exceeded and slippage starts to occur.

4. There are two kinds of gap elements: adaptive gap and nonadaptive gap. If TMAX≥0.0, the adaptive gap element is selected by the program. When TMAX=0.0, penalty values will not be adjusted, but other adaptive features will be active (i.e., the gap-induced stiffness update, gap-induced bisection, and subincremental process). The value of TMAX = -1.0 selects the nonadaptive (old)

gap element. The recommended allowable penetration TMAX is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the gap.

5. The maximum adjustment ratio MAR is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K^{init}}{\mathrm{MAR}} \le K \le K^{init} \cdot \mathrm{MAR}$$

where K^{init} is either KA or KT.

6. TRMIN is used only for the penalty value adjustment in the adaptive gap element. The lower bound for the allowable penetration is computed by TRMIN * TMAX. The penalty values are decreased if the penetration is below the lower bound.

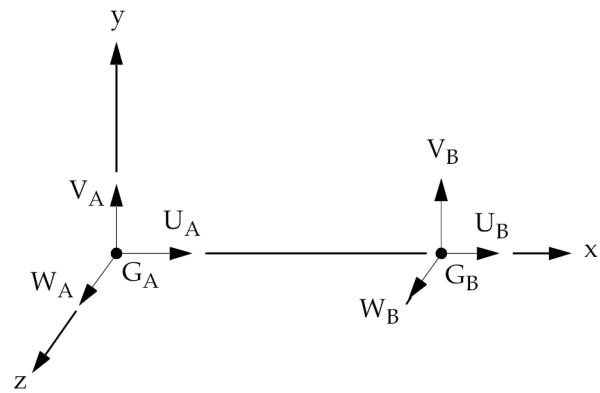


Figure 16-13. The CGAP Element Coordinate System

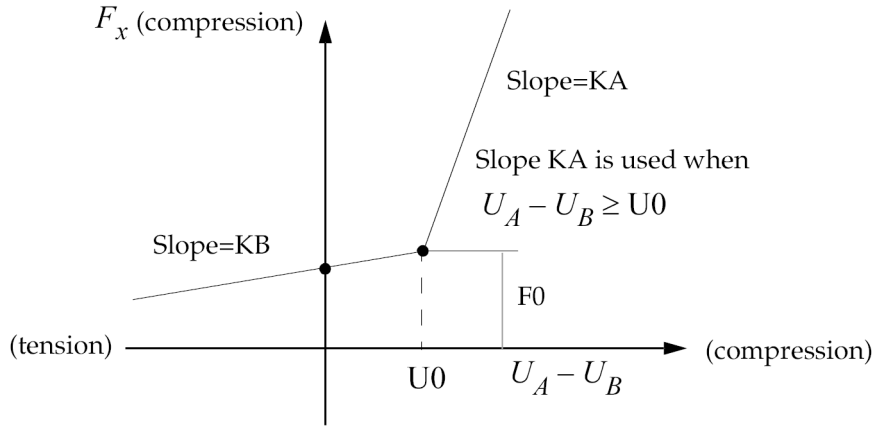


Figure 16-14. CGAP Element Force-Deflection Curve for Nonlinear Analysis

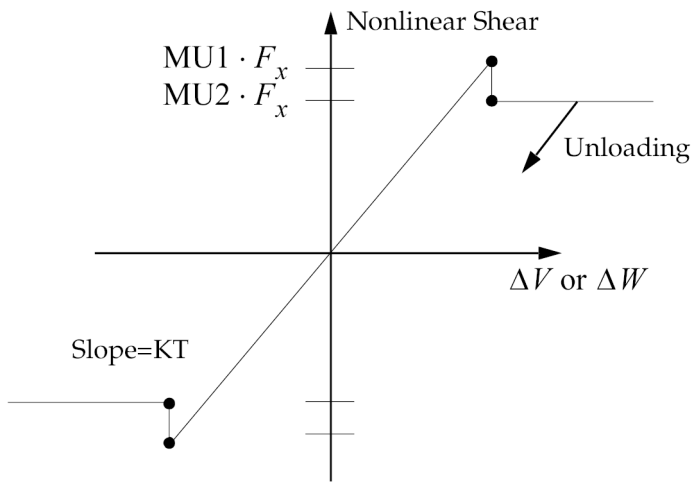


Figure 16-15. Shear Force for CGAP Element

7. If U0 is specified negative and GA and GB are not coincident, then the direction for closing must be controlled by the use of the CID field on the CGAP entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. The gap element is simulated by a nonlinear spring element in SOLs 601 and 701 where the axial properties are represented but the transverse properties are ignored.

2. KT, MU1, MU2, TMAX, MAR, and TRMIN are ignored.

REMARKS RELATED TO CONTACT IN SOL 101 AND CONSECUTIVE SOLUTIONS 103, 111 AND 112:

1. For solution 101 and consecutive solutions 103, 111 and 112 with contact defined (BCSET card exists), CGAP elements are treated as linear contact elements if the system cell (412) OLDGAPS is set to 0 (default). See the chapter on surface contact in the NX Nastran User's Guide.

   For contact in a linear solution, the CGAP element will use U0, MU1, KA, and KT. The following rules apply to the definition of the contact penalty factors.

   - If KA is defined, it is used for the contact penalty factor in the normal direction.

   - If KA is undefined and KB is defined, KB is used for the contact penalty factor in the normal direction.

   - If KA and KB are both undefined, the PENN field on the associated BCTPARM is used for the contact penalty factor in the normal direction. The automatic penalty factor calculation used for surface-to-surface contact does not apply to CGAP elements.

   - If KA, KB, and PENN are all undefined, the default value of PENN is used for the contact penalty factor in the normal direction. PENTYP=2 should not be used in this case.

   - If KT is undefined, the contact penalty factor in the transverse direction is calculated from the PENT and PENTYP fields on the associated BCTPARM.

   - When NASTRAN SYSTEM(464) = 1, KA and KB are ignored, and PENN and PENT defined on the BCTPARM bulk entry are used for the penalty factor in the normal and transverse directions.

2. MU1 is used as the coefficient of friction in both transverse directions.

3. F0, MU2, TMAX, MAR and TRMIN are ignored.

PGPLSN

Generalized Plane Strain Element Property for SOL 401

Defines the properties of generalized plane strain elements for SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PGPLSN	PID	MID	CGID	T					
	KN	KR1	KR2						

EXAMPLE:

PGPLSN	100	1	85	10.0					

FIELDS:

Field	Contents
PID	Property identification number. See Remarks 1 and 2. (Integer > 0)
MID	Identification number of a MAT1 or MAT11 entry. (Integer > 0; No default)
CGID	Identification number of control grid point. (Integer > 0; No default)
T	Undeformed element thickness. (Real > 0.0; No default)
KN	Optional user-specified additive normal stiffness relative to the planar area defined by the mesh of generalized plane strain elements. (Real ≥ 0.0 or Integer > 0; Default = 0.0) If real entry, value of stiffness at all times. If integer entry, identification number of a TABLEDi entry that contains value of stiffness as a function of time.

Field	Contents
KRi	Optional user-specified additive rotational stiffness in the units moment/radian about the ith-axis of the displacement coordinate system for the control grid point. See Remark 3. (Real ≥ 0.0 or Integer > 0; Default = 0.0) If real entry, value of stiffness at all times. If integer entry, identification number of a TABLEDi entry that contains value of stiffness as a function of time.

REMARKS:

1. All PGPLSN property bulk entries must have unique identification numbers with respect to all other property bulk entries.

2. PGPLSN bulk entries can only be referenced by CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 element bulk entries for SOL 401. When these elements reference a PGPLSN bulk entry, their generalized plane strain formulation is used.

3. Any planar area contains a pair of principal axes. The values for the optional user-specified additive rotational stiffness should be defined about axes that pass through the control point and are parallel to the principal axes of the planar area defined by the mesh of generalized plane strain elements. To do this, specify a displacement coordinate system for the control grid point and define two of the axes for the displacement coordinate system as parallel to the principal axes of the planar area as follows:

   - If the axisymmetric plane is the XY-plane of the basic coordinate system, define the X- and Y-axes of the displacement coordinate system as parallel to the principal axes of the planar area.

   - If the axisymmetric plane is the XZ-plane of the basic coordinate system, define the X- and Z-axes of the displacement coordinate system as parallel to the principal axes of the planar area.

4. The generalized plane strain element is not supported in a glue or contact region.

PHBDY

CHBDYP Geometric Element Definition

A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF	D1	D2					

EXAMPLE:

PHBDY	2	.02	1.0	1.0					

FIELDS:

Field	Contents
PID	Property identification number. (Unique Integer among all PHBDY entries). (Integer > 0)
AF	Area factor of the surface used only for CHBDYP element TYPE = “POINT”, TYPE = “LINE”, TYPE = “TUBE”, or TYPE = “ELCYL”. For TYPE = “TUBE”, AF is the constant thickness of the hollow tube. (Real > 0.0 or blank)
D1, D2	Diameters associated with the surface. Used with CHBDYP element TYPE = “ELCYL”, “TUBE”, and “FTUBE”. (Real > 0.0 or blank; Default for D2 = D1)

REMARKS:

1. The PHBDY entry is used with CHBDYP entries.

2. AF

   - For TYPE = “POINT” surfaces, AF is the area.

   - For TYPE = “LINE” or TYPE = “ELCYL” surfaces, AF is the effective width: area = AF · (length).

   - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces

$$\text{area} = \pi \cdot \left(\frac{D1 + D2}{2}\right) \cdot \sqrt{(LGTH)^2 + \left(\frac{D1 - D2}{2}\right)^2}$$

3. D1 and D2 are used only with TYPE = “ELCYL”, TYPE=“TUBE”, and TYPE=“FTUBE” surfaces.

   - For TYPE=“ELCYL” surfaces, D1 and D2 are the two diameters associated with the ellipse.

   - For TYPE=“FTUBE” and outer TYPE=“TUBE” surfaces, D1 and D2 are the diameters associated with the first and second grid points, respectively.

   - For inner TYPE=“TUBE” surfaces, the diameters are reduced by twice the thickness (2 · *AF*).

PLCYISO

Isotropic hardening rule for plastic-cyclic material (SOLs 601 and 701 only)

Specifies the dependence of the radius of the yield surface (stress radius) on accumulated effective plastic strains for the plastic-cyclic material.

FORMAT FOR BILINEAR HARDENING (TYPE = BILIN):

1	2	3	4	5	6	7	8	9	10
PLCYISO	HID	TYPE	YIELD	EP					

FORMAT FOR MULTILINEAR HARDENING (TYPE = MTLIN):

1	2	3	4	5	6	7	8	9	10
PLCYISO	HID	TYPE							
	AEPS1	SR1	AEPS2	SR2	AEPS3	SR3	AEPS4	SR4	
	AEPS5	SR5	-etc.-						

FORMAT FOR EXPONENTIAL HARDENING (TYPE = EXP):

1	2	3	4	5	6	7	8	9	10
PLCYISO	HID	TYPE	YIELD	Q	B				

FORMAT FOR MEMORY EXPONENTIAL HARDENING (TYPE = MEMEXP):

1	2	3	4	5	6	7	8	9	10
PLCYISO	HID	TYPE	YIELD	Q0	QM	MU	B	ETA	

EXAMPLE:

PLCYISO	1	BILIN	1600.0	2.e4					

FIELDS:

Field	Contents
HID	Isotropic hardening identification number referenced by the MATPLCY entry. (Integer > 0)
TYPE	Type of isotropic hardening (Character) BILIN: Bilinear hardening MTLIN: Multilinear hardening EXP: Exponential hardening MEMEXP: Memory exponential hardening
YIELD	Yield stress (radius of yield surface). (Real > 0.0)
EP	Hardening modulus (change in yield stress with respect to accumulated effective plastic strain). (Real ≥ 0.0, Default = 0.0)
AEPSi	Accumulated effective plastic strain at data point i. (Real ≥ 0.0)
SRi	Radius of yield surface corresponding to AEPSi. (Real > 0.0)
Q, B	Parameters giving the change in yield stress with respect to accumulated effective plastic strain, see Advanced Nonlinear Theory and Modeling Guide. (Real; Default = 0.0)
Q0, QM, MU, B	Parameters giving the change in yield stress with respect to accumulated effective plastic strain and strain memory, see Advanced Nonlinear Theory and Modeling Guide. (Real; Default = 0.0)
ETA	Strain memory surface parameter. (Real ≥ 0.0; Default = 0.5)

PLCYKIN

Kinematic hardening rule for plastic-cyclic material (SOLs 601 and 701 only)

Specifies the dependence of the back stresses on the plastic strains for the plastic-cyclic material.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLCYKIN	HID	TYPE							
	H1	ZETA1	H2	ZETA2	-etc-				

EXAMPLE:

PLCYKIN	3	AF							
	1.0e5	2000.0							

FIELDS:

Field	Contents
HID	Kinematic hardening identification number referenced by the MATPLCY entry. (Integer > 0)
TYPE	Type of kinematic hardening. Currently, only TYPE = AF (Armstrong-Fredrick) is allowed.
Hi	Linear kinematic hardening constant at data point i. (Real ≥ 0.0)
ZETAi	Nonlinear kinematic hardening constant at data point i. (Real ≥ 0.0, Default = 0.0)

PLCYRUP

Rupture criterion for plastic-cyclic material (SOLs 601 and 701 only)

Specifies the rupture criterion for the plastic-cyclic material.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLCYRUP	RID	TYPE	RSTR						

EXAMPLE:

PLCYRUP	1	AEPS	.2708e-2						

FIELDS:

Field	Contents
RID	Rupture criterion identification number referenced by the MATPLCY entry. (Integer > 0)
TYPE	Type of rupture criterion. Currently, only TYPE = AEPS (Accumulated effective plastic strain) is allowed.
RSTR	The accumulated effective plastic strain at which the material ruptures. (Real ≥ 0.0)

PLOAD

Static Pressure Load

Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

EXAMPLE:

PLOAD	1	-4.0	16	32	11				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure. (Real)
Gi	Grid point identification numbers. (Integer > 0; G4 may be zero or blank.)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is zero or blank, the surface is triangular.

3. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in Figure 16-16.

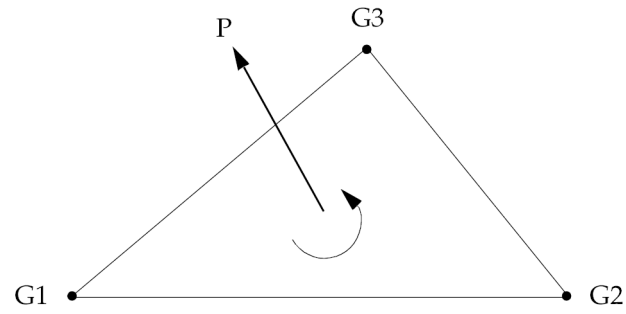


Figure 16-16. Pressure Convention for Triangular Surface of Surface Elements and/or the Faces of Solid Elements

The total load on the surface (see Figure 16-17), AP, is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

4. In the case of a quadrilateral surface, the grid points G1, G2, G3, and G4 should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:

   - The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.

   - One-half of the pressure is applied to each triangle, which is then treated in the manner described in Remark 2 .

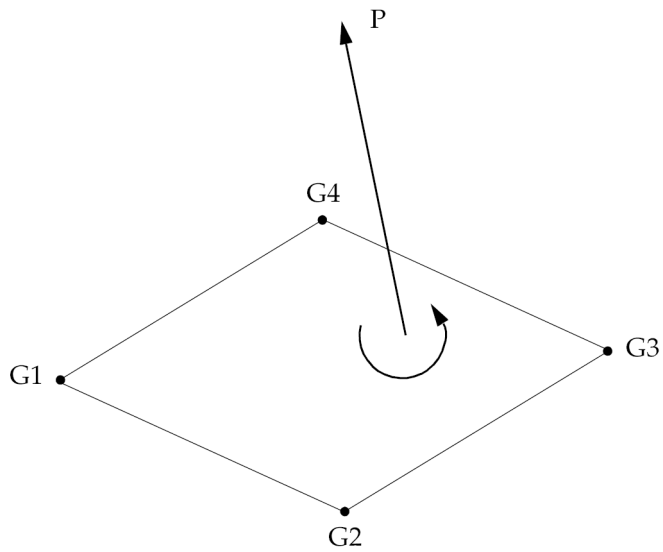


Figure 16-17. Pressure Convention for Quadrilateral Surface of Surface Elements and/or the Faces of Solid Elements

5. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The linear solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "FOLLOWK"). Follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

   For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply a pressure load with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

2. To apply a time-dependent pressure load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

3. In large deformation analysis, the direction of pressure loads follows the deformation of the element by default. The use of LOADOPT = 0 in NXSTRAT entry causes pressure loads to be independent of deformation, i.e., the direction of pressure loads maintains its original direction.

PLOAD1

Applied Load on CBAR, CBEAM or CBEND Elements

Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

EXAMPLE:

PLOAD1	25	1065	MY	FRPR	0.2	2.5E3	0.8	3.5E3	

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number. (Integer > 0)
TYPE	Load type. (Character: “FX”, “FY”, “FZ”, “FXE”, “FYE”, “FZE”, “MX”, “MY”, “MZ”, “MXE”, “MYE”, “MZE”)
SCALE	Determines scale factor for X1, X2. (Character: “LE”, “FR”, “LEPR”, “FRPR”)
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank; 0 ≤ X1 ≤ X2)
P1, P2	Load factors at positions X1, X2. (Real or blank)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. If X2 ≠ X1, a linearly varying distributed load will be applied to the element between positions X1 and X2, having an intensity per unit length of bar equal to P1 at X1 and equal to P2 at X2, except as noted in Remarks 8 and 9.

3. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.

4. If P1 = P2 and X2 ≠ X1, a uniform distributed load of intensity per unit length equal to P1 will be applied between positions X1 and X2 except as noted in Remarks 8 and 9.

5. Load TYPE is used as follows to define loads:

   “FX”, “FY” or “FZ”: Force in the x, y, or z direction of the basic coordinate system.

   “MX”, “MY” or “MZ”: Moment in the x, y, or z direction of the basic coordinate system.

   “FXE”, “FYE” or “FZE”: Force in the x, y, or z direction of the element’s coordinate system.

   “MXE”, “MYE” or “MZE”: Moment in the x, y, or z direction of the element’s coordinate system.

6. If SCALE = “LE” (length), the xi values are actual distances along the element axis, and, if $X2 \neq X1$, then Pi are load intensities per unit length of the element.

7. If SCALE = “FR” (fractional), the xi values are ratios of the distance along the axis to the total length, and (if $X2 \neq X1$) Pi are load intensities per unit length of the element.

8. If SCALE = “LEPR” (length projected), the xi values are actual distances along the element axis, and (if $X2 \neq X1$) the distributed load is input in terms of the projected length of the element.

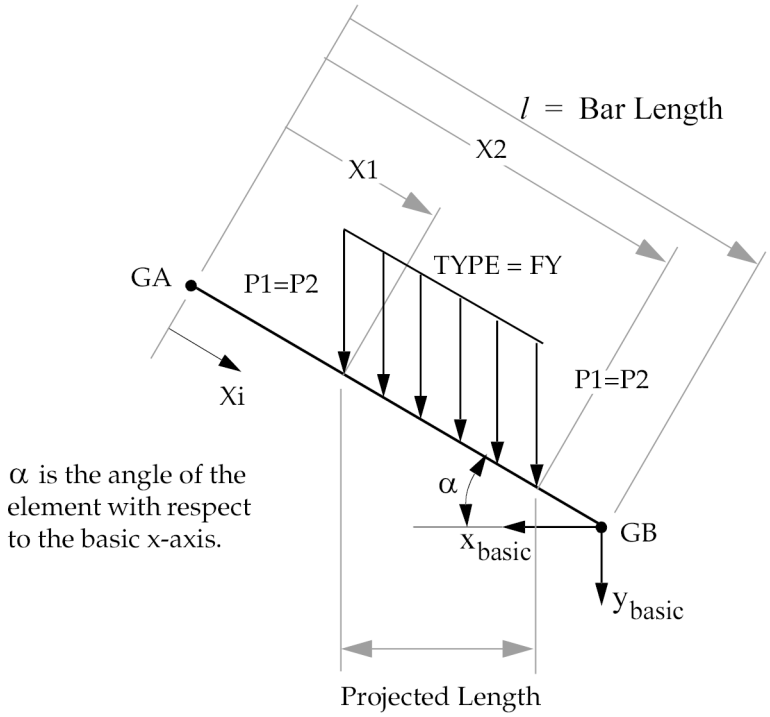


Figure 16-18. PLOAD1 Convention on Beam or Bar Elements

If SCALE = “LE”, the total load applied to the bar is P1 ($X2 - X1$) in the y-basic direction.

If SCALE = “LEPR”, the total load applied to the bar is P1 ($X2 - X1$)cosα in the y-basic direction.

9. If SCALE = “FRPR” (fractional projected), the Xi values are ratios of the actual distance to the length of the bar (CBAR entry), and if $X2 \neq X1$, then the distributed load is specified in terms of the projected length of the bar.

10. Element identification numbers for CBAR, CBEAM, and CBEND entries must be unique.

11. For the CBEND element, the following coordinate equivalences must be made for the element coordinates

$$R_{elem} \equiv X_{elem}$$
$$\theta_{elem} \equiv Y_{elem}$$

12. Only distributed loads applied over the entire length of the CBEND element may be applied.

13. Projected loads are not applicable to the CBEND element.

14. Loads on CBEAM elements defined with PLOAD1 entries are applied along the line of the shear centers.

15. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress locations Xi=0 and Xi = *l* will always be output. This output format will be used for all beam and bar elements.

16. If on the TYPE field of the PLOAD1 entry, the element coordinate system direction (e.g. TYPE = FYE) option is selected, then the projection (i.e. SCALE = FRPR or LEPR) option is ignored and the result is the same as the SCALE = FR (or LE) option.

REMARKS RELATED TO SOLS 601 AND 701:

1. TYPE = FXE, MXE, MYE, or MZE is not supported. Only SCALE = “FR” is supported.

2. TYPE = MX, MY, or MZ is only supported if X2 = X1 or X2 is blank, i.e., as concentrated load.

3. TYPE = FYE or FZE is only supported if X2 is not equal to X1, i.e. as distributed load.

4. If X2 is not equal to X1, a distributed load will be applied over the entire length of the element, i.e., only X1 = 0.0 and X2 = 1.0 is allowed.

5. If X2 = X1 or X2 is blank, a concentrated load of value P1 will be applied at the X1 location and only X1=0.0 or X1=1.0 is allowed.

6. To apply a load with constant magnitude (with respect to time), SID is selected by Case Control command LOAD = SID for both static and transient analyses.

7. To apply a time-dependent load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

8. In large deformation analysis, the direction of distributed loads (TYPE = FYE or FZE only) follows the deformation of the element by default. The use of LOADOPT = 0 in NXSTRAT entry causes distributed loads to be independent of deformation, i.e., the direction of distributed loads maintains its original direction.

PLOAD2

Uniform Normal Pressure Load on a Surface Element

Defines a uniform static pressure load applied to CQUAD4, CQUADR, CSHEAR, CTRIA3, or CTRIAR two-dimensional elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

EXAMPLE:

PLOAD2	21	-3.6		4	16		2		

ALTERNATE FORMAT AND EXAMPLE:

PLOAD2	SID	P	EID1	“THRU”	EID2				
PLOAD2	1	30.4	16	THRU	48				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
EIDi	Element identification number. (Integer ≥ 0 or blank; for the “THRU” option, EID1 < EID2.)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. At least one positive EID must be present on each PLOAD2 entry.

3. If the alternate form is used, all elements EID1 through EID2 must be two-dimensional.

4. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry. Refer to the PLOAD entry.

5. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.

6. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "FOLLOWK"). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply a pressure load with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses.

2. To apply a time-dependent pressure load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

3. In large deformation analysis, the direction of pressure loads follows the deformation of the element by default. The use of LOADOPT = 0 in NXSTRAT entry causes pressure loads to be independent of deformation, i.e., the direction of pressure loads maintains its original direction.

PLOAD4

Pressure Load on Shell and Solid Element Faces

Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CPYRAM, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3					

EXAMPLE:

PLOAD4	2	1106	10.0	8.0	5.0		48		
	6	0.0	1.0	0.0					

ALTERNATE FORMAT AND EXAMPLE

(See Remark 8):

PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3					

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)

Field	Contents
EID EID1 EID2	Element identification number. (Integer > 0; for the “THRU” option, EID1 < EID2)
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1.)
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)
G3	For CHEXA, CPYRAM, or CPENTA quadrilateral faces, G3 is the identification number of a grid point connected to a corner diagonally opposite to G1. Required for quadrilateral faces of CHEXA, CPYRAM and CPENTA elements only. For CPYRAM element triangle faces, G1 and G3 are adjacent corner nodes on the quadrilateral face, and the load is applied on the triangular face which includes those grids. For CPENTA element triangle faces, G3 must be omitted.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)
CID	Coordinate system identification number. See Remark 2. (Integer ≥ 0; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark 2. (Real)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over

the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.

3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.

4. For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)

5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.

6. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.

7. G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.

8. The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.

9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.

10. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.

11. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The linear solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter “FOLLOWK”). Follower force effects are included in

the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply a pressure load with constant magnitude (with respect to time), SID is selected by Case Control command LOAD = SID for both static and transient analyses.

2. To apply a time-dependent pressure load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

3. In large deformation analysis, the direction of normal pressure loads (i.e., CID, N1, N2 and N3 not specified) follows the deformation of the element by default. The use of LOADOPT = 0 in NXSTRAT entry causes pressure loads to be independent of deformation, i.e., the direction of pressure loads maintains its original direction.

4. CID, if specified, must be a rectangular coordinate system. Otherwise an error message will be issued.

PLOADE1

Edge Load on Plane Strain and Plane Stress Elements

Defines a surface traction that acts along the edge of CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, or CPLSTS8 elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOADE1	SID	EID	PA	PB	GA	GB	THETA		

EXAMPLE:

PLOADE1	200	35	3.5	10.5	10	30	20.		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA in force per unit area. (Real)
PB	Surface traction at grid point GB in force per unit area. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between surface traction and inward normal to the line segment in degrees. (Real; Default = 0.0)

REMARKS:

1. In static solution sequences, the load set ID (SID) is selected by the LOAD case control command. In dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the LOADSET case control command.

2. The surface traction varies linearly along the element edge between GA and GB.

3. When THETA=0.0, the load is normal to the line segment connecting GA and GB. When Pi is defined positive, relative to THETA=0.0, the load is directed into the element. When the elements are defined on the XZ-plane, a positive THETA rotates the load in the negative Y-direction using the right-hand-rule. When the elements are defined on the XY-plane, a positive THETA rotates the load in the negative Z-direction using the right-hand-rule. See Figures 16-19 and 16-20.

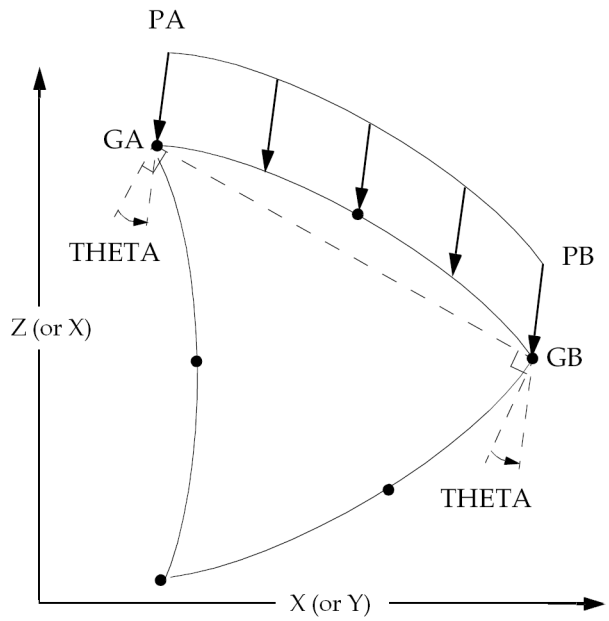


Figure 16-19. Pressure Load Example

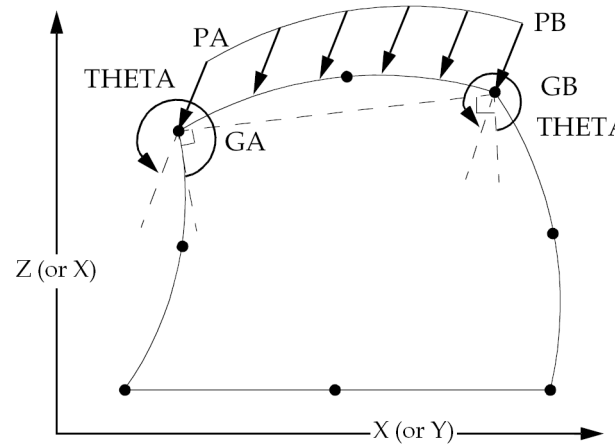


Figure 16-20. Pressure Load on CPLSTN4, CPLSTN8, CPLSTS4, and CPLSTS8 Elements

4. If THETA = 0.0 and the edge of the element is a straight line, the surface traction is simply an applied pressure loading.

REMARKS RELATED TO SOL 601:

1. To define a surface traction as time-independent, use LOAD = SID in the case control.

2. To define a surface traction as time-dependent, reference the SID in the EXCITEID field of a TLOAD1 entry and include a DLOAD case control command that references the TLOAD1 entry.

3. In large deformation analysis, the direction of the surface traction follows the deformation of the element by default. The use of LOADOPT = 0 in an NXSTRAT entry causes the surface tractions to act in their original direction throughout the analysis.

PLOADX1

Pressure Load on Axisymmetric and Chocking Elements

Defines a surface traction that acts along the edge of CTRAX3, CTRAX6, CQUADX4, or CQUADX8 axisymmetric elements, and CCHOCKi chocking elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOADX1	SID	EID	PA	PB	GA	GB	THETA		

EXAMPLE:

PLOADX1	200	35	3.5	10.5	10	30	20.		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA in force per unit area. (Real)
PB	Surface traction at grid point GB in force per unit area. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between the surface traction and the inward normal to the line segment in degrees. (Real; Default = 0.0)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. The surface traction is assumed to vary linearly along the element side between GA and GB.

3. For chocking elements, the software uses the circumference minus the product of the gap size and number of gaps when it calculates the forces that act at the grid points that lie on the edge defined by GA and GB.

4. When THETA=0.0, the load is normal to the line segment connecting GA and GB. When Pi is defined positive, relative to THETA=0.0, the load is directed into the element. When the elements are defined on the XZ-plane, a positive THETA rotates the load in the negative Y-direction using the right-hand-rule. When the elements are defined on the XY-plane, a positive THETA rotates the load in the negative Z-direction using the right-hand-rule. See Figures 16-21 and 16-22.

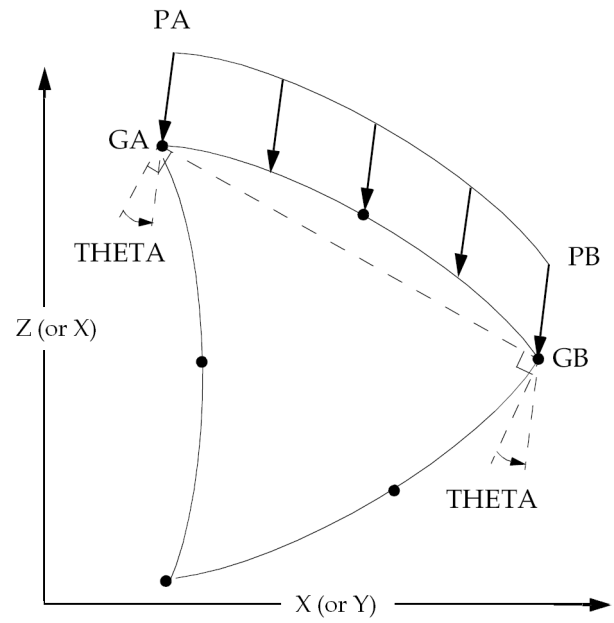


Figure 16-21. Pressure Load on CTRAX3 and CTRAX6 Elements

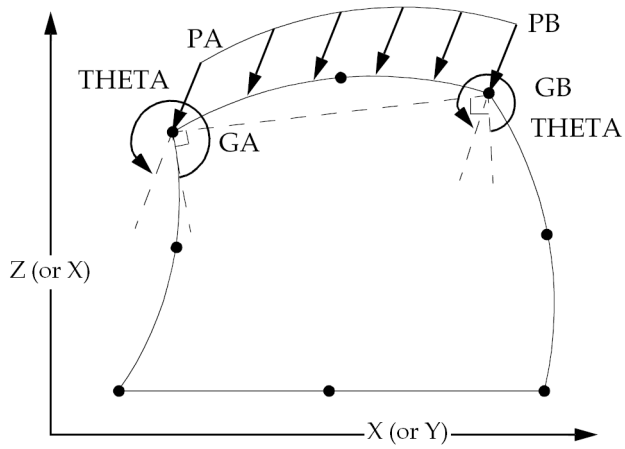


Figure 16-22. Pressure Load on CQUADX4 and CQUADX8 Elements

REMARKS RELATED TO SOL 601:

1. To apply a pressure load with constant magnitude (with respect to time), SID is selected by Case Control command LOAD = SID for both static and transient analysis.

2. To apply a time-dependent pressure load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

3. In large deformation analysis, the direction of pressure loads follows the deformation of the element by default. The use of LOADOPT = 0 in NXSTRAT entry causes pressure loads to be independent of deformation, i.e., the direction of pressure loads maintains its original direction.

PLOTEL

Dummy Plot Element Definition

Defines a one-dimensional dummy element for use in plotting.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLOTEL	EID	G1	G2						

EXAMPLE:

PLOTEL	29	35	16						

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

REMARKS:

1. This element is not used in the model during any of the solution phases of a problem. It is used to simplify plotting of structures with large numbers of colinear grid points, where the plotting of each grid point along with the elements connecting them would result in a confusing plot.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one PLOTEL element may be defined on a single entry.

4. In superelement analysis, PLOTELs, as well as other elements such as CBAR, CQUAD4, etc., will affect the formation of the superelement tree. The PLOTEL EIDs will also appear in the superelement map output; see the description of PARAM,SEMAPPRT in “Parameters and Parameter Applicability Tables”.

5. Only grid points connected by structural elements appear on structure plots. This does not include points connected only by rigid or general elements or MPCs. A plot element in parallel with elements that do not plot will cause these points to be present.

PLPLANE

Fully Nonlinear Plane Element Properties

Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLPLANE	PID	MID	CID	STR	T				

EXAMPLE:

PLPLANE	203	204	201		0.11				

FIELDS:

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP or MATHE entry. See Remark 4. (Integer > 0)
CID	Identification number of a coordinate system defining the plane of deformation. See Remark 3. (Integer ≥ 0; Default = 0)
STR	Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”)
T	Thickness for plane stress elements (SOL 601 only). If T is blank or zero then the thickness must be specified for Ti on the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 entries. T is ignored for plane strain elements. (Real ≥ 0.0 or blank)

REMARKS:

1. All PLPLANE property entries should have unique identification numbers with respect to all other property entries.

2. PLPLANE can be referenced by CQUAD, CQUAD4, CQUAD8, CTRIA3, and CTRIA6 entries in solutions 106 and 129. The CQUAD4, CQUAD8, CTRIA3, and CTRIA6 entries are treated as hyperelastic plane strain elements. See SOL 601 Remark 2.

3. The plane strain hyperelastic elements CQUAD, CQUAD4, CQUAD8, CTRIA3, CTRIA6 must lie on the x-y plane of the basic coordinate system. Stress and strain is output in the CID coordinate system for these elements. See SOL 601 Remarks 1 and 2.

4. For SOL 106, MID must refer to a MATHP entry.

REMARKS RELATED TO SOL 601:

1. CID is not supported. Stresses and strains are output in the basic coordinate system.

2. PLPLANE can be referenced by CQUAD, CQUAD4, CQUAD8, CTRIA3, CTRIA6, CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 entries in solution 601. The CQUAD4, CQUAD8, CTRIA3, and CTRIA6 entries are treated as hyperelastic plane strain elements.

3. The plane stress and plane strain hyperelastic elements CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8 must lie on the x-z plane of the basic coordinate system.

4. The plane strain hyperelastic elements CQUAD, CQUAD4, CQUAD8, CTRIA3, CTRIA6 must lie on the x-y plane of the basic coordinate system.

5. STR is ignored. Stresses and strains are output at grid points.

6. In a model where there are many plane stress elements with different thicknesses, it is recommended that the thickness values be specified in the element entries (Ti value in CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8) instead of assigning one PLPLANE entry to one thickness value and resulting in many PLPLANE entries.

PLSOLID

Fully Nonlinear Solid Element Properties

Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	STR						

EXAMPLE:

PLSOLID	20	21							

FIELDS:

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP or MATHE entry. See Remark 3. (Integer > 0)
STR	Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”)

REMARKS:

1. PLSOLID can be referenced by a CHEXA, CPENTA, CPYRAM (SOL 601 only), CTETRA, CTRAX3, CTRAX6, CQUADX4, or CQUADX8 entry.
2. Stress and strain are output in the basic coordinate system.
3. For SOL 106, MID must refer to a MATHP entry.

REMARKS RELATED TO SOL 601:

1. STR is ignored. Stresses and strains are output at grid points.

PMASS

Scalar Mass Property

Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

EXAMPLE:

PMASS	7	4.29	6	13.2					

FIELDS:

Field	Contents
PIDi	Property identification number. (Integer > 0)
Mi	Value of scalar mass. (Real)

REMARKS:

1. Mass values are defined directly on the CMASS2 and CMASS4 entries, and therefore do not require a PMASS entry.
2. Up to four mass values may be defined by this entry.
3. For a discussion of scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

PMIC

Dummy Property for Microphone Elements

Defines dummy property for microphone elements for acoustic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PMIC	ID								

EXAMPLE:

PMIC	1								

FIELDS:

Field	Contents
ID	Microphone element property identification number. (Integer > 0) See Remark 1.

REMARKS:

1. ID is referenced by the PID of elements to designate them as microphone elements. The allowable element types are CROD, CTRIA3, CQUAD4, CHEXA, CPENTA, CTETRA, and CPYRAM. PMIC entries must have unique identification numbers with respect to all other property entries.

POINT

Point for SELOC Entry

Define a point for use with the SELOC entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
POINT	ID	CP	X1	X2	X3				

EXAMPLE:

POINT	12	1	1.	2.	5.				

FIELDS:

Field	Contents	Type	Default
ID	Point identification number.	Integer ≥ 0	Required
CP	Identification number of coordinate system in which the location of point is defined.	Integer ≥ 0	0
X1, X2, X3	Location of the point in coordinate system CP.	Real	0.0

REMARKS:

1. POINT is used to specify geometric points for use with the SELOC bulk entry.

   There are no degrees-of-freedom assigned to a point.

2. Only a SELOC entry can refer to POINT entries.

3. ID of POINTs must be unique with respect to ID of GRID entries.

PPLANE

Properties of Plane Stress or Plane Strain Elements

Defines the properties of plane stress elements or plane strain elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PPLANE	PID	MID	T	NSM	FOROPT				

EXAMPLES:

PPLANE	203	204	0.11						

PPLANE	1	3	0.5	0.0	1				

FIELDS:

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Material identification number. (Integer >0)
T	Thickness for plane stress elements. If T is blank or zero then the thickness must be specified for Ti on the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 entries. T is ignored for plane strain elements. (Real ≥ 0.0 or blank)
NSM	Nonstructural mass per unit area. (Real)
FOROPT	Formulation option number. See Remark 6. (Integer; Default = 0)

REMARKS:

1. All PPLANE property entries should have unique identification numbers with respect to all other property entries.

2. The entry is referenced by the CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 entries via PID.

3. For structural problems, MID must reference a MAT1, MAT3, or MAT11 material property entry.

4. For heat transfer problems, MID must reference a MAT4 or MAT5 material property entry.

5. In a model where there are many plane stress elements with different thicknesses, it is recommended that the thickness values be specified in the element entries (Ti value in CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8) instead of assigning one PPLANE entry to one thickness value and resulting in many PPLANE entries.

6. The default value of 0 uses the standard formulation. A value of 1 specifies the Mean Dilatational Formulation option for nearly incompressible materials.

7. The fields on the PPLANE cannot be associated to design variables in SOL 200.

REMARKS RELATED TO SOL 601:

1. NSM and FOROPT are ignored.

2. For structural problems, MID must reference a MAT1, MAT3, MAT10, MATSMA, MATVE or MATPLCY material property entry.

PRAC2D

CRAC2D Element Property

Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PRAC2D	PID	MID	T	IPLANE	NSM	GAMMA	PHI		

EXAMPLE:

PRAC2D	108	2	0.10	0	.17	.50	180.		

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
T	Element thickness. (Real > 0.0)
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress. (Integer = 0 or 1)
NSM	Nonstructural mass per unit area. (Real ≥ 0.0; Default = 0)
GAMMA	Exponent used in the displacement field. See Remark 4 . (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated. See Remark 4 . (Real; Default = 180.0)

REMARKS:

1. All PRAC2D property entries should have unique identification numbers with respect to all other property entries.

2. PRAC2D entry may refer to MAT1, MAT2, or MAT8 material property entries.

3. For plane strain analysis, only MAT1 type data should be used.

4. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PRAC3D

CRAC3D Element Property

Defines the properties of the CRAC3D structural element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PRAC3D	PID	MID	GAMMA	PHI					

EXAMPLE:

PRAC3D	108	2	.50	180.					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GAMMA	Exponent used in the displacement field. See Remark 3 . (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x axis along which stress intensity factors are to be calculated. See Remark 3 . (Real; Default = 180.0)

REMARKS:

1. All PRAC3D property entries should have unique identification numbers with respect to all other property entries.
2. Either isotropic (MAT1) or anisotropic (MAT9) material entries may be referenced.

3. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PROD

Rod Property

Defines the properties of a rod element (CROD entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

EXAMPLE:

PROD	17	23	42.6	17.92	4.2356	0.5			

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2 and 3. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)

REMARKS:

1. PROD entries must all have unique property identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.

3. For heat transfer problems, MID must reference a reference MAT4 or MAT5 entry.

4. The formula used to calculate torsional stress is

$$\tau = \frac{CM_\theta}{J}$$

where M_θ is the torsional moment.

REMARKS RELATED TO SOLS 601 AND 701:

1. J, C, and NSM are ignored.

2. CROD defines a truss element with no torsional stiffness, i.e., only axial force is transmitted by the element.

3. For structural problems, MID must reference a MAT1, MATSMA, MATVE or MATPLCY material entry for SOL 601, and a MAT1 or MATPLCY material entry for SOL 701.

4. For heat transfer problems, MID must reference a MAT4 entry.

PSHEAR

Shear Panel Property

Defines the properties of a shear panel (CSHEAR entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

EXAMPLE:

PSHEAR	17	23	42.6	17.92	4.236	0.5			

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT1 entry. (Integer > 0)
T	Thickness of shear panel. (Real ≠ 0.0)
NSM	Nonstructural mass per unit area. (Real)
F1	Effectiveness factor for extensional stiffness along edges 1-2 and 3-4. See Remark 2 (Real ≥ 0.0; Default = 0.0)
F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4. See Remark 2 (Real ≥ 0.0; Default = 0.0)

REMARKS:

1. All PSHEAR entries should have unique identification numbers with respect to all other property entries.

2. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If F1 ≤ 1.01, the areas of the rods on edges 1-2 and 3-4 are set equal to (F1 · T · PA)/(L12 + L34) where PA is the panel surface area-half the vector cross product area of the diagonals-and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if F1 = 1.0, the panel is fully effective for extension in the 1-2 direction. If F1 > 1.01, the areas of the rods on edges 1-2 and 3-4 are each set equal to $0.5 \cdot F1 \cdot T^2$.

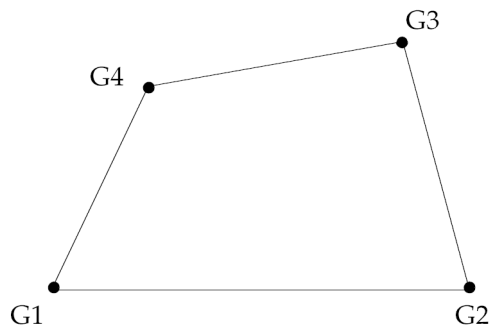


Figure 16-23. Extensional Area for Shear Panel

Thus, if F1 = 30, the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to 15T. The significance of F2 for edges 2-3 and 1-4 is similar.

3. Poisson's ratio coupling for extensional effects is ignored.

PSHELL

Shell Element Property

Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T**3	MID3	TS/T	NSM	
	Z1	Z2	MID4						

EXAMPLE:

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	
	+.95	-.95							

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID1	Material identification number for the membrane. (Integer ≥ 0 or blank)
T	Default membrane thickness for Ti on the connection entry. If T is blank then the thickness must be specified for Ti on the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 entries. (Real or blank)
MID2	Material identification number for bending. (Integer ≥ -1 or blank)
12I/T**3	Bending moment of inertia ratio, $12I/T^3$. Ratio of the actual bending moment inertia of the shell, I, to the bending moment of inertia of a homogeneous shell, $T^3/12$. The default value is for a homogeneous shell. (Real > 0.0; Default = 1.0)

Field	Contents
MID3	Material identification number for transverse shear. (Integer > 0 or blank; unless MID2 > 0, must be blank.)
TS/T	Transverse shear thickness ratio, T_s/T. Ratio of the shear thickness, (T_s), to the membrane thickness of the shell, T. The default value is for a homogeneous shell. (Real > 0.0; Default = .833333)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. See Remark 11 for defaults. (Real or blank)
MID4	Material identification number for membrane-bending coupling. See Remark 6. (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2.)

REMARKS:

1. All PSHELL property entries should have unique identification numbers with respect to all other property entries.

2. The structural mass is calculated from the density using the membrane thickness and membrane material properties.

3. The results of leaving an MID field blank (or MID2 = -1) are:

MID1	No membrane or coupling stiffness.
MID2	No bending, coupling, or transverse shear stiffness.
MID3	No transverse shear flexibility.
MID4	No membrane-bending coupling unless ZOFFS is specified on the connection entry. See Remark 6.
MID2=-1	See Remark 12. Note: MID1 and MID2 must be specified if the ZOFFS field is also specified on the connection entry.

4. The continuation entry is not required.

5. The structural damping (GE on the MATi entry) is obtained from MID1 material.

6. The following should be considered when using MID4.

   - The MID4 field should be left blank if the material properties are symmetric with respect to the middle surface of the shell. If the element centerline is offset from the plane of the grid points but the material properties are symmetric, the preferred method for modeling the offset is by use of the ZOFFS field on the connection entry. Although the MID4 field may be used for this purpose, it may produce ill-conditioned stiffness matrices (negative terms on factor diagonal) if done incorrectly.

   - Only one of the options MID4 or ZOFFS should be used; if both methods are specified the effects are cumulative. Since this is probably not what the user intended, unexpected answers will result. Note that the mass properties are not modified to reflect the existence of the offset when the ZOFFS and MID4 methods are used. If the weight or mass properties of an offset plate are to be used in an analysis, the RBAR method must be used to represent the offset. See “Plate and Shell Elements” in the *NX Nastran Element Library*.

   - The effects of MID4 are not considered in the calculation of differential stiffness. Therefore, it is recommended that MID4 be left blank in buckling analysis.

7. This entry is referenced by the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries via PID.

8. For structural problems, MIDi must reference a MAT1, MAT2, or MAT8 material property entry.

9. If the transverse shear material MID3 references a MAT2 entry, then G33 must be zero. If MID3 references a MAT8 entry, then G1Z and G2Z must not be zero.

10. For heat transfer problems, MIDi must reference a MAT4 or MAT5 material property entry.

11. The default for Z1 is -T/2, and for Z2 is +T/2. T is the local plate thickness defined either by T on this entry or by membrane thicknesses at connected grid points, if they are input on connection entries.

12. For plane strain analysis, set MID2=-1 and set MID1 to reference a MAT1 entry. In-plane loads applied to plane strain elements are interpreted as line-loads with a value equal to the load divided by the thickness. Thus, if a thickness of “1.0” is used, the value of the line-load equals the load value. Pressure can be approximated with multiple line loads where the pressure value equals the line-load divided by the length between the loads.

13. A PSHELL entry with a MID1 or MID2 greater than or equal to 10^8 requires the parameter NOCOMPS to be 0 or -1 for any stress or strain recovery. See the parameter NOCOMPS for more information.

14. For a material nonlinear property, MID1 must reference a MATS1 entry and be the same as MID2, unless a plane strain (MID2 = -1) formulation is desired. Also, MID3 cannot reference a MATS1 entry.

15. If transverse shear flexibility is specified for a model with curved shells where the loading is dominated by twist, results will not converge and may be inaccurate. PARAM,SNORM should be set for this unique model condition.

REMARKS RELATED TO SOLS 601 AND 701:

1. 12I/T**3, MID3, TS/T, NSM, Z1, Z2, and MID4 are ignored.

2. MID1 must be specified and must be greater than 0. MID1 is used to evaluate all material properties of the element and must reference a MAT1, MAT2, MAT8, MATSMA, MATVE or MATPLCY material property entry.

3. The shell element includes membrane, bending and transverse shear effects. Membrane and bending effects are always coupled.

4. For SOL 601, MID2 is used only to check for the condition MID2=-1 for plane strain analysis. Plane strain elements must lie in the X-Y plane. For SOL 701, plane strain is not supported.

   If THETA is defined on the element definitions (CQUADi and CTRIi entries) and MID2=-1, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity defined on the element definitions and using the right-hand-rule, must be consistent with the positive z-direction of the basic system.

5. By default, incompatible modes are used for 4-node shell elements in SOL 601 but not in SOL 701. The use of incompatible modes for 4-node shell elements can be set by the ICMODE parameter in NXSTRAT entry. Note that incompatible modes are not used for 4-node plane strain (i.e., MID2=-1) quadrilateral elements.

6. In a model where there are many shell elements with different thicknesses, it is recommended that the thickness values be specified in the element entries (Ti values in CQUAD4, CQUAD8, CTRIA3, CTRIA6) instead of assigning one PSHELL entry to one thickness value and resulting in many PSHELL entries.

PSHL3D

3D-Shell Element Property (SOLs 601 and 701 only)

Defines the properties of 3D-shell elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PSHL3D	PID	MID	T						

EXAMPLE:

PSHL3D	2	5	0.05						

FIELDS:

Field	Contents
PID	Property identification number referenced by a CQUAD4 or CTRIA3 entry. (Integer > 0)
MID	Identification number of a MAT1, MATPLCY, MATHP or MATHE entry. (Integer > 0)
T	Default shell thickness. If T is blank, then the thickness must be specified for Ti on the CQUAD4 and CTRIA3 entries. (Real > 0.0 or blank)

REMARKS:

1. The 3D-shell element can only be used with the 3-node (CTRIA3) or 4-node (CQUAD4) element.
2. By default, the mixed displacement pressure (u/p) formulation is not used for the elastic material but is used for the plastic-cyclic, Ogden, Mooney-Rivlin,

Arruda-Boyce and Sussman-Bathe materials. The u/p formulation cannot be used for the hyperfoam material.

PSOLCZ

Properties of Cohesive Elements

Defines the properties of cohesive elements (CHEXCZ and CPENTCZ entries).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PSOLCZ	PID	MID	CORDM	THICK					

EXAMPLE:

PSOLCZ	1	4	3	0.75					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT11, or MATCZ entry. See Remark 2. (Integer > 0; No default)
CORDM	Identification number of the material coordinate system. Enter “0” or leave blank to use the basic coordinate system. See Remarks 3 and 4. (Integer; Default = 0)
THICK	Thickness of cohesive element. See Remark 2. (Real > 0.0; Default = 1.0)

REMARKS:

1. PSOLCZ entries should have unique identification numbers with respect to all other property entries.

2. Three stiffness values are associated with cohesive elements: K01 and K02 are transverse shear stiffness values; K03S is the out-of-plane normal stiffness value.

   The physical thickness of a cohesive element as defined by the geometric coordinates of the grids is not used to determine the stiffness of the cohesive element.

   If the PSOLCZ entry references a MAT1 entry, the software calculates the stiffness of the cohesive elements to be K01 = G / THICK, K02 = G / THICK, and K03S = E / THICK.

   If the PSOLCZ entry references a MAT11 entry, the software calculates the stiffness of the cohesive elements to be K01 = G13 / THICK, K02 = G23 / THICK, and K03S = E3 / THICK.

   If the PSOLCZ entry references a MATCZ entry, you specify the stiffness of the cohesive element directly in the K01, K02, and K03S fields of the MATCZ entry and the THICK field is ignored.

   The unit of stiffness for cohesive elements is force per unit length cubed.

3. The Z-axis of the material coordinate system for a cohesive element is normal to the plane defined by grids G1-G2-G3-G4 for a CHEXCZ element or the plane defined by grids G1-G2-G3 for a CPENTCZ element. The positive sense of the Z-axis is the same direction as the vector obtained by crossing a vector directed from G1 to G2 with a vector directed from G1 to G4 for the CHEXCZ element or a vector directed from G1 to G3 for the CPENTCZ element.

   The X-axis of the material coordinate system for a cohesive element depends on whether a MATCID entry references the element.

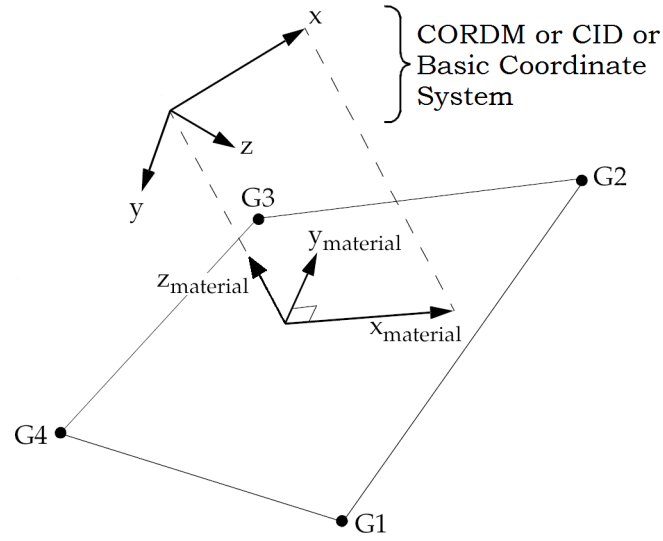
   - If a MATCID entry references the element, the X-axis of the material coordinate system is the projection of the X-axis of the coordinate system listed in the CID field of the MATCID entry onto the plane defined by grids G1-G2-G3-G4 for a CHEXCZ element or the plane defined by grids G1-G2-G3 for a CPENTCZ element.

   - If a MATCID entry does not reference the element, the X-axis of the material coordinate system depends on how the CORDM field is specified.

     - If the CORDM field is specified, but not set to "0", the X-axis of the material coordinate system is the projection of the X-axis of the CORDM coordinate system onto the plane defined by grids G1-G2-G3-G4 for a CHEXCZ element or the plane defined by grids G1-G2-G3 for a CPENTCZ element.

o If the CORDM field is blank or is set to "0", the X-axis of the material coordinate system is the projection of the X-axis of the basic coordinate system onto the plane defined by grids G1-G2-G3-G4 for a CHEXCZ element or the plane defined by grids G1-G2-G3 for a CPENTCZ element.

The Y-axis of the material coordinate system lies in the plane defined by grids G1-G2-G3-G4 for a CHEXCZ element or the plane defined by grids G1-G2-G3 for a CPENTCZ element, and together with the X-axis and Z-axis forms a right-handed coordinate system.



The software cannot project the X-axis of the CORDM, CID, or basic coordinate system onto the plane of a cohesive element if it is parallel to the Z-axis of the material coordinate system for the cohesive element. If they are parallel, a fatal error results.

4. If MID references a MAT11 or MATCZ entry, then CORDM defines the principal material coordinates.

5. Use the CZRESULTS case control command to request results for CHEXCZ and CPENTCZ elements. Results are output for corner grid points only.

6. A cohesive element can use temperature-dependent materials MAT1 and MATT1, or MAT11 and MATT11. However, logarithmic interpolation is not supported if a TABLEM1 is referenced. The software does not issue a warning message and reverts to linear interpolation if a logarithmic AXIS is specified.

PSOLID

Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, CPYRAM, CQUADX4, CQUADX8, CTETRA, CTRAX3, and CTRAX6 entries).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

EXAMPLE:

PSOLID	2	100	6	TWO	GRID	REDUCED			

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT3, MAT4, MAT5, MAT9, MAT10, MAT11, or MATPOR entry. See Remarks 2 and 4. (Integer > 0)
CORDM	Identification number of the material coordinate system. See Remarks 5, 6, and 8. (Integer; Default = 0, which is the basic coordinate system)
IN	Controls the element integration order and options for bubble function degrees-of-freedom. The control varies by element and solution type. See Remarks 9, 10, 11, and 12. (Integer, Character, or blank)
STRESS	Location selection for stress and strain output. See Remarks 8, 9, and 13. (Integer, Character, or blank)

Field	Contents
ISOP	Controls the option for reduced shear integration. The control varies by element and solution type. See Remarks 9, 10, 11, and 12. (Integer, Character, or blank)
FCTN	Fluid element flag. (Character: “PFLUID” indicates a fluid element, “SMECH” indicates a structural element; Default = “SMECH.”)

REMARKS:

1. PSOLID entries should have unique identification numbers with respect to all other property entries.

2. Isotropic (MAT1 or MAT4), axisymmetric solid orthotropic (MAT3), anisotropic (MAT5, MAT9), fluid (MAT10), and 3-D solid orthotropic (MAT11) material properties may be referenced. If FCTN=“PFLUID”, then MID must reference a MAT10 entry.

3. In solutions 103, 107–112, and 200, if an axisymmetric element references a PSOLID entry, the PSOLID entry cannot reference a MAT10 entry. This is supported by SOL 601.

4. For material nonlinear analysis using SOLs 106 or 129, the MATS1 entry can be used with the solid elements CHEXA, CPENTA, CPYRAM, and CTETRA only.

5. See the CHEXA, CPENTA, CPYRAM or CTETRA entry for the definition of the element coordinate system. The material coordinate system (CORDM) may be the basic system (0 or blank), any defined system (Integer > 0), or the element coordinate system (-1). The default value for CORDM is zero unless it is overridden by the NASTRAN statement with the CORDM keyword. See The nastran Command.

6. If MID references a MAT9 or MAT11 entry, then CORDM defines the material property coordinate system for Cij.

7. The initial element coordinate system for the CQUADX4, CQUADX8, CTRAX3, and CTRAX6 elements is the basic coordinate system. The material coordinate system on these elements is defined by “TH” on the element definition, and not by CORDM.

8. Components of stress are output in the material coordinate system except:

- Stress output labeled "NONLINEAR STRESSES", which is output in the element coordinate system. For SOL 106 geometric nonlinear analysis, the nonlinear stresses are output in deformed element coordinate system.

- Hyperelastic element stress, which is always output in the basic coordinate system, including when in the stress output labeled "NONLINEAR STRESSES".

- For SOL 401, stress and strain are output in the basic coordinate system.

See SOL 601/701 Remarks 1 and 4 for Advanced Nonlinear exceptions.

9. The tables following the PSOLID remarks indicate the allowed options and combination of options. If a combination not found in the table is used, then a warning message will be issued and default values will be assigned for all options.

10. For CHEXA and CPENTA elements with no midside nodes, reduced shear integration with bubble functions (ISOP = blank or “REDUCED” and IN = blank or “BUBBLE”) is the default. This is recommended because it minimizes shear locking and Poisson’s ratio locking and does not cause modes of deformation that lead to no strain energy. The effects of using non-default values are as follows:

    a. IN = “THREE” or 3 produces an overly stiff element.

    b. If IN = “TWO” or 2 and the element has midside nodes, modes of deformation may occur that lead to no strain energy.

    c. Standard isoparametric integration (ISOP = “FULL” or 1 and IN = “TWO” or 2; or “THREE” or 3) produces an element overly stiff in shear. This type of integration is more suited to nonstructural problems.

11. IN = “BUBBLE” is not allowed for CTETRA elements or for CHEXA and CPENTA elements with midside nodes.

12. For CTETRA and fluid elements (FCTN = “PFLUID”), standard isoparametric integration (ISOP = “FULL” or 1 and IN = “TWO” or 2; or “THREE” or 3) is the default and the only option available.

13. For linear solutions: Stress and strain output may be requested at the Gauss points (STRESS = “GAUSS” or 1) on the CHEXA and CPENTA elements with no midside grids, on the CTETRA and CPYRAM elements with or without midside grids, and on the CTRAX3, CTRAX6, CQUADX4, CQUADX8 elements.

    For nonlinear solutions 106 and 129: Stress and strain output may be requested at the Gauss points on the CHEXA, CPENTA, CPYRAM and CTETRA elements with or without midside grids.

For SOL 401, stress and strain can be requested at the Gauss points on CHEXA, CPENTA, CPYRAM and CTETRA elements with or without midside grids using the GSTRESS, GSTRAIN, GTHSTRN, and GELSTRN case control commands.

14. The Gauss point locations for the solid elements are documented in Elements for Nonlinear Analysis in the *NX Nastran Basic Nonlinear Analysis User's Guide*.

REMARKS RELATED TO SOLS 601 AND 701:

1. For structural problems, MID is the identification number of a MAT1, MAT9, MAT10, MAT11, MATG, MATPLCY, MATSMA or MATVE entry for 3D solid elements and a MAT1, MAT3, MAT10, MATPLCY, MATSMA or MATVE entry for axisymmetric elements. For 3D solid elements in linear analysis, CORDM is ignored if MID does not reference a MAT9 or MAT11 entry and stresses are output in the basic coordinate system. For axisymmetric elements in a linear analysis, if MID does not reference a MAT3 entry, stresses are output in the basic coordinate system.

2. For geometric nonlinear analysis, it is recommended not to use incompatible modes by setting IN=2 or 3. If bending behavior is significant, it is highly recommended that elements with midside nodes be used.

3. For SOL 701, IN=2 is the default, i.e. incompatible modes are not used by default for SOL 701. To use incompatible modes in SOL 701, IN=BUBBLE must be specified.

4. By default, stresses labeled "NONLINEAR STRESSES" are output in the undeformed element coordinate system. However, for 3D solid elements which reference a MAT9 or MAT11 entry, ELRESCS=1 may be specified in NXSTRAT entry to request the output of these stress results in the material property coordinate system with CORDM=-1 being treated as basic coordinate system. For 3D solid elements which do not reference a MAT9 or MAT11 entry, ELRESCS=1 may be specified to output stresses in the basic coordinate system. Hyperelastic element stress is always output in the basic coordinate system.

5. ISOP is ignored. 2x2x2 integration (and equivalent) is used for elements with no midside nodes. 3x3x3 integration (and equivalent) is used for elements with midside nodes.

6. FCTN is not supported.

CHEXA ENTRY OPTIONS

CHEXA Entry Options – Linear Solutions(1)				
CHEXA	**Integration**	**IN**	**STRESS (Default: GRID)**	**ISOP**
8 Node	2x2x2 Reduced Shear with Bubble Function (Default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or 0 or REDUCED (Default)(2)
	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	
	2x2x2 Standard Isoparametric			FULL or 1
	3x3x3 Reduced Shear Only	THREE or 3		Blank or 0 or REDUCED
	3x3x3 Standard Isoparametric			FULL or 1
9 -20 Node	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or 0 or REDUCED
	2x2x2 Standard Isoparametric			FULL or 1
	3x3x3 Reduced Shear Only	Blank or 0 or THREE or 3 (Default)		Blank or 0 or REDUCED (Default)(2)
	3x3x3 Standard Isoparametric	THREE or 3		FULL or 1

1. All solutions except SOLs 106, 129, 153/159 with ANALYSIS=STRUCTURE, 601, and 701.
2. REDUCED is the default only for structural elements (FCTN="SMECH").

CHEXA Entry Options – SOL 401				
CHEXA	**Integration**	**IN**	**STRESS**	**ISOP**
8 Node(1)	2x2x2 with Bubble Function (Default)	BUBBLE or Blank or 0 (Default)	Ignored	Ignored
	2x2x2 Standard Isoparametric	TWO or 2		
9 -20 Node	3x3x3 Standard Isoparametric	Not Used(2)		

1. If IN=3 is specified, a value of IN=0 will be used instead as IN=3 is not supported. A warning message will also be issued in the f06 file.

2. A valid entry must be specified since the PSOLID entry may be used by other element types.

CHEXA Entry Options – Basic Nonlinear Solutions(1)

CHEXA	Integration	IN	STRESS (Default: GRID)	ISOP
8 Node(3)	2x2x2 Reduced Shear with Bubble Function (Default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or 0 or REDUCED (Default)(2)
	2x2x2 Reduced Shear Only	TWO or 2		
	2x2x2 Standard Isoparametric			FULL or 1
9 -20 Node	3x3x3 Standard Isoparametric	Not Used(4)		Not Used(4)

1. SOLs 106, 129, and 153/159 with ANALYSIS=STRUCTURE. When running a linear solution (geometric nonlinearity off and no nonlinear material), the “CHEXA Entry Options – Linear Solutions” table should be used.

2. REDUCED is the default only for structural elements (FCTN=“SMECH”).

3. If IN=3 is specified, a value of IN = 0 will be used instead as IN=3 is not supported. A warning message will also be issued in the f06 file.

4. A valid entry must be specified since the PSOLID entry may be used by other element types.

CHEXA Entry Options – Advanced Nonlinear Solutions(1)

CHEXA	Integration	IN	STRESS	ISOP

8 Node	2x2x2 Standard Isoparametric with Incompatible Modes (Default)	BUBBLE or Blank or 0 (Default)(2)(3)	Blank or GRID or GAUSS or 1	Ignored
	2x2x2 Standard Isoparametric	TWO or 2		
	2x2x2 U/P Formulation(3)			
20 Node(4)	3x3x3 Standard Isoparametric	Ignored		
	3x3x3 U/P Formulation(3)			

1. SOLs 601 and 701.
2. For SOL 701, IN=2 is the default, i.e. incompatible modes are not used by default for SOL 701. To use incompatible modes in SOL 701, IN=BUBBLE must be specified.
3. U/P formulation can be set by UPFORM parameter entry in NXSTRAT. When U/P formulation is used, it will overwrite IN=BUBBLE option.
4. SOL 701 only supports 8 grid CHEXA.

CPENTA ENTRY OPTIONS

CPENTA Entry Options – Linear Solutions(1)				
CPENTA	**Integration**	**IN**	**STRESS (Default: GRID)**	**ISOP**
6 Node	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	Blank or GRID or GAUSS or 1	Blank or 0 or REDUCED (Default)(2)
	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	
	2x3 Standard Isoparametric			FULL or 1
	3x7 Reduced Shear Only	THREE or 3		Blank or 0 or REDUCED
	3x7 Standard Isoparametric			FULL or 1

7-15 Node	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or 0 or REDUCED
	2x3 Standard Isoparametric			FULL or 1
	3x7 Reduced Shear Only (Default)	Blank or 0 or THREE or 3 (Default)		Blank or 0 or REDUCED (Default)(2)
	3x7 Standard Isoparametric	THREE or 3		FULL or 1

1. All solutions except SOLs 106, 129, 153/159 with ANALYSIS=STRUCTURE, 601, and 701.
2. REDUCED is the default only for structural elements (FCTN="SMECH").

CPENTA Entry Options – SOL 401				
CPENTA	**Integration**	**IN**	**STRESS**	**ISOP**
6 Node(1)	2x3 with Bubble Function (Default)	BUBBLE or Blank or 0 (Default)	Ignored	Ignored
	2x3 Standard Isoparametric	TWO or 2		
7 -15 Node	3x7 Standard Isoparametric	Not Used(2)		

1. If IN=3 is specified, a value of IN=0 will be used instead as IN=3 is not supported. A warning message will also be issued in the f06 file.
2. A valid entry must be specified since the PSOLID entry may be used by other element types.

CPENTA Entry Options – Basic Nonlinear Solutions[1]				
CPENTA	**Integration**	**IN**	**STRESS (Default: GRID)**	**ISOP**
6 Node(3)	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	Blank or GRID or GAUSS or 1	Blank or 0 or REDUCED (Default)(2)
	2x3 Reduced Shear Only	TWO or 2		
	2x3 Standard Isoparametric			FULL or 1
7–15 Node	3x7 Standard Isoparametric	Not used(4)		Not used(4)

1. SOLs 106, 129, and 153/159 with ANALYSIS=STRUCTURE. When running a linear solution (geometric nonlinearity off and no nonlinear material), the "CPENTA Entry Options – Linear Solutions" table should be used.
2. REDUCED is the default only for structural elements (FCTN="SMECH").
3. If IN=3 is specified, a value of IN = 0 will be used instead as IN=3 is not supported. A warning message will also be issued in the F06 file.
4. A valid entry must be specified because the PSOLID card may be used by other element types.

CPENTA Entry Options – Advanced Nonlinear Solutions(1)

CPENTA	Integration	IN	STRESS	ISOP
6 Node	2x2x2 Standard Isoparametric with Incompatible Modes (Default)	Blank or 0 or BUBBLE (Default)(2)(3)	Blank or GRID or GAUSS or 1	Ignored
	2x2x2 Standard Isoparametric	TWO or 2		
	2x2x2 U/P Formulation(3)			
15 Node(4)	3x3x3 Standard Isoparametric	Ignored		
	3x3x3 U/P Formulation(3)			

1. SOLs 601 and 701.
2. For SOL 701, IN=2 is the default, i.e. incompatible modes are not used by default for SOL 701. To use incompatible modes in SOL 701, IN=BUBBLE must be specified.
3. U/P formulation can be set by UPFORM parameter entry in NXSTRAT. When U/P formulation is used, it will overwrite IN=BUBBLE option.
4. SOL 701 only supports 6 node CPENTA.

CPYRAM ENTRY OPTIONS

CPYRAM Entry Options – Linear Solutions(1)

CPYRAM	Integration		IN	STRESS (Default: GRID)	ISOP
	Structural	Heat Transfer			

5 Node	2x2x2 Bedrosian(4) Isoparametric	2x2x2 Bedrosian(4) Isoparametric	Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or 2 or 0 (Default)(2)
	3x3x3 Bedrosian(4) Isoparametric		1(3)		
6–13 Node	2x2x2 Bedrosian(4) Isoparametric	3x3x3 Bedrosian(4) Isoparametric	Blank or 0 (Default)		
	3x3x3 Bedrosian(4) Isoparametric		1(3)		

1. All solutions except SOLs 106, 129, 153/159 with ANALYSIS=STRUCTURE, 601, and 701.
2. The default value of Blank or 0 uses a standard formulation. A value of 2 specifies the B-BAR option for nearly incompressible materials.
3. For CPYRAM, IN=1 produces an overly stiff element.
4. For additional information, see G. Bedrosian, “Shape Functions and Integration Formulas for Three-Dimensional Finite Element Analysis”, *International Journal for Numerical Methods in Engineering*, vol. 35, 95–108 (1992).

CPYRAM Entry Options – SOL 401

CPYRAM	Integration	IN	STRESS	ISOP
5 Node	2x2x2 Bedrosian(2) Isoparametric (Default)	Blank or 0 (Default)	Ignored	Ignored
	3x3x3 Bedrosian(2) Isoparametric	1(1)		
6-13 Node	2x2x2 Bedrosian(2) Isoparametric (Default)	Blank or 0 (Default)		
	3x3x3 Bedrosian(2) Isoparametric	1(1)		

1. For CPYRAM, IN=1 produces an overly stiff element.
2. For additional information, see G. Bedrosian, “Shape Functions and Integration Formulas for Three-Dimensional Finite Element Analysis”, *International Journal for Numerical Methods in Engineering*, vol. 35, 95–108 (1992).

CPYRAM Entry Options – Basic Nonlinear Solutions(1)

CPYRAM	Integration	IN	STRESS (Default: GRID)	ISOP

5 Node	2x2x2 Bedrosian(4) Isoparametric	Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or 2 or 0 (Default)(2)
	3x3x3 Bedrosian(4) Isoparametric	1(3)		
6–13 Node	2x2x2 Bedrosian(4) Isoparametric	Blank or 0 (Default)		
	3x3x3 Bedrosian(4) Isoparametric	1(3)		

1. SOLs 106, 129, and 153/159 with ANALYSIS=STRUCTURE. Only for geometric nonlinearity. When geometric nonlinearity is off, the "CPYRAM Entry Options – Linear Solutions" table should be used.
2. The default value of Blank or 0 uses a standard formulation. A value of 2 specifies the B-BAR option for nearly incompressible materials.
3. For CPYRAM, IN = 1 produces an overly stiff element.
4. For additional information, see G. Bedrosian, "Shape Functions and Integration Formulas for Three-Dimensional Finite Element Analysis", *International Journal for Numerical Methods in Engineering*, vol. 35, 95–108 (1992).

CPYRAM Entry Options – Advanced Nonlinear Solutions(1)

CPYRAM	Integration	IN	STRESS	ISOP
5 Node	2x2x2 Standard Isoparametric with Incompatible Modes (Default)	Blank or 0 or BUBBLE (Default)(2)(3))	Blank or GRID or GAUSS or 1	Ignored
	2x2x2 Standard Isoparametric	TWO or 2		
	2x2x2 U/P Formulation(3)			
6-13 Node(4)	3x3x3 Standard Isoparametric	Ignored		
	3x3x3 U/P Formulation(3)			

1. SOLs 601 and 701.
2. For SOL 701, IN=2 is the default, i.e. incompatible modes are not used by default for SOL 701. To use incompatible modes in SOL 701, IN=BUBBLE must be specified.
3. U/P formulation can be set by UPFORM parameter entry in NXSTRAT. When U/P formulation is used, it will overwrite IN=BUBBLE option.
4. SOL 701 only supports 5 node CPYRAM.

CTETRA ENTRY OPTIONS

CTETRA Entry Options – Linear Solutions(1)				
CTETRA	**Integration**	**IN**	**STRESS (Default: GRID)**	**ISOP**
4 Node	1-Point Standard Isoparametric (Default)	Blank or 0 or TWO or 2 (Default)	Blank or GRID or GAUSS or 1	Blank or FULL
	5-Point Standard Isoparametric	THREE or 3		
5–10 Node	5-Point Standard Isoparametric	Not used(2)		

1. All solutions except SOLs 106, 129, 153/159 with ANALYSIS=STRUCTURE, 601, and 701.
2. A valid entry must be specified because the PSOLID card may be used by other element types.

CTETRA Entry Options – SOL 401				
CTETRA	**Integration**	**IN**	**STRESS**	**ISOP**

4 Node	1-Point Standard Isoparametric (Default)	Blank or 0 or TWO or 2 (Default)	Ignored	Ignored
	5-Point Standard Isoparametric	THREE or 3		
5–10 Node	4-Point Standard Isoparametric (Default)	Blank or 0 or TWO or 2 (Default)		
	5-Point Standard Isoparametric	THREE or 3		

CTETRA Entry Options – Basic Nonlinear Solutions(1)				
CTETRA	**Integration**	**IN**	**STRESS (Default: GRID)**	**ISOP**
4 Node	1-Point Standard Isoparametric (Default)	Blank or 0 or TWO or 2 (Default)	Blank or GRID or GAUSS or 1	Blank or FULL
	5-Point Standard Isoparametric	THREE or 3		
5–10 Node	4-Point Standard Isoparametric (Default)	Blank or 0 (Default)		
	5-Point Standard Isoparametric	THREE or 3		
1. SOLs 106, 129, and 153/159 with ANALYSIS=STRUCTURE. When running a linear solution (geometric nonlinearity off and no nonlinear material), the "CTETRA Entry Options – Linear Solutions" table should be used.				

CTETRA Entry Options – Advanced Nonlinear Solutions(1)				
CTETRA	**Integration**	**IN**	**STRESS**	**ISOP**
4 Node	1-Point Standard Isoparametric	Ignored	Blank or GRID or GAUSS or 1	Ignored
5–10 Node(3)	17-Point Standard Isoparametric			
	17-Point U/P Formulation(2)			

1. SOLs 601 and 701.
2. U/P formulation can be set by UPFORM parameter entry in NXSTRAT.
3. SOL 701 only supports 5 node CTETRA.

AXISYMMETRIC OPTIONS FOR CQUADX4

CQUADX4 Entry Options – Linear Solutions(1)			
Integration	**IN**	**STRESS (Default: GRID)**	**Nonlinear Capability**
Full Integration using internal shape functions	Blank or 0 (default)	Blank or GRID (corner) or GAUSS or 1	No
Full Integration without using internal shape functions	1		No
Mean Dilatational formulation	2		No
1. All solutions except SOLs 601 and 701.			

CQUADX4 Entry Options – Advanced Nonlinear Solutions(1)			
Integration	**IN**	**STRESS**	**ISOP**
Full Integration with incompatible modes	Blank or 0 or BUBBLE (Default)	Blank or GRID or GAUSS or 1	Ignored
Full Integration without incompatible modes	1, 2 or 3		
1. SOL 601.			

AXISYMMETRIC OPTIONS FOR CQUADX8, CTRAX3, AND CTRAX6

CQUADX8, CTRAX3, and CTRAX6 Entry Options – Linear Solutions(1)			
Integration	**IN**	**STRESS (Default: GRID)**	**Nonlinear Capability**

Full Integration	Blank or 0 (default)	Blank or GRID (corner) or GAUSS or 1	No
Mean Dilatational formulation	2		No
1. All solutions except SOLs 601 and 701.			

CQUADX8, CTRAX3, and CTRAX6 Entry Options – Advanced Nonlinear Solutions	
The IN and ISOP fields are ignored by the CQUADX8, CTRAX3, and CTRAX6 elements in a SOL 601 solution.	**STRESS (Default: GRID)**
	Blank or GRID or GAUSS or 1

PTUBE

Tube Property

Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM	OD2			

EXAMPLE:

PTUBE	2	6	6.29	0.25					

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 3 and 4. (Integer > 0)
OD	Outside diameter of tube. (Real > 0.0)
T	Thickness of tube. (Real; T ≤ OD/2.0)
NSM	Nonstructural mass per unit length. (Real)
OD2	Diameter of tube at second grid point (G2) on CTUBE entry. (Real; Default = OD)

REMARKS:

1. If T is zero, a solid circular rod is assumed.

2. PTUBE entries must all have unique property identification numbers.

3. For structural problems, MID must reference a MAT1 material entry.

4. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.

5. Tapered OD tubes with constant wall thickness are available for heat transfer only. The effective diameter is given by:

$$D_{effective} = \mathrm{T} + \frac{D_2 - D_1}{log_e\left(\frac{D_2 - \mathrm{T}}{D_1 - \mathrm{T}}\right)}$$

where:

D_1 = OD

D_2 = $\begin{cases} \mathrm{OD2 \text{ if } OD2} \neq 0 \\ \mathrm{OD \text{ if } OD2} = 0 \text{ or blank} \end{cases}$

PVISC

Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

EXAMPLE:

PVISC	3	6.2	3.94						

FIELDS:

Field	Contents
PIDi	Property identification number. (Integer > 0)
CE1, CE2	Viscous damping values for extension in units of force per unit velocity. (Real)
CR1, CR2	Viscous damping values for rotation in units of moment per unit velocity. (Real)

REMARKS:

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.

PWELD

WELD Element Property

Defines the properties of connector (CWELD) elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

EXAMPLE:

PWELD	100	3	1.0						

FIELDS:

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remark 1 . (Integer > 0)
D	Diameter of the connector. See Remark 1 . (Real > 0)
MSET	Flag to eliminate m-set degrees of freedom.See Remark 2 . (Character: “ON” or “OFF”; Default= “OFF”).
TYPE	Character string indicating the type of connection. See Remark 3 . (Character: ”SPOT” or blank; Default=blank, “SPOT”= spot weld connector, blank= general connector)
LDMIN	Smallest ratio of weld length to diameter. See Remark 4 . (Real > 0; Default=0.2)
LDMAX	Largest ratio of weld length to diameter. See Remark 4 . (Real > 0; Default=5.0)

REMARKS:

1. Material MID, diameter D, and the weld length are used to calculate the stiffness of the connector in 6 directions. MID can only refer to the MAT1 Bulk Data entry.

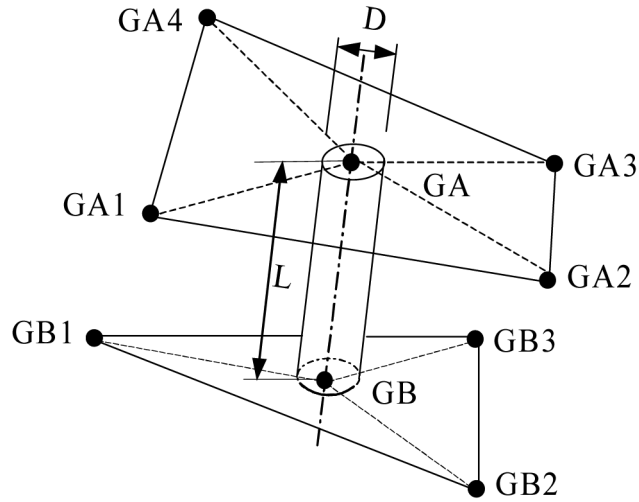


Figure 16-24. Length and Diameter of the Weld Connector

2. MSET is only used for formats “GRIDID” and “ELEMID”. MSET=”ON” generates explicit m-set constraints. MSET=”OFF” (default) incorporates constraints at the element stiffness matrix level avoiding explicit m-set constraint equations. For example, if a patch-to-patch connection is specified with the formats “GRIDID” or “ELEMID” on the CWELD entry, and MSET=ON is specified, 2x6 explicit constraints are generated which connect the 6 degrees of freedom of GA to the translational degrees of freedom of GAI and the 6 degrees of freedom of GB to GBI. The 2x6 degrees of freedom of GA and GB are put into the m-set. The constraints are labeled “RWELD”. The identification numbers of the generated RWELD constraint elements start with an offset specified on PARAM, OSWELM. If MSET=OFF is specified, the 2x6 constraint equations are built into the stiffness matrix of the CWELD element thereby condensing the 2x6 degrees of freedom of GA and GB. MSET=OFF is only available for the patch-to-patch connection (GRIDID or ELEMID).

3. When TYPE= “SPOT”, and the format on the CWELD entry is either “ELEMID”, “ELPAT”, or “PARTPAT”, the effective length for the stiffness of the CWELD element is set to L_e=1/2(t_A+t_B) regardless of the distance GA to GB. t_A and t_B are the shell thicknesses of shell *A* and *B* respectively. For all other cases, the effective length of the CWELD element is equal to the true length, the distance of

GA to GB, as long as the ratio of length to diameter is in the range $0.2 \leq L/D \leq 5.0$. If L is below the range, the effective length is set to $L_e=0.2D$ and if L is above the range, the effective length is set to $L_e=5.0D$.

4. When TYPE is blank, the weld length is the distance from GA to GB, as long as the L/D ratio is within LDMIN≤L/D≤LDMAX. If the ratio is below LDMIN, the weld length is set to LDMIN*D, and if the ratio is above LDMAX, the weld length is set to LDMAX*D.

Chapter 17: Bulk Data Entries Q—S

QBDY1

Boundary Heat Flux Load for CHBDYj Elements, Form 1

Defines a uniform heat flux into CHBDYj elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	EID2	EID3	EID4	EID5	EID6	

EXAMPLE:

QBDY1	109	1.-5	721						

ALTERNATE FORMAT AND EXAMPLE:

QBDY1	SID	Q0	EID1	“THRU”	EID2				
QBDY1	109	1.-5	725	THRU	735				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Heat flux into element. (Real)
EIDi	CHBDYj element identification numbers. (Integer ≥ 0 or “THRU”. For “THRU” option EID2 >EID1.)

REMARKS:

1. QBDY1 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into an element is given by the equation:

$$P_{in} = (\text{Effective area}) \cdot Q0$$

2. QBDY1 entries must be referenced on a TLOAD entry for use in transient analysis. The total power into an element is given by the equation:

$$P_{in}((t) = (\text{Effective area}) \cdot Q0 \cdot F(t - \tau))$$

   where the function of time $F(t-\tau)$ is specified on a TLOAD1 or TLOAD2 entry.

3. The sign convention for Q0 is positive for heat input.

QBDY2

Boundary Heat Flux Load for CHBDYj Elements, Form 2

Defines grid point heat flux into CHBDYj elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q01	Q02	Q03	Q04	Q05	Q06	
	Q07	Q08							

EXAMPLE:

QBDY2	109	721	1.-5	1.-5	2.-5	2.-5			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer >0)
EID	Identification number of an CHBDYj element. (Integer > 0)
Q0i	Heat flux at the i-th grid point on the referenced CHBDYj element. (Real or blank)

REMARKS:

1. QBDY2 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into each point i on an element is given by

$$P_i = AREA_i \cdot Q0i$$

2. QBDY2 entries must be referenced on a TLOAD entry for use in transient analysis. All connected grid points will have the same time function but may have individual delays. The total power into each point i on an element is given by

$$P_i(t) = AREA_i \cdot Q0i \cdot F(t - \tau_i)$$

where $F(t-\tau_i)$ is a function of time specified on a TLOAD1 or TLOAD2 entry.

3. The sign convention for Q0i is positive for heat flux input to the element.

REMARKS RELATED TO SOL 601:

1. Heat flux values on midside grid points are not supported. Q05 to Q08 are ignored.

2. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions (similar to PLOAD4 load application).

QBDY3

Boundary Heat Flux Load for a Surface

Defines a uniform heat flux load for a boundary surface.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

EXAMPLE:

QBDY3	2	20.0	10	1	THRU	50	BY	2	

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real)
CNTRLND	Control point for thermal flux load. (Integer ≥ 0; Default=0)
EIDi	CHBDYj element identification numbers. (Integer > 0 or “THRU” or “BY”)

REMARKS:

1. QBDY3 entries must be selected in Case Control (LOAD=SID) to be used in steady state. The total power into a surface is given by the equation:
   - if CNTRLND ≤ 0 then P_{in}=(Effective area) ·Q0

- if CNTRLND > 0 then P_{in}=(Effective area) $\cdot Q0 \cdot u_{CNTRLND}$

  where $u_{CNTRLND}$ is the temperature of the control point and is used as a load multiplier.

2. In transient analysis SID is referenced by a TLOADi Bulk Data entry through the DAREA entry. A function of time $F(t-\tau)$ defined on the TLOADi multiplies the general load, with τ specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD=SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

3. The CNTRLND multiplier cannot be used with any higher-order elements.

4. When using “THRU” or “BY”, all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.

REMARKS RELATED TO SOL 601:

1. CNTRLND is ignored.

QHBDY

Boundary Heat Flux Load

Defines a uniform heat flux into a set of grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	FLAG	Q0	AF	G1	G2	G3	G4	
	G5	G6	G7	G8					

EXAMPLE:

QHBDY	2	AREA4	20.0		101	102	104	103	

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
FLAG	Type of face involved (must be one of the following: “POINT”, “LINE”, “REV”, “AREA3”, “AREA4”, “AREA6”, “AREA8”)
Q0	Magnitude of thermal flux into face. Q0 is positive for heat into the surface. (Real)
AF	Area factor depends on type. (Real > 0.0 or blank)
Gi	Grid point identification of connected grid points. (Integer > 0 or blank)

REMARKS:

1. The continuation entry is optional.

2. For use in steady state analysis, the load set is selected in the Case Control Section (LOAD=SID).

3. In transient analysis SID is referenced by a TLOADi Bulk Data entry through the DAREA entry. A function of time defined on the TLOADi entry multiplies the general load. specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD=SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

4. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.

5. The flux is applied to each point i by the equation

$$P_i = AREA_i \cdot Q0$$

6. The number of connected points for the types are 1 (POINT), 2 (LINE, REV), 3 (AREA3), 4 (AREA4), 4-6 (AREA6), 5-8 (AREA8).

7. The area factor AF is used to determine the effective area for the POINT and LINE types. It equals the area and effective width, respectively. It is not used for the other types, which have their area defined implicitly and must be left blank.

8. The type of face (FLAG) defines a surface in the same manner as the CHBDYi data entry. For physical descriptions of the geometry involved, see the CHBDYG discussion.

REMARKS RELATED TO SOL 601:

1. AF is used only for POINT type.

QSET

Generalized Degrees-of-Freedom

Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

QSET	15	123456	1	7	9	2	105	6	

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.

3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

QSET1

Generalized Degrees-of-Freedom (Alternate Form of QSET Entry)

Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

EXAMPLE:

QSET1	123456	1	7	9	22	105	6	22	
	52	53							

ALTERNATE FORMAT AND EXAMPLE:

QSET1	C	ID1	“THRU”	ID2					
QSET1	0	101	THRU	110					

FIELDS:

Field	Contents
C	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2.)

REMARKS:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.

3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

QVECT

Thermal Vector Flux Load

Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc.-						

EXAMPLE:

QVECT	10	20.0	1000.0		1.0	1.0	1.0	101	
	20	21	22	23					

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Magnitude of thermal flux vector into face. (Real or blank)
TSOUR	Temperature of the radiant source. (Real or blank)
CE	Coordinate system identification number for thermal vector flux. (Integer ≥ 0 or blank)
Ei	Vector components (direction cosines in coordinate system CE) of the thermal vector flux. (Real; Default=0.0)
TIDi	TABLEDi entry identification numbers defining the components as a function of time. (Integer > 0)
CNTRLND	Control point. (Integer ≥ 0; Default=0)

Field	Contents
EIDi	Element identification number of a CHBDYE, CHBDYG, or CHBDYP entry. (Integer ≠ 0 or “THRU”)

REMARKS:

1. The continuation entry is required.

2. If the coordinate system CE is not rectangular, then the thermal vector flux is in different directions for different CHBDYi elements. The direction of the thermal vector flux over an element is aligned to be in the direction of the flux vector at the geometric center of the element. The geometric center is measured using the grid points and includes any DISLIN specification on the VIEW entry for TYPE=LINE CHBDYi elements. The flux is presumed to be uniform over the face of each element; i.e., the source is relatively distant.

3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD=SID). The total power into an element is given by:

   - If CNTRLND=0 then, $P_{in}=-\alpha A\,(\vec{e}\cdot\vec{n})\cdot Q0$.

   - If CNTRLND > 0 then, $P_{in}=-\alpha A\,(\vec{e}\cdot\vec{n})\cdot Q0\cdot u_{\mathrm{CNTRLND}}$.

     where

     α = face absorptivity (supplied from a RADM statement).

     A = face area as determined from a CHBDYi surface element.

     $\vec{e}$ = vector of direction cosines E1, E2, E3.

     $\vec{n}$ = face normal vector. See CHBDYi entries.

     $\vec{e}\cdot\vec{n}$ = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

     u_{cntrlnd} = temperature value of the control point used as a load multiplier.

4. If the absorptivity is constant, its value is supplied by the ABSORP field on the RADM entry. If the absorptivity is not a constant, the thermal flux is assumed to have a wavelength distribution of a black body at the temperature TSOUR.

- For a temperature-dependent absorptivity, the element temperature is used to determine α.
- For a wavelength-dependent absorptivity, the integration of the flux times is computed for each wavelength band. The sum of the integrated thermal fluxes over all the wavelength bands is Q0. The wave bands are specified with the RADBND entry.
- The user has the responsibility of enforcing Kirchhoff's laws.

5. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the DAREA specification. A function of time $F(t-\tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. $F(t-\tau)$ is a function of time specified on the TLOADi entry. The value of is calculated for each loaded grid point. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD=SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

   The total power into an element is given by:

   - If CNTRLND=0 then, $P_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot Q0 \cdot F(t-\tau)$.
   - If CNTRLND > 0 then,

     $P_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot F(t-\tau) Q0 \cdot u_{\mathrm{CNTRLND}}$.

6. If the referenced face is of TYPE=ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.

7. If the referenced face is of TYPE=REV, the thermal flux vector must be parallel to the axis of symmetry if an axisymmetric boundary condition is to be maintained.

8. When applied to a surface element associated with a radiation enclosure cavity, any incident energy that is not absorbed (α < 1.0) is lost from the system and is not accounted for in a reflective sense (α + ρ=1.0).

QVOL

Volume Heat Addition

Defines a rate of volumetric heat addition in a conduction element.

FORMAT:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

EXAMPLE:

QVOL	5	10.0	101	10	12	11	9		

FIELDS:

Field	Contents
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0; Default=0)
EIDi	A list of heat conduction elements. (Integer > 0 or “THRU” or “BY”)

REMARKS:

1. EIDi has material properties (MAT4) that include HGEN, the element material property for heat generation, which may be temperature dependent. This

association is made through the element EID. If HGEN is temperature dependent, it is based on the average element temperature.

2. QVOL provides either the constant volumetric heat generation rate or the load multiplier. QVOL is positive for heat generation. For steady-state analysis, the total power into an element is

   - If CNTRLND=0, then P_{in}=volume · HGEN · QVOL.
   - If CNTRLND > 0, then P_{in}=volume · HGEN · QVOL · u_{CNTRLND}.

     where u_{CNTRLND}. is the temperature multiplier.

3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD=SID).

4. In transient analysis SID is referenced by a TLOADi Bulk Data entry. A function of time $F\,[t - \tau]$ defined on the TLOADi entry multiplies the general load where τ specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD=SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

5. For “THRU” or “BY”, all intermediate referenced heat conduction elements must exist.

6. The CNTRLND multiplier cannot be used with any higher-order elements.

REMARKS RELATED TO SOL 601:

1. CNTRLND is ignored.

RADBC

Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADBC	NODAMB	FAMB	CNTRLND	EID1	EID2	EID3	-etc.-		

EXAMPLE:

RADBC	5	1.0	101	10					

FIELDS:

Field	Contents
NODAMB	Ambient point for radiation exchange. (Integer > 0)
FAMB	Radiation view factor between the face and the ambient point. (Real ≥ 0.0)
CNTRLND	Control point for radiation boundary condition. (Integer ≥ 0; Default=0)
EIDi	CHBDYi element identification number. (or “THRU” or “BY”)

REMARKS:

1. The basic exchange relationship is:
   - if CNTRLND=0, then $q=\sigma \cdot \text{FAMB} \cdot (\varepsilon_e T4_e - \alpha_e T^4_{\text{amb}})$
   - if CNTRLND > 0, then

$$q = \sigma \cdot \text{FAMB} \cdot u_{\text{CNTRLND}} \cdot (\varepsilon_e T^4_e - \alpha_e T^4_{\text{amb}})$$

2. NODAMB is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space. No surface element that is a member of a radiation enclosure cavity may also have a radiation boundary condition applied to it.

3. Two PARAM entries are required when stipulating radiation heat transfer:

   - TABS defines the absolute temperature scale; this value is added internally to any specified temperature given in the problem. Upon solution completion, this value is subtracted internally from the solution vector.

   - SIGMA (σ) is the Stefan-Boltzmann constant.

4. RADBC allows for surface radiation to space in the absence of any cavity behavior. The emissivity and absorptivity are supplied from a RADM entry.

5. When using “THRU” or “BY”, all intermediate referenced CHBDYi surface elements must exist.

REMARKS RELATED TO SOL 601:

1. CNTRLND is ignored.

2. The basic radiation heat exchange is:
   $q = \sigma \bullet \text{FAMB} \bullet \varepsilon\ (T^4 - T^4_{\text{amb}})$

3. Note that absorptivity is assumed to be equal to emissivity.

4. The temperature on NODAMB must be prescribed on a TEMPBC or SPC entry.

RADBND

Radiation Wavelength Band Definition

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADBND	NUMBER	PLANCK2	λ1	λ2	λ3	λ4	λ5	λ6	
	λ7	etc.							

EXAMPLE:

RADBND	6	14388.0	1.0	2.0	4.0	8.0	12.0		

FIELDS:

Field	Contents
NUMBER	Number of radiation wave bands. See Remarks. (Integer > 1)
PLANCK2	Planck's second radiation constant. See Remarks. (Real > 0.0)
λi (LAMBDAi)	Highest wavelength of the i-th wave band. See Remarks. (Real ≥ 0.0)

REMARKS:

1. Only one RADBND entry may be specified in the Bulk Data Section and must always be used in conjunction with the RADM entry.
2. PLANCK2 has the units of wavelength times temperature. The same units of length must be used for LAMBDAi as for PLANCK2. The units of temperature

must be the same as those used for the radiating surfaces. For example: 25898. μm °R or 14388. μm °K.

3. The first wavelength band extends from 0 to LAMBDA1 and the last band extends from LAMBDAn to infinity, where n=NUMBER − 1.

4. Discontinuous segments in the emissivity versus wavelength piecewise linear curve must be treated as a wavelength band of zero width.

5. LAMBDAi must be specified in ascending order, and all LAMBDAi fields where i is greater than or equal to NUMBER must be blank.

RADCAV

Radiation Cavity Identification

Identifies the characteristics of each radiant enclosure.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPCH	NFECI	RMAX		
	SET11	SET12	SET21	SET22	SET31	SET32	etc.		

EXAMPLE:

RADCAV	1	1					.99		
	3	5	4	5	7	5			

FIELDS:

Field	Contents
ICAVITY	Unique cavity identification number associated with enclosure radiation. (Integer > 0)
ELEAMB	CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.)
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character=“YES” or “NO”; Default=“YES”)
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. (0.0 ≤ Real ≤ 1.0; Default=0.0)
PRTPCH	Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer=0, 1, 2, or 3; Default=0):

Field	Contents
NFECI	Controls whether finite difference or contour integration methods are to be used in the calculation of view factors in the absence of a VIEW3D Bulk Data entry. (Character="FD" or "CONT"; See Remark 4 . for default.)
RMAX	Subelement area factor. See Remark 5 . (Real ≥ 0.0; Default=0.1)
SETij	Set identification pairs for the calculation of global view factors. Up to 30 pairs may be specified (i=1 to 2 and j=1 to 30). (Integer > 0)

PRTPCH	Print/Punch
0 (default)	print and punch
1	no print
2	no punch
3	no print or punch

REMARKS:

1. For the surfaces of an incomplete enclosure (view factors add up to less than 1.0), a complete enclosure may be achieved (SUM=1.0) by specifying an ambient element, ELEAMB. When multiple cavities are defined, each cavity must have a unique ambient element if ambient elements are desired. No elements can be shared between cavities.

2. Third-body shadowing is ignored in the cavity if SHADOW="NO". In particular, if it is known a priori that there is no third-body shadowing, SHADOW=NO overrides KSHD and KBSHD fields on the VIEW Bulk Data entry as well as reduces the calculation time immensely.

3. The view factors for a complete enclosure may add up to slightly more than 1.0 due to calculation inaccuracies. SCALE can be used to adjust all the view factors proportionately to acquire a summation equal to the value specified for SCALE. If SCALE is left blank or set to 0.0, no scaling is performed.

4. If the VIEW3D Bulk Data entry is not specified, the view factors are calculated using finite difference and contour integration methods. If NFECI="FD", then all view factors are calculated using the finite difference technique. NFECI="CONT" invokes contour integration for all view factor calculations. If NFECI is blank, the

program selects a method to use between any two particular elements based on RMAX.

5. The comparison value for RMAX is equal to A_s / d^2_{rs} where A_s is the area of a subelement and d_{rs} is the distance between two subelements r and s for which view factors are being computed. When NFECI is blank, the program selects the contour integral method only if A_s / d^2_{rs} > RMAX.

6. When a number of elements are grouped together and considered as a conglomerate surface, view factors can be calculated between these groups. These are referred to as global view factors. The SET1 Bulk Data entry is used to define the conglomerate. When using this feature, negative EIDs are not allowed.

7. If a RADLST and RADMTX entry exists for this cavity ID, new view factors are not computed and the existing RADLST and RADMTX are used in the thermal analysis.

RADLST

Listing of Enclosure Radiation Faces

Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADLST	ICAVITY	MTXTYP	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	-etc.-							

EXAMPLE:

RADLST	2	1	2	3	4	5	6	7	

FIELDS:

Field	Contents
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation faces. (Integer > 0)
MTXTYP	Type of radiation exchange matrix used for this cavity. (Integer ≤ 4 and ≠ 0; Default=1 for an enclosure without an ambient element. Default=4 for an enclosure with an ambient element as specified on the RADCAV entry.) 1: Symmetric view factor matrix [F] and nonconservative radiation matrix [R]. 2: Symmetric exchange factor matrix [$\mathfrak{J}$] and conservative radiation matrix [R]. 3: Unsymmetrical exchange factor matrix [$\mathfrak{J}$] and conservative radiation matrix [R].

Field	Contents
	4: Symmetric view factor matrix [F] and conservative radiation matrix [R].
	-n: The first n CHBDYi elements may lose energy to space but the remainder may not. Symmetric exchange factor matrix [F] and nonconservative radiation matrix [R].
EIDi	Identification numbers for the CHBDYi elements in this radiation cavity. (Integer ≠ 0 or “THRU”)

REMARKS:

1. A radiation EIDi list isolates those CHBDYi surface element faces that are to communicate in a radiation enclosure. View-factor calculation and RADMTX formation for an enclosure is performed only for (or among) those faces identified within the same RADCAV.

2. A radiation exchange matrix (RADMTX) can only reference one radiative face list (RADLST). The companion RADCAV, RADLST, and RADMTX must share a unique ICAVITY.

3. For each EIDi, the appropriate CHBDYi element is located, and the proper RADM entry ID field found.

4. If the radiation exchange matrix or any radiation boundary conditions are available from an external source, the RADMTX must be user generated.

5. Multiple RADLST entries may be specified.

RADM

Radiation Boundary Material Property

Defines the radiation properties of a boundary element for heat transfer analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

EXAMPLE:

RADM	11		.45	.33	.29	.20	.17	.13	

FIELDS:

Field	Contents
RADMID	Material identification number. (Integer > 0)
ABSORP	Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2 . (0.0 ≤ Real ≤ 1.0)
EMISi	Surface emissivity at wavelength LAMBDAi or the temperature function curve multiplier if EMISi is variable (See the RADBND entry.) (0.0 ≤ Real ≤ 1.0)

REMARKS:

1. The RADM entry is directly referenced only by one of the CHBDYE, CHBDYG, or CHBDYP type surface element entries.

2. For radiation enclosure problems, ABSORP is set equal to emissivity. For QVECT loads, absorptivity is specified by ABSORP.

3. If there is more than one EMISi, then:

   - There must be a RADBND entry.

   - The number of EMISi may not exceed the number of LAMBDAi on the RADBND entry.

   - The emissivity values are given for a wavelength specified by the corresponding LAMBDAi on the RADBND entry. Within each discrete wavelength band, the emissivity is assumed to be constant.

   - At any specific wavelength and surface temperature, the absorptivity is exactly equal to the emissivity.

4. To perform any radiation heat transfer exchange, the user must furnish PARAM entries for:

   - TABS to define the absolute temperature scale.

   - SIGMA σ to define the Stefan-Boltzmann constant in appropriate units.

REMARKS RELATED TO SOL 601:

1. Only RADMID and EMIS1 are supported.

2. Note that absorptivity is assumed to be equal to emissivity.

RADMT

Radiation Boundary Material Property Temperature Dependence

Specifies table references for temperature dependent RADM entry radiation boundary properties.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADMT	RADMID	T(A)	T(ε_1)	T(ε_2)	T(ε_3)	T(ε_4)	T(ε_5)	T(ε_6)	
	T(ε_7)	-etc.-							

EXAMPLE:

RADMT	11		1	2	3	4	5	6	

FIELDS:

Field	Contents
RADMID	Material identification number. (Integer > 0)
T(A)	TABLEMj identifier for surface absorptivity. (Integer ≥ 0 or blank)
T(ε_i)	TABLEMj identifiers for surface emissivity. (Integer ≥ 0 or blank)

REMARKS:

1. The basic quantities on the RADM entry of the same RADMID are always multiplied by the corresponding tabular function.
2. Tables T(A) and T(ε_i) have an upper bound that is less than or equal to one and a lower bound that is greater than or equal to zero.

3. The TABLEMj enforces the element temperature as the independent variable. Blank or zero fields means there is no temperature dependence of the referenced property on the RADM entry.

REMARKS RELATED TO SOL 601:

1. Only RADMID and $T(\varepsilon_1)$ are supported.
2. Note that absorptivity is assumed to be equal to emissivity.

RADMTX

Radiation Exchange Matrix

Provides the $F_{ji}=A_j\ f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADMTX	ICAVITY	INDEX	Fi,j	Fi+1,j	Fi+2,j	Fi+3,j	Fi+4,j	Fi+5,j	
	Fi+6,j	-etc.-							

EXAMPLE:

RADMTX	2	1	0.0	0.1	0.2	0.2	0.3	0.2	

FIELDS:

Field	Contents
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation surface elements. (Integer > 0)
INDEX	Column number in the matrix. (Integer > 0)
$F_{k,j}$	If symmetric, the matrix values start on the diagonal (i=j) and continue down the column (k=i + 1, i + 2, etc.). If unsymmetrical, the values start in row (i=1). i refers to EIDi on the RADLST entry. (Real ≥ 0)

REMARKS:

1. If the matrix is symmetric, only the lower triangle is input, and i=j=INDEX. If the matrix is unsymmetrical, i=1, and j=INDEX.

2. Only one ICAVITY may be referenced for those faces that are to be included in a unique radiation matrix.

3. Coefficients are listed by column with the number of columns equal to the number of entries in the RADLST.

4. All faces involved in any radiation enclosure must be defined with an CHBDYi element.

RADSET

Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RADSET	ICAV1	ICAV2	ICAV3	ICAV4	ICAV5	ICAV6	ICAV7	ICAV8	
	ICAV9	-etc.-							

EXAMPLE:

RADSET	1	2	3	4					

FIELDS:

Field	Contents
ICAVi (ICAVITYi)	Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0)

REMARKS:

1. For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

RANDPS

Power Spectral Density Specification

Defines load set power spectral density factors for use in random analysis having the frequency-dependent form:

$$S_{jk}(F) = (X + iY)G(F)$$

FORMAT:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

EXAMPLES:

RANDPS	5	3	7	2.0	2.5	4			

RANDPS	5	3	7	2.0	2.5	7.5			

FIELDS:

Field	Contents
SID	Random analysis set identification number. (Integer > 0)
J	Subcase identification number of the excited load set. (Integer > 0)
K	Subcase identification number of the applied load set. (Integer with K ≥ J)
X, Y	Components of the complex number. (Real)
TID	*G(F)* (Real or Integer ≥ 0 or blank; for default behavior, see Remark 3) If real entry, value of *G(F)* used over all frequencies.

Field	Contents
	If integer entry, identification number of a TABRNDi entry that defines *G(F)*.

REMARKS:

1. Set identification numbers must be selected with the Case Control command (RANDOM = SID).

2. For auto spectral density, J = K, X must be greater than zero and Y must be equal to zero.

3. *G(F)* = 1.0 if the TID entry field is left blank, if TID = 0.0 (real), or if TID = 0 (integer).

4. RANDPS may only reference subcases included within a single loop (a change in direct matrix input is not allowed).

5. See the *NX Nastran Advanced Dynamic Analysis User's Guide* for a discussion of random analysis.

6. In the case of cyclic symmetry, Solution Sequence 118, J and K must refer to the coded subcase IDs. See "Using Cyclic Symmetry" in the *NX Nastran User's Guide* for the coding procedure.

7. In superelement analysis, J and K must refer to the superelement subcases. For example, if superelement 10 has SUBCASEs 1 and 2 and superelement 20 has SUBCASEs 3 and 4, then a separate RANDPS entry is required for each superelement, even though X, Y, and TID may be identical.

8. For uncoupled PSDF (no K > J entries), only one K = J entry is allowed for a unique value of J. For coupled PSDF (some K > J entries), any number of entries are allowed.

RANDT1

Autocorrelation Function Time Lag

Defines time lag constants for use in random analysis autocorrelation function calculation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

EXAMPLE:

RANDT1	5	10	3.2	9.6					

FIELDS:

Field	Contents
SID	Random analysis set identification number. (Integer > 0)
N	Number of time lag intervals. (Integer > 0)
T0	Starting time lag. (Real ≥ 0.0)
TMAX	Maximum time lag. (Real > T0)

REMARKS:

1. Time lag sets must be selected with the Case Control command RANDOM=SID.
2. At least one RANDPS entry must be present with the same set identification number.
3. The time lags defined on this entry are given by

$$T_i = T0 + \frac{\mathrm{TMAX} - T0}{N}(i - 1),\ i = 1, \mathrm{N} + 2$$

4. See the *NX Nastran Advanced Dynamic Analysis User's Guide* for a discussion of random analysis.

RBAR

Rigid Bar

Defines a rigid bar with six degrees-of-freedom at each end.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	

EXAMPLE:

RBAR	5	1	2	234	123				

FIELDS:

Field	Contents
EID	Element identification number.
GA, GB	Grid point identification number of connection points. (Integer > 0)
CNA, CNB	Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark 1 . (Integers 1 through 6 with no embedded blanks, or zero or blank.)
CMA, CMB	Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 2 and 3. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
ALPHA	Thermal expansion coefficient.

REMARKS:

1. The total number of components in CNA and CNB must equal six; for example, CNA=1236, CNB=34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.

2. If both CMA and CMB are zero or blank, all of the degrees-of-freedom not in CNA and CNB will be made dependent; i.e., they will be made members of the m-set.

3. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.

4. Element identification numbers should be unique with respect to all other element identification numbers.

5. Rigid elements, unlike MPCs, are not selected through the Case Control Section.

6. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

7. Rigid elements are ignored in heat transfer problems.

8. See R-Type Elements" in the *NX Nastran Element Library* for a discussion of rigid elements.

9. The RIGID case control command must be assigned to "LAGRAN" to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.

10. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

    The rigid element thermal strains are calculated by
    $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$.

    If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

REMARKS RELATED TO SOL 601 AND 701:

1. For SOL 601, RBAR may be simulated by rigid link, constraint equations, beam element or spring elements depending on whether it is a large displacement analysis and the parameter EQRBAR in the NXSTRAT entry. The values of CNA, CNB, CMA, and CMB also determine how RBAR is treated. See Section 2.7 of Advanced Nonlinear Theory and Modeling Guide for details on how the RBAR element is handled.

2. For SOL 701, RBAR is always treated as rigid, i.e., as rigid link or constraint equations.

3. No output of forces of multipoint constraint may be requested.

4. By default, a dependent degree-of-freedom may not be assigned as independent by a rigid link (from RBE2 or another RBAR entry) or multipoint constraint (from MPC, RBE2, RBE3 or another RBAR entry). An error message will be issued in such a case. However, if GENMPC=1 is specified in the NXSTRAT entry, a dependent degree-of-freedom of a multipoint constraint can (MPC option, see section 2.7.1 of the Advanced Nonlinear Theory and Modeling Guide) be assigned as independent by a rigid link or another multipoint constraint.

5. When an RBAR is simulated by a beam element or spring elements, no dependent degree-of-freedom is created for the RBAR.

6. ALPHA is ignored.

RBE1

Rigid Body Element, Form 1

Defines a rigid body connected to an arbitrary number of grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RBE1	EID	GN1	CN1	GN2	CN2	GN3	CN3		
		GN4	CN4	GN5	CN5	GN6	CN6		
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	-etc.-	ALPHA				

EXAMPLE:

RBE1	59	59	123	60	456				
	UM	61	246						

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
GNi	Grid points at which independent degrees-of-freedom for the element are assigned. (Integer > 0)
CNi	Independent degrees-of-freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark 1 . (Integers 1 through 6 with no embedded blanks.)
“UM”	Indicates the start of the degrees-of-freedom belonging to m-set. (Character)
GMj	Grid points at which dependent degrees-of-freedom are assigned. (Integer > 0)

Field	Contents
CMj	Dependent degrees-of-freedom in the global coordinate system at grid point(s) GMj. (Integers 1 through 6 with no embedded blanks.)
ALPHA	Thermal expansion coefficient. (Real; Default = 0.0)

REMARKS:

1. The total number of components in CN1 to CN6 must equal six; for example, CN1=123, CN2=3, CN3=2, CN4=3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.
2. The first continuation entry is not required if there are fewer than four GN points.
3. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
4. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
5. Element identification numbers should be unique with respect to all other element identification numbers.
6. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.
8. Rigid elements are ignored in heat transfer problems.
9. See R-Type Elements in the *NX Nastran Element Library* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.
11. The RIGID case control command must be assigned to “LAGRAN” to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.
12. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2

and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

The rigid element thermal strains are calculated by
$\varepsilon_{thermal} = \alpha(AVGTEMP(LOAD) - AVGTEMP(INIT))$.

If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

RBE2

Rigid Body Element, Form 2

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-	ALPHA				

EXAMPLE:

RBE2	9	8	12	10	12	14	15	16	
	20								

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.)
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. (Real; Default = 0.0)

REMARKS:

1. The components indicated by CM are made dependent (members of the m-set) at all grid points GMi.
2. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
3. Element identification numbers should be unique with respect to all other element identification numbers.
4. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
5. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.
6. Rigid elements are ignored in heat transfer problems.
7. See R-Type Elements in the *NX Nastran Element Library* for a discussion of rigid elements.
8. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.
9. The RIGID case control command must be assigned to "LAGRAN" to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.
10. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

    The rigid element thermal strains are calculated by
    $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$.

    If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

REMARKS RELATED TO SOLS 601 AND 701:

1. For SOL 601, RBE2 may be simulated by rigid links, constraint equations, beam elements or spring elements depending on whether it is a large displacement analysis and the parameter EQRBE2 in the NXSTRAT entry. See Section 2.7

of the Advanced Nonlinear Theory and Modeling Guide for details on how the RBE2 element is handled.

2. For SOL 701, RBE2 is always treated as a rigid, i.e., as a rigid link or constraint equations.

3. No output of forces of multipoint constraint may be requested.

4. By default, a dependent degree-of-freedom may not be assigned as independent by a rigid link (from RBAR or another RBE2 entry) or multipoint constraint (from MPC, RBAR, RBE3 or another RBE2 entry). An error message will be issued in such a case. However, if GENMPC=1 is specified in the NXSTRAT entry, a dependent degree-of-freedom of a multipoint constraint (MPC option, see section 2.7.1 of the Advanced Nonlinear Theory and Modeling Guide) can be assigned as independent by a rigid link or another multipoint constraint.

5. When an RBE2 is simulated by beam or spring elements, no dependent degree-of-freedom is created for the RBAR.

6. ALPHA is ignored.

RBE3

Interpolation Constraint Element

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	-etc.-	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			
	“ALPHA”	ALPHA							

EXAMPLE:

RBE3	14		100	1234	1.0	123	1	3	
	5	4.7	1	2	4	6	5.2	2	
	7	8	9	5.1	1	15	16		
	UM	100	14	5	3	7	2		
	ALPHA	17.3E-6							

FIELDS:

Field	Contents
EID	Element identification number. Unique with respect to other rigid elements. (Integer > 0)
REFGRID	Reference grid point identification number. (Integer > 0)

Field	Contents
REFC	Component numbers at the reference grid point. (Any of the Integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the Integers 1 through 6 with no embedded blanks.)
Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
“UM”	Indicates the start of the degrees-of-freedom belonging to the m-set. The default action is to assign only the components in REFC to the m-set. (Character)
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
“ALPHA”	Indicates that the next field is the coefficient of thermal expansion. (Character)
ALPHA	Coefficient of thermal expansion. See Remark 10. (Real; Default = 0.0)

REMARKS:

1. It is recommended that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.

2. Blank spaces may be left at the end of a Gi,j sequence.

3. The default for “UM” should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for “UM”:

- The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).

- The components specified after “UM” must be a subset of the components specified under REFC and (Gi,j, Ci).

- The coefficient matrix $[R_m]$ described in“Multipoint Constraint Operations in SubDMAPs SEKR and SEMR2 and SEMRB” in the *NX Nastran User’s Guide* must be nonsingular. PARAM,CHECKOUT in SOLs 101 through 200 may be used to check for this condition.

4. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

5. Rigid elements, unlike MPCs, are not selected through the Case Control Section.

6. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

7. Rigid elements are ignored in heat transfer problems.

8. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

9. The formulation for the RBE3 element was changed in MSC.Nastran Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees of freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark 1 . The formulation prior to MSC.Nastran Version 70.7 may be obtained by setting SYSTEM(310)=1.

10. The RIGID case control command must be assigned to “LAGRAN” to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.

11. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

    The rigid element thermal strains are calculated by
    $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$.

    If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

REMARKS RELATED TO SOLS 601 AND 701:

1. All GMi and CMi are ignored.

2. No output of forces of multipoint constraint may be requested.

3. By default, a dependent degree-of-freedom may not be assigned as independent by a rigid link (from RBAR or RBE2 entry) or multipoint constraint (from MPC, RBAR, RBE2 or another RBE3 entry). An error message will be issued in such a case. However, if GENMPC=1 is specified in the NXSTRAT entry, a dependent degree-of-freedom of a multipoint constraint can be assigned as independent by a rigid link or another multipoint constraint.

4. ALPHA is ignored.

RCROSS

Cross-Power Spectral Density and Cross-Correlation Function Output

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RCROSS	SID	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID	

EXAMPLE:

RCROSS	20	DISP	50	2	STRESS	150	8	4	

FIELDS:

Field	Contents
SID	Case control RCROSS identification number for cross-power spectral density and cross-correlation functions. (Integer>0)
RTYPEi	Type of response quantity. At least one field must be selected. See remark 2. (Character or blank)
IDi	Element, grid, or scalar point identification number. (Integer > 0)
COMPi	Component code (item) identification number. See remark 3. (Integer > 0)
CURID	Curve identification number. See remark 5. (Integer > 0 or blank)

REMARKS:

1. This entry is required for computing the cross-power spectral density and cross-correlation functions. SID must be selected with the case control command (RCROSS=SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity; fields RTYPE2, ID2, and COMP2 represent the second response quantity.

2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

   If either RTYPE1 or RTYPE2 is blank, then the blank field takes the default from the defined field.

3. For elements, the item code COMPi represents a component of the element stress, strain or force and is described in Tables “Element Stress-Strain Item Codes” and “Element Force Item Codes”. For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function.

   For grid points, the item code is one of 1,2,3,4,5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar points, always use 1.

4. Elements defined as a laminate cannot be selected by the RCROSS entry. The RCROSSC entry should be used to request cross-power spectral density and cross-correlation function output for shell composites defined with the PCOMP

and PCOMPG property entries, and solid element composites defined with the PCOMPS property entry.

5. Field CURID is optional. It is for your convenience to identify the output by using a single index.

RCROSSC

Cross-Power Spectral Density and Cross-Correlation Function Output for Shell and Solid Element Composites

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis for shell and solid element composites.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RCROSSC	SID	RTYPE1	ID1	COMP1	PLY1				
		RTYPE2	ID2	COMP2	PLY2			CURID	

EXAMPLE:

RCROSSC	20	DISP	50	2					
		STRESS	150	8	2			4	

FIELDS:

Field	Contents
SID	RCROSS case control command identification number. (Integer>0)
RTYPEi	Response quantity. (Character; For default behavior, see Remark 2)
IDi	Element, grid, or scalar point identification number. (Integer > 0; No default)
COMPi	Component (item) code identification number. See Remark 3. (Integer > 0; No default)
PLYi	Ply number. (Integer ≥ 0; For default behavior, see Remark 4)
CURID	Optional curve identification number. See Remark 5. (Integer > 0; No default)

REMARKS:

1. The RCROSSC entry is used to request cross-power spectral density and cross-correlation function output for shell composites defined with the PCOMP and PCOMPG property entries, and solid element composites defined with the PCOMPS property entry.

2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

   If either the RTYPE1 or RTYPE2 field is blank, then the blank field defaults to the response quantity listed in the defined field.

3. For elements, the component (item) code COMPi represents a component of the element stress, strain or force as described in the “Element Stress (or Strain) Item Codes” and “Element Force Item Codes” tables. For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function.

   For grid points, the item code is one of 1,2,3,4,5, and 6, which represent T1, T2, T3, R1, R2, and R3, respectively.

   For scalar points, always use 1.

4. PLY1 and PLY2 cannot both be zero or blank. If it is desired to have both zero, use the RCROSS bulk entry.

For a non-composite element, grid point, or scalar point, specify PLYi = 0 or leave the PLYi field blank. For a non-composite element, grid point, or scalar point, if PLYi > 0, the ply number specification is ignored.

For a composite element, if PLYi = 0 or the PLYi field is blank or PLYi is greater than the actual number of plies, the ply number specification is ignored.

5. To identify the output with a single index, specify the index in the CURID field.

RELEASE

Superelement Boundary Grid Point Release

Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RELEASE	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

EXAMPLE:

RELEASE	15	456	3	7	11	2	156	9	
	152	162							

ALTERNATE FORMATS AND EXAMPLE:

RELEASE	SEID	C	G1	“THRU”	G2				
RELEASE	6	2	15	THRU	127				

RELEASE	SEID	C	“ALL”						
RELEASE	127	156	ALL						

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)

Field	Contents
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)
Gi	Grid point identification numbers. (Integer > 0, “THRU”, or “ALL”; For THRU option, G1 < G2.)

REMARKS:

1. A grid point referenced on this entry must be an exterior grid point of the superelement referenced on the entry.
2. In the first alternate format, all grid points in the sequence G1 through G2 are not required to be exterior grid points. Such grid points will collectively produce a warning message but will otherwise be ignored.
3. If the “ALL” option is used, all exterior grid points are released for the referenced degrees-of-freedom.
4. The RELEASE entry is applicable to only the superelement solution sequences (SOLs 101 through 200). It may not reference the residual structure (SEID=0).
5. This entry is not supported for partitioned superelements.

RFORCE

Rotational Force

Defines a static loading condition due to an angular velocity and/or acceleration.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB							

EXAMPLE:

RFORCE	2	5		-6.4	0.0	0.0	1.0	2	
	1.0								

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer ≥ 0; No default)
CID	Coordinate system defining the components of the rotation vector. See Remark 19. (Integer ≥ 0, Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real; No default)
R1, R2, R3	Rectangular components of the rotation vector $\vec{R}$. The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$; Defaults R1 = 1.0, R2 = 0.0, R3 = 0.0)

Field	Contents
METHOD	Method used to compute centrifugal forces due to angular velocity. For angular acceleration, see Remarks 6 and 7. (Integer = 1 or 2; Default = 1)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = –1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 18. (Integer; Default = 0)

REMARKS:

1. In Figure 17-1, the force vector at grid point Gi is given by

$$\left\{\vec{F}\right\}_i = \left[m\right]_i \left[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)\right]$$

where $[m]_i$ is a 3 × 3 translational mass matrix at grid point Gi.

$\left(A \cdot \vec{R}\right)$ and $\left(RACC \cdot \vec{R}\right)$ have the units of revolutions per unit time and revolutions per unit time squared, respectively. The software multiplies each by 2π to compute ω and α:

Angular velocity	$\vec{\omega} = 2\pi\left(A \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time}\right)$
Angular acceleration	$\vec{\alpha} = 2\pi\left(RACC \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time^2}\right)$

Note: The equation for $\vec{F}_i$ will have additional terms if the mass is offset and METHOD = 1 is selected.

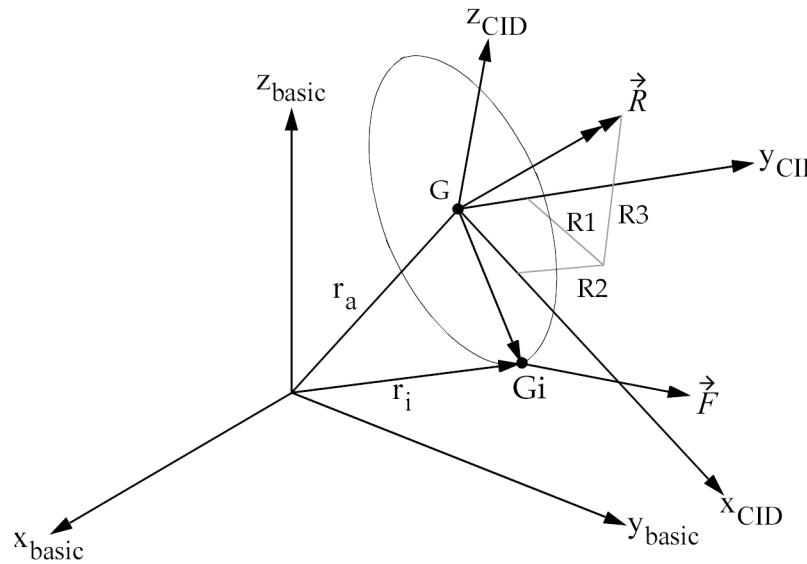


Figure 17-1. RFORCE Vector at Grid Point Gi

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. G = 0 signifies that the rotation vector acts through the origin of the basic coordinate system.

4. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.

5. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.

6. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	Offset
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

7. SOL 401 always uses METHOD = 2. As a result, the METHOD field is ignored by SOL 401.

8. In SOL 401, if a rotational force is defined with the RFORCE or RFORCE1 entry, and spin softening is requested with the SPINK parameter on the NLCNTL entry, if the rotation center is defined with a grid point which is not used in the connectivity of any element, all translational DOF for this grid point must be constrained.

9. In SOL 401, loads selected with the LOAD=n case control command are time unassigned. Time unassigned loads are linearly ramped in a subcase when the LVAR parameter on the NLCNTL entry is set to RAMP (default). The following rules apply to the time unassigned and ramped RFORCE and RFORCE1 entries selected in sequentially dependent subcase.

   - RFORCE and RFORCE1 entries cannot be defined in the same subcase.

   - If multiple RFORCE1 entries are defined in a subcase, the grids associated with each RFORCE1 entry must be mutually exclusive.

   - The rotational load vector direction defined with the R1, R2, R3 fields and the coordinate system selected with the CID field cannot change from subcase to subcase.

   - The scale factors A and RACC can change, and RFORCE and RFORCE1 entries can be added or removed, as long as the rotational load vectors and coordinate systems do not change.

10. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.

11. For superelement analysis, G should reference a residual structure point that is exterior to all superelements. If it is not exterior to a superelement, then centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.

12. In a geometric nonlinear static analysis (SOL 106 and 401 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.

13. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE input (and not the angular velocity vector) is scaled linearly. This

means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE loading is applied in a single increment.

14. The continuation entry is optional.

15. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.

16. Loads derived from this entry do not include effects due to mass specified for scalar points.

17. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter FOLLOWK). Follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

    For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

18. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.

19. If CID is not a rectangular coordinate system, RFORCE will treat it as if it were and unexpected answers may result.

20. Follower force stiffness (param,followk,yes) is supported for method 2 only.

21. In SOL 401, when RFORCE or RFORCE1 entries are referenced by the EXCITEID field on a TLOAD1 entry, the data on the associated TABLEDi, along with the scale factors S and Si on a DLOAD entry (if defined), scale the angular velocity (ω) and acceleration (α), which are used to compute an inertia force in the equation F = [m] [ω x (ω x r)) + α x r]. Since ω is squared in the force computation, the resulting scaling is not linearly related to the computed force (F). All other solutions, including SOL 601, scale the computed force (F).

REMARKS RELATED TO SOLS 601 AND 701:

1. METHOD and MB are ignored.

2. To apply rotational force with constant magnitude, SID is selected by Case Control command LOAD=SID for both static and transient analyses. For magnitude change due to large deformation, see Remark 4.

3. To apply a time-dependent rotational load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

4. By default, in large deformation analysis, the magnitude of the rotational force changes due to the deformation. The use of LOADOPT = 0 in NXSTRAT entry causes the rotational load to be independent of deformation.

5. Only one RFORCE can be applied in an analysis.

6. CID must be a rectangular coordinate system.

RFORCE1

Rotational Force with repeated SID

Defines a static loading condition due to an angular velocity and/or acceleration.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RFORCE1	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB	GROUPID						

EXAMPLE:

RFORCE1	2	5		-6.4	0.0	0.0	1.0	2	
	1.0								

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer ≥ 0; No default)
CID	Coordinate system defining the components of the rotation vector. See Remark 20. (Integer ≥ 0, Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real; No default)
R1, R2, R3	Rectangular components of rotation vector $\vec{R}$. The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$; Defaults R1 = 1.0, R2 = 0.0, R3 = 0.0)

Field	Contents
METHOD	Method used to compute centrifugal forces due to angular velocity. For angular acceleration, see Remarks 7 and 8. (Integer = 1 or 2; Default = 2)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = –1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 19. (Integer; Default = 0)
GROUPID	Group identification number. The GROUP entry referenced in the GROUPID field selects the grid points to which the load is applied. See Remark 23. (Integer > 0)

REMARKS:

1. The RFORCE and RFORCE1 entries have the following difference.
   - The RFORCE entry applies loads that result from rotational velocities and accelerations to all grid points.
   - The RFORCE1 entry applies loads that result from rotational velocities and accelerations to select grid points. The grid points are selected using a GROUP bulk entry.
2. In Figure 17-2, the force vector at a grid point is given by

$$\left\{\vec{F}\right\}_i = \left[m\right]_i \left[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)\right]$$

where $[m]_i$ is a 3 × 3 translational mass matrix at grid point Gi.

$\left(A \cdot \vec{R}\right)$ and $\left(RACC \cdot \vec{R}\right)$ have the units of revolutions per unit time and revolutions per unit time squared, respectively. The software multiplies each by 2π to compute ω and α:

Angular velocity	$\vec{\omega} = 2\pi\left(A \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time}\right)$
Angular acceleration	$\vec{\alpha} = 2\pi\left(RACC \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time^2}\right)$

Note: The equation for $\vec{F}_i$ will have additional terms if the mass is offset and METHOD = 1 is selected.

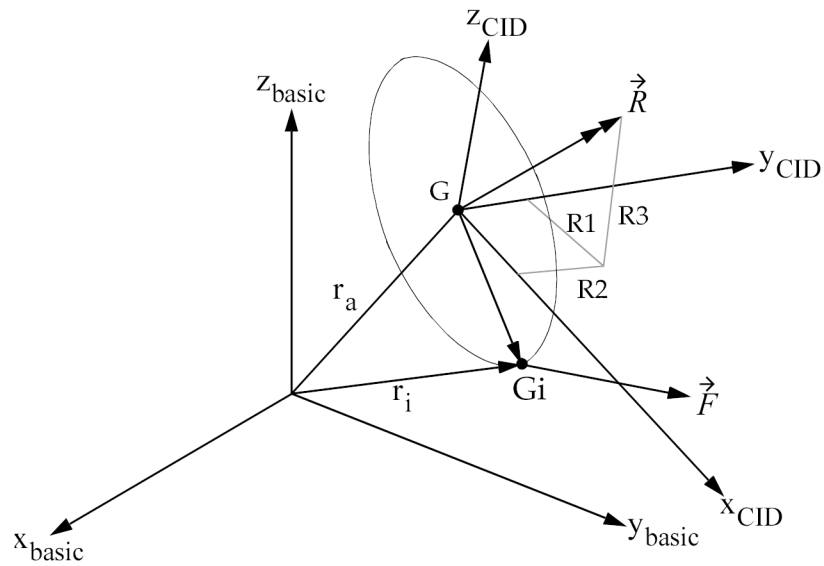


Figure 17-2. RFORCE1 Vector at a Grid Point

3. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

4. G = 0 signifies that the rotation vector acts through the origin of the basic coordinate system.

5. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.

6. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.

7. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	**Offset**
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

8. SOL 401 always uses METHOD = 2. As a result, the METHOD field is ignored by SOL 401.

9. In SOL 401, if a rotational force is defined with the RFORCE or RFORCE1 entry, and spin softening is requested with the SPINK parameter on the NLCNTL entry, if the rotation center is defined with a grid point which is not used in the connectivity of any element, all translational DOF for this grid point must be constrained.

10. In SOL 401, loads selected with the LOAD=n case control command are time unassigned. Time unassigned loads are linearly ramped in a subcase when the LVAR parameter on the NLCNTL entry is set to RAMP (default). The following rules apply to the time unassigned and ramped RFORCE and RFORCE1 entries selected in sequentially dependent subcase.

    - RFORCE and RFORCE1 entries cannot be defined in the same subcase.

    - If multiple RFORCE1 entries are defined in a subcase, the grids associated with each RFORCE1 entry must be mutually exclusive.

    - The rotational load vector direction defined with the R1, R2, R3 fields and the coordinate system selected with the CID field cannot change from subcase to subcase.

    - The scale factors A and RACC can change, and RFORCE and RFORCE1 entries can be added or removed, as long as the rotational load vectors and coordinate systems do not change.

11. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.

12. For superelement analysis, G should reference a residual structure point that is exterior to all superelements. If it is not exterior to a superelement, then centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.

13. In a geometric nonlinear static analysis (SOL 106 and 401 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.

14. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE1 input (and not the angular velocity vector) is scaled linearly. This means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE1 loading is applied in a single increment.

15. The continuation entry is optional.

16. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.

17. Loads derived from this entry do not include effects due to mass specified for scalar points.

18. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter FOLLOWK). Follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

    For SOL 401, follower force effects are included in the force balance in the nonlinear static solution if geometric nonlinear effects are turned on with PARAM,LGDISP,1. For additional information, see the NLCNTL bulk entry.

19. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.

20. If CID is not a rectangular coordinate system, RFORCE1 will treat it as if it were and unexpected answers may result.

21. Follower force stiffness (param,followk,yes) is supported for method 2 only.

22. In SOL 401, when RFORCE or RFORCE1 entries are referenced by the EXCITEID field on a TLOAD1 entry, the data on the associated TABLEDi, along with the scale factors S and Si on a DLOAD entry (if defined), scale the angular velocity (ω) and acceleration (α), which are used to compute an inertia force in the equation F = [m] [ω x (ω x r)) + α x r]. Since ω is squared in the force computation, the resulting scaling is not linearly related to the computed force (F). All other solutions, including SOL 601, scale the computed force (F).

23. Grid points are either listed on the GROUP entry or are related to elements or properties listed on the GROUP entry as follows:

    - If TYPE = “GRID” on the GROUP entry, the GROUP entry lists the grid points to which the load is applied.
    - If TYPE = “ELEM” on the GROUP entry, the GROUP entry contains a list of elements. The load is applied to the grid points used in the connectivity of the listed elements.
    - If TYPE = “PROP” on the GROUP entry, the GROUP entry contains a list of properties. The load is applied to the grid points used in the connectivity of elements that reference the listed properties.

REMARKS RELATED TO SOLS 601 AND 701:

1. METHOD and MB are ignored.
2. To apply rotational force with constant magnitude, SID is selected by Case Control command LOAD = SID for both static and transient analyses. For magnitude change due to large deformation, see Remark 4.
3. To apply a time-dependent rotational load, SID is referenced by the field EXCITEID = SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.
4. By default, in large deformation analysis, the magnitude of the rotational force changes due to the deformation. The use of LOADOPT = 0 in NXSTRAT entry causes the rotational load to be independent of deformation.
5. CID must be a rectangular coordinate system.
6. Elements which are not selected with your GROUP bulk entry will not contribute to the resulting grid point mass used to compute the load. For example, if a grid point is used by elements A and B, but only element A is included in the GROUP, the mass of element A contributes to the grid point mass used to compute the RFORCE1 load. The mass of element B does not contribute. This behavior is unique to solutions 601 and 701. Using the same example, the other NX Nastran

solutions use the mass contribution of both elements A and B when computing the RFORCE1 load.

RFORCE2

Rotational Force for SOL 101 Rotor Dynamic Analysis

Defines a static loading condition due to an angular velocity and/or angular acceleration for maneuver load rotor dynamic analysis. Valid for SOL 101 only.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RFORCE2	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB	GYROP						

EXAMPLE:

RFORCE2	2	5		–6.4	0.0	0.0	1.0	2	
	1.0		1						

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. See Remark 5. (Integer ≥ 0; Default = 0)
CID	Coordinate system defining the components of the rotation vector. See Remarks 6 and 7. (Integer ≥ 0, Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real; Default = 1.0)
R1, R2, R3	Dimensionless rectangular components of the rotation vector that passes through grid point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$; Defaults R1 = 1.0, R2 = 0.0, R3 = 0.0)

Field	Contents
METHOD	Method used to compute centrifugal forces. See Remarks 8 and 9. (Integer = 1 or 2; Default = 2)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section or the partitioned superelement Bulk Data Section. Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 10. (Integer = –1 or 0; Default = 0) = –1 if the CID coordinate system is defined in the main Bulk Data Section = 0 if the CID coordinate system is defined in the partitioned superelement Bulk Data Section
GYROP	Force calculation option. (Integer = 0 or 1 or 2; Default = 0) = 0 to include centrifugal and gyroscopic force terms = 1 to include centrifugal force terms only = 2 to include gyroscopic force terms only

REMARKS:

1. If RFORCE2 entries are used in any analysis other than a SOL 101 rotor dynamic analysis, an error message is issued and the run is terminated.

2. In Figure 17-3, the force vector at a grid point Gi is given by:

$$\left\{\vec{F}\right\}_i = \left[m\right]_i \left[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)\right]$$

where $[m]_i$ is a 3 × 3 translational mass matrix at grid point Gi.

The angular velocity and angular acceleration vectors are given by:

Angular velocity	$\vec{\omega} = 2\pi\left(A \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time}\right)$
Angular acceleration	$\vec{\alpha} = 2\pi\left(RACC \cdot \vec{R}\right) = \left(\dfrac{radians}{unit\,time^2}\right)$

where *A* is the angular velocity scale factor, *RACC* is the angular acceleration scale factor, and $\vec{R}$ is the dimensionless rotation vector.

The equation for $\vec{F}_i$ will have additional terms if the mass is offset and METHOD = 1 is selected.

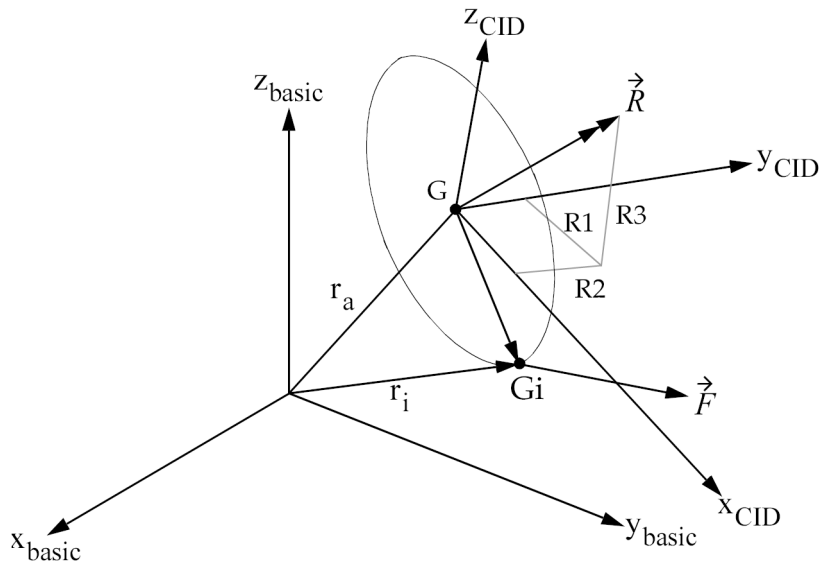


Figure 17-3. RFORCE2 Vector at a Grid Point

3. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

4. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.

5. G = 0 (Default) signifies that the rotation vector acts through the origin of the basic coordinate system.

6. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.

7. If CID is not a rectangular coordinate system, RFORCE2 will treat it as if it were and unexpected answers may result.

8. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	**Offset**
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

9. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.

10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.

11. For superelement analysis, G should reference a residual structure point that is exterior to all superelements. If it is not exterior to a superelement, then centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.

12. The continuation entry is optional.

13. Loads derived from this entry do not include effects due to mass specified for scalar points.

14. To model angular motion where the angular velocity and angular acceleration vectors have different directions, use two RFORCE2 entries that have the same SID. Use one entry to define the angular velocity vector, and use the other entry to define the angular acceleration vector.

RLOAD1

Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form:

$$\{P(f)\} = \{A[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}\}$$

for use in frequency response problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	EXCITEID	DELAY	DPHASE	TC	TD	TYPE		

EXAMPLES:

RLOAD1	5	3	100	30.0	1				

RLOAD1	5	3	0.025	4	1	10			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of a DAREA entry, SPCD entry, SELOAD entry, static load set, or thermal load set (for heat transfer analysis) that lists each degree of freedom to apply the excitation and the corresponding scale factor, *A*, for the excitation. See Remark 7 and Remark 8. (Integer > 0)
DELAY	Time delay, τ. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of τ for all degrees of freedom in EXCITEID entry.

Field	Contents
	If integer entry, identification number of a DELAY entry that contains values of τ for all degrees of freedom in EXCITEID entry. See Remark 3.
DPHASE	Phase angle, θ, in degrees. See Remark 10. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of θ for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a DPHASE entry that contains values of θ for all degrees of freedom in EXCITEID entry. See Remark 3.
TC	*C(f)* (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of *C(f)* used over all frequencies for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a TABLEDi entry that defines *C(f)* for all degrees of freedom in EXCITEID entry.
TD	*D(f)* (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of *D(f)* used over all frequencies for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a TABLEDi entry that defines *D(f)* for all degrees of freedom in EXCITEID entry.
TYPE	Defines the type of the dynamic excitation. See Remark 7 and Remark 8. (Integer, character or blank; Default = 0)

REMARKS:

1. Dynamic excitation sets must be selected with DLOAD = SID in the case control section.

2. If any of DELAY, DPHASE, TC, and TD fields are blank or zero (either integer zero or real zero), the corresponding value for τ, θ, *C(f)*, and *D(f)* used by the software is real zero.

3. For degrees of freedom in the EXCITEID entry that are not specified on the DELAY entry, the software uses real zero as the value for τ. For degrees of

freedom in the EXCITEID entry that are not specified on the DPHASE entry, the software uses real zero as the value for θ.

4. RLOAD1 excitations may be combined with RLOAD2 excitations using a DLOAD bulk entry.

5. SID must be unique for all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries.

6. If ACSRCE and RLOADi entries are combined with a DLOAD bulk entry, the identification numbers of the TABLEDi selected with the TP field on the ACSRCE entries must be different from the identification numbers of the TABLEDi selected with the TC and TD fields on RLOAD1 entries, and the TB and TP fields on RLOAD2 entries.

7. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

8. TYPE determines the manner in which EXCITEID is used by the program.

   - TYPE = 0 : Applied load excitation.

     EXCITEID may reference DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, PLOAD1, QHBDY, QBDYi, QVECT, QVOL, and SELOAD entries.

   - TYPE = 1, 2, or 3 : Enforced motion excitation.

     - o If EXCITEID references SPC/SPCD entries, the software will use the SPCD method of enforced motion. You directly specify displacements, velocities, or accelerations with SPC/SPCD entries.

     - o If EXCITEID references a load entry (DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entries), the software will use the large mass method of enforced motion. You create a large mass and load at the grid and degree of freedom where the enforced motion is desired. The software then computes the enforced motion the same for TYPE = 1, 2, and 3.

Type = 1, Enforced displacement = Dynamic load input / Large mass
Type = 2, Enforced velocity = Dynamic load input / Large mass
Type = 3, Enforced acceleration = Dynamic load input / Large mass

See the “Enforced Motion” chapter of the *NX Nastran Basic Dynamic Analysis User’s Guide* for details on both the SPCD and large mass methods of enforced motion.

9. The legacy method of selecting a DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entry for a dynamic loading required the LOADSET case control command which selected the LSEQ bulk entry, which in turn selected the load entry. Because the DAREA, FORCEi, MOMENTi, PLOAD, PLOAD1, and SPCD entries are now selected directly with the EXCITEID on the dynamic load entries DLOADi, TLOADi, and RLOADi, the LOADSET case control and LSEQ bulk entry are no longer required. The legacy method is still supported, and is described on the LSEQ entry remarks.

10. The software converts the phase angle, θ, to radians.

RLOAD2

Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form:

$$\{P(f)\} = \{A \cdot B(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}\}$$

for use in frequency response problems.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	EXCITEID	DELAY	DPHASE	TB	TP	TYPE		

EXAMPLES:

RLOAD2	5	3	0.2	15.0	7				

RLOAD2	5	3		6	7	8			

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of a DAREA entry, SPCD entry, SELOAD entry, static load set, or thermal load set (for heat transfer analysis) that lists each degree of freedom to apply the excitation and the corresponding scale factor, *A*, for the excitation. See Remark 7 and Remark 8. (Integer > 0)
DELAY	Time delay, *τ*. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2)

Field	Contents
	If real entry, value of τ for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a DELAY entry that contains values of τ for all degrees of freedom in EXCITEID entry. See Remark 3.
DPHASE	Phase angle, θ, in degrees. See Remark 10. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of θ for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a DPHASE entry that contains values of θ for all degrees of freedom in EXCITEID entry. See Remark 3.
TB	*B(f)* (Real or Integer ≥ 0; No default) If real entry, value of *B(f)* used over all frequencies for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a TABLEDi entry that defines *B(f)* for all degrees of freedom in EXCITEID entry.
TP	*φ(f)* in degrees. See Remark 10. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 2) If real entry, value of *φ(f)* used over all frequencies for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a TABLEDi entry that defines *φ(f)* for all degrees of freedom in EXCITEID entry.
TYPE	Defines the type of the dynamic excitation. See Remark 7 and Remark 8. (Integer, character or blank; Default = 0)

REMARKS:

1. Dynamic excitation sets must be selected with DLOAD = SID in the case control section.

2. If any of DELAY, DPHASE, and TP fields are blank or zero (either integer zero or real zero), the corresponding value for τ, θ, and *φ(f)* used by the software is real zero.

3. For degrees of freedom in the EXCITEID entry that are not specified on the DELAY entry, the software uses real zero as the value for τ. For degrees of freedom in the EXCITEID entry that are not specified on the DPHASE entry, the software uses real zero as the value for θ.

4. RLOAD2 excitations may be combined with RLOAD1 excitations using a DLOAD bulk entry.

5. SID must be unique for all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries.

6. If ACSRCE and RLOADi entries are combined with a DLOAD bulk entry, the identification numbers of the TABLEDi selected with the TP field on the ACSRCE entries must be different from the identification numbers of the TABLEDi selected with the TC and TD fields on RLOAD1 entries, and the TB and TP fields on RLOAD2 entries.

7. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

8. TYPE determines the manner in which EXCITEID is used by the program.
   - TYPE = 0 : Applied load excitation.

     EXCITEID may reference DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, PLOAD1, QHBDY, QBDYi, QVECT, QVOL, and SELOAD entries.

   - TYPE = 1, 2, or 3 : Enforced motion excitation.
     - o If EXCITEID references SPC/SPCD entries, the software will use the SPCD method of enforced motion. You directly specify displacements, velocities, or accelerations with SPC/SPCD entries.
     - o If EXCITEID references a load entry (DAREA, FORCEi, MOMENTi, PLOAD, and PLOAD1 entries), the software will use the large mass method of enforced motion. You create a large mass and load at the grid and degree-of-freedom where the enforced motion is desired. The software then computes the enforced motion the same for TYPE = 1, 2, and 3.

Type = 1, Enforced displacement = Dynamic load input / Large mass
Type = 2, Enforced velocity = Dynamic load input / Large mass
Type = 3, Enforced acceleration = Dynamic load input / Large mass

See the Enforced Motion chapter in the *NX Nastran Basic Dynamic Analysis User's Guide* for details on both the SPCD and large mass methods of enforced motion.

9. The legacy method of selecting a DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entry for a dynamic loading required the LOADSET case control command which selected the LSEQ bulk entry, which in turn selected the load entry. Since the FORCEi, MOMENTi, PLOAD, PLOAD1, and SPCD entries are now selected directly with the EXCITEID on the dynamic load entries DLOADi, TLOADi, and RLOADi, the LOADSET case control and LSEQ bulk entry are no longer required. The legacy method is still supported, and is described on the LSEQ entry remarks.

10. The software converts the phase angle, θ, and frequency-dependent function, *φ(f)*, to radians.

ROTORB

Bearing Grids Selection

Selects the stationary grid from each pair of grids that define the connectivity of bearing supports for a specific rotor.

FORMAT 1: (FORMATS 1 AND 2 CANNOT BE COMBINED ON THE SAME LINE)

1	2	3	4	5	6	7	8	9	10
ROTORB	RSETID	G1	G2	G3	G4	G5	G6	G7	

FORMAT 2:

ROTORB	RSETID	G1	“THRU”	G2	“BY”	INC			

CONTINUATION FORMAT 1: (CONTINUATION FORMATS 1 AND 2 CANNOT BE COMBINED

ON A SINGLE CONTINUATION LINE)

	G8	G9	G10	G11	–etc.–				

CONTINUATION FORMAT 2:

	G3	“THRU”	G4	“BY”	INC				

EXAMPLE:

ROTORB	14	101	THRU	190	BY	5			
	46	23	57	82	9	16			
	201	THRU	255						
	93	94	95	97					

ROTORB

FIELDS:

Field	Contents
RSETID	References an RSETi on the ROTORD bulk entry. (Integer>0)
Gi	Grids defining bearing locations on a rotor. (Integer>0)
THRU	Specifies a range of grid ID's. (Optional)
BY	Specifies an increment when using THRU option. (Optional)
INC	Increment used with BY option. (Integer>0; Default=1)

REMARKS:

1. By default, there are no bearing locations on a rotor. The RSETID field is referred to by the RSETi field on the ROTORD bulk data entry. If a model contains multiple rotors, use a separate ROTORB entry for each rotor.

ROTORD

Define Rotor Dynamics Solution Options

Defines rotor dynamics solution options.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ROTORD	SID	RSTART	RSTEP	NUMSTEP	REFSYS	CMOUT	RUNIT	FUNIT	
	ZSTEIN	ORBEPS	ROTPRT	SYNC	ETYPE	EORDER	THRSHOLD	MAXITER	
	RID1	RSET1	RSPEED1	RCORD1	W3_1	W4_1	RFORCE1	BRGSET1	
	RID2	RSET2	RSPEED2	RCORD2	W3_2	W4_2	RFORCE2	BRGSET2	
....									
	RIDi	RSETi	RSPEEDi	RCORDi	W3_i	W4_i	RFORCEi	BRGSETi	
....									
	RID10	RSET10	RSPEED10	RCORD10	W3_10	W4_10	RFORCE10	BRGSET10	

EXAMPLE:

ROTORD	998	0.0	250.0	58	fix	-1.0	cps		
	no								
	1	11		1	0.0	0.0	1	101	
	2	12		1	0.0	0.0		102	
	3	13	1.5	1	0.0	0.0		103	
	4	14	1.75	1	0.0	0.0		104	
	5	15	1.75	1	0.0	0.0		105	
	6	16		1	0.0	0.0		106	
	7	17	2.0	1	0.0	0.0		107	
	8	18	2.25	1	0.0	0.0		108	
	9	19	7.5	1	0.0	0.0		109	
	10	20		1	0.0	0.0	10	110	

FIELDS:

Field	Contents
SID	Set identifier for all rotors. Must be selected in the case control deck by RMETHOD = SID. (Integer > 0)
RSTART	Starting value of reference rotor speed. See Remark 2 and Remark 4. (Real)
RSTEP	Step-size of reference rotor speed. See Remark 3. (Real ≠ 0.0)
NUMSTEP	Number of steps for reference rotor speed including RSTART. See Remark 14. (Integer ≥ 0 for SOLs 107 and 110, Integer > 0 for all other supported SOLs)
REFSYS	Reference system. (Character; Default = ‘ROT’)
	= ‘FIX’ analysis is performed in the fixed reference system.
	= ‘ROT’ analysis is performed in the rotational reference system.
CMOUT	Determines the rotor speeds at which eigenvectors are calculated and output. (Real; Default = 0.0)
	= 0.0 no eigenvectors are calculated; no eigenvectors are output; no whirl directions are output.
	> 0.0 eigenvectors are calculated at the specified speed; eigenvectors are output at the specified speed; whirl direction is output at the specified speed.
	= -1.0 eigenvectors are calculated at all speeds; eigenvectors are output at all speeds; whirl directions are output at all speeds.
RUNIT	Units used for rotor speed inputs (CMOUT, RSTART and RSTEP), output (units for output list and Campbell diagram output), and speed-dependent bearing property tables. (Character; Default = ‘RPM’)
	= ‘RPM’ revolutions per minute.
	= ‘CPS’ cycles per second.
	= ‘HZ’ cycles per second.
	= ‘RAD’ radians per second.

Field	Contents
FUNIT	Units used for frequency output (Campbell diagram output). (Character; Default = ‘RPM’)
	= ‘RPM’ revolutions per minute.
	= ‘CPS’ cycles per second.
	= ‘HZ’ cycles per second.
	= ‘RAD’ radians per second.
ZSTEIN	Option to incorporate Steiner’s inertia terms. (Character; Default = 'NO')
	= 'YES' Steiner’s inertia terms are included.
	= 'NO' Steiner’s inertia terms are not included.
ORBEPS	Threshold value for detection of whirl direction. (Real > 0.0; Default = 1.E-6)
ROTPRT	Controls .f06 output options. (Integer; Default = 0))
	= 0 no print.
	= 1 print generalized matrices; print final nonlinear bearing values at each frequency or time.
	= 2 print eigenvalue summary and eigenvectors at each RPM; print intermediate nonlinear bearing values for each iteration.
	= 3 combination of 1 & 2.
SYNC	Option to select synchronous or asynchronous analysis for frequency response analysis. (Integer; Default = 1)
	= 1 synchronous
	= 0 asynchronous
ETYPE	Excitation type. (Integer; Default = 1)
	= 1 Mass unbalanced. Specify mass unbalance = m x r on DLOAD bulk entry and the program will multiply by Ω^2.
	= 0 Force excitation. Specify force = m x r x Ω^2 on DLOAD bulk entry.

Field	Contents
EORDER	Excitation order. (Real; Default 1.0) = 1.0 (Default) (modes crossing with 1P line in the fixed system) = 0.0 Forward whirl (modes crossing with 0P line in the rotating system) = 2.0 Backward whirl (modes crossing with 2P line in the rotating system)
THRSHOLD	Convergence threshold when iterating to determine bearing stiffness or viscous damping for CBEAR elements that have speed and displacement or speed and force dependent stiffness or viscous damping. See Remark 15. (Real > 0.0; Default = 0.02)
MAXITER	Maximum number of iterations to determine bearing stiffness or viscous damping for CBEAR elements that have speed and displacement or speed and force dependent stiffness or viscous damping. A value of 0 implies that no iterations are performed. See Remark 15. (Integer ≥ 0; Default = 10)
RIDi	Identification number of rotor i. (Integer > 0 with RID(i+1) > RIDi; Default = i)
RSETi	Refers to the RSETID value on the ROTORG, ROTORB, and ROTSE bulk entries for rotor RIDi. (Integer > 0 or blank if only one rotor)
RSPEEDi	Multiplier of reference rotor speed for rotor i. (Real ≠ 0.0 or Integer > 0 or blank; Default = 1.0) If real entry, value of multiplier at all reference rotor speeds. If integer entry, identification number of a TABLEDi entry that contains value of multiplier as a function of reference rotor speed.
RCORDi	Identification number of the Cartesian coordinate system whose Z-axis is the axis of rotation axis for rotor i. (Integer; Default = 0 for basic coordinate system)
W3_i	Reference frequency for structural damping defined by PARAM,G for rotor i. (Real; Default = 0.0)
W4_i	Reference frequency for structural damping defined by GE for rotor i. See Remark 8. (Real; Default = 0.0)

Field	Contents
RFORCEi	Points to RFORCE, RFORCE1, or RFORCE2 bulk entry for rotor i. (Integer; Default = 0 for no rotational force applied; a rotational force is required for differential stiffness to be calculated.)
BRGSETi	Identification number of a GROUP bulk entry that lists the CBEAR elements for the corresponding RIDi. Only the GROUP type ELEM is supported. (Integer ≥ 0 or blank; no default)

REMARKS:

1. There is a maximum limit of 10 rotors (i.e. 11 continuation lines).

2. The rotation direction for a rotor depends on the algebraic sign of the numerical values for RSTART and RSPEEDi as follows:

RSTART	RSPEEDi(1)	Direction of rotation for RIDi(2)
> 0	> 0	Positive
> 0	< 0	Negative
< 0	> 0	Negative
< 0	< 0	Positive

(1)If the RSPEEDi field references a TABLEDi bulk entry, the numerical value the software uses from the TABLEDi lookup.
(2)The direction of rotation is defined in accordance with the right-hand rule. *Positive* indicates rotation about the +Z-axis of the RCORDi coordinate system. *Negative* indicates rotation about the –Z-axis of the RCORDi coordinate system.

3. A negative value for RSTEP causes the reference rotor speed to algebraically decrease. For example, assume RSTEP = –200 rpm. If the current reference rotor speed is +1200 rpm, the reference rotor speed will be +1000 rpm after one step. After five additional steps the reference rotor speed would be 0 rpm. Addtional steps would then cause the reference rotor speed to increase in the opposite rotational direction.

4. When the ROTCOUP parameter is specified and REFSYS = ROT, the equation of motion includes time-dependent terms and is solved at discrete azimuth angles. The software can either solve the equation of motion over a range of azimuth angles at a single rotor speed, or solve the equation of motion at a single azimuth angle over a range of rotor speeds.

- To solve over a range of azimuth angles at a single rotor speed, use the PHIBGN, PHIDEL, and PHINUM parameters to specify the azimuth angle range, and use the RSTART field to specify the rotor speed.

- To solve over a range of rotor speeds at a single azimuth angle, use the PHIBGN parameter to specify the azimuth angle, omit the PHIDEL and PHINUM parameters, and use the RSTART, RSTEP, and NUMSTEP fields to specify the rotor speed range. If you also omit the PHIBGN parameter, the solve is at an azimuth angle of zero because the default value for the PHIBGN parameter is zero.

5. The Steiner's term option (ZSTEIN) should only be used when analyzing solid models in the fixed system.

6. The W3 parameter defines the reference frequency for structural damping defined by PARAM,G.

7. The W4 parameter defines the reference frequency for structural damping defined by GE.

8. Depending on element type, GE is specified on either material bulk entries, property bulk entries, or on the element bulk entries themselves.

9. The W3 and W4 parameters are required for all direct solutions. In the modal solutions, the eigenvalues are used as default. If the W3 and W4 parameters are defined for modal analysis, they will be used, but these parameters are not recommended for modal solutions.

10. The static centrifugal force is calculated for unit speed measured in rad/sec. On the RFORCE, RFORCE1, or RFORCE2 bulk entry, the unit of Hz is used, thus the conversion $1/(2\pi) = 0.159155$ must be used by the user.

11. For calculating frequency response using synchronous analysis, the rotation speeds are defined by the RSTART, RSTEP, and NUMSTEP fields on the ROTORD bulk entry. The frequencies corresponding to these rotation speeds must match the frequencies specified with the FREQUENCY case control command. If the frequencies differ, a fatal error is issued.

12. For calculating the frequency or transient response using asynchronous analysis, the unique rotation speed is defined by the RSTART field on the ROTORD bulk entry. The RSTEP and NUMSTEP fields in this case will be ignored. The frequency and dynamic load definitions are defined with the standard FREQ, DLOAD, RLOAD, etc. bulk entries for frequency response, and with the TSTEP, DLOAD, TLOAD, etc. bulk entries for transient response.

13. The rotor speed defined by RSTART, RSTEP, and NUMSTEP is called the reference rotor speed. Rotors with relative speed defined by RSPEED will spin at the defined factor multiplied by the reference rotor speed.

14. The SYNC, ETYPE, and EORDER fields are ignored during a SOL 107 or SOL 110 complex eigenvalue analysis. During a SOL 107 or SOL 110 complex eigenvalue analysis, if NUMSTEP = 0 and PARAM,ROTSYNC,YES (default) are specified, only synchronous analysis is performed. If NUMSTEP = 0 and PARAM,ROTSYNC,NO are specified, no analysis is performed.

15. For additional information on how the values in the THRSHOLD and MAXITER fields are used, see the PBEARbulk entry.

ROTORG

Rotor Grids Selection

Selects the grids that define a rotor.

FORMAT 1: (FORMATS 1 AND 2 CANNOT BE COMBINED ON THE SAME LINE)

1	2	3	4	5	6	7	8	9	10
ROTORG	RSETID	G1	G2	G3	G4	G5	G6	G7	

FORMAT 2:

ROTORG	RSETID	G1	“THRU”	G2	“BY”	INC			

CONTINUATION FORMAT 1: (CONTINUATION FORMATS 1 AND 2 CANNOT BE COMBINED ON A SINGLE CONTINUATION LINE)

	G8	G9	G10	G11	–etc.–				

CONTINUATION FORMAT 2:

	G3	“THRU”	G4	“BY”	INC				

EXAMPLE:

ROTORG	14	101	THRU	190	BY	5			
	46	23	57	82	9	16			
	201	THRU	255						
	93	94	95	97					

FIELDS:

Field	Contents
RSETID	References an RSETi on the ROTORD bulk entry. (Integer>0)
Gi	Grids defining a rotor. (Integer>0)
THRU	Specifies a range of grid ID's. (Optional)
BY	Specifies an increment when using THRU option. (Optional)
INC	Increment used with BY option. (Integer>0; Default=1)

REMARKS:

1. For a rotor dynamic analysis input file with ROTORG entries present, the software assumes grids listed on the ROTORG entries are rotating and grids not listed on the ROTORG entries are stationary. For a rotor dynamic analysis input file without ROTORG entries present, the software assumes all grids are rotating.

2. If a model contains both rotating and stationary portions, list the grids associated with each rotor on a unique ROTORG entry. On the ROTORD entry, have each RSETi field reference the RSETID field of the corresponding ROTORG entry.

3. If an entire model is rotating, you can do either of the following:

   - Include a ROTORG entry that contains all the grids and have the RSET1 field on the ROTORD entry reference the RSETID field of the ROTORG entry.

   - Do not include a ROTORG entry and leave the RSET1 field blank on the ROTORD entry.

ROTPARM

Parameter Specification for Rotor Dynamic Analysis

Defines solution control parameters for frequency response and complex eigenvalue rotor dynamic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
ROTPARM	SID	PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3		
	PARAM4	VALUE4	PARAM5	VALUE5	-etc-				

EXAMPLE:

ROTPARM	1	SEFILT	YES	SETHRSH	20.0	KETHRSH	15.0		
	REFROT	1							

FIELDS:

Field	Contents
SID	Set identification number. See Remark 1. (Integer > 0)
PARAMi	Parameter name. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. (Real, Integer, or Character)

ROTPARM PARAMETERS:

Name	Description
ROTSEKE	Print energy calculations. (Character “YES” or “NO”; Default = “NO”)
REFROT	Identification number (RIDi on the ROTORD bulk entry) of the reference rotor. The reference rotor is used as the rotor whose strain or kinetic energy is used in the calculation of ratio of energy of the rotor to total energy of the system, and is used as the reference rotor for rotor speed calculations. See Remark 2. (Integer > 0; For default behavior, see Remark 3)
SEFILT	Identify modes that have a ratio of strain energy of the reference rotor to total strain energy of the system that exceeds the strain energy threshold value. See Remarks 4, 5, and 6. (Character “YES” or “NO”; Default = “NO”)
KEFILT	Identify modes that have a ratio of kinetic energy of the reference rotor to total kinetic energy of the system that exceeds the kinetic energy threshold value. See Remarks 4, 5, and 6. (Character “YES” or “NO”; Default = “NO”)
SETHRSH	Strain energy threshold value in percent. (Real > 0.0; Default = 30.0)
KETHRSH	Kinetic energy threshold value in percent. (Real > 0.0; Default = 30.0)

REMARKS:

1. The SID of the ROTPARM bulk entry must be the same as the SID of the ROTORD bulk entry.

2. If the REFROT field is not specified, to obtain meaningful results, on the ROTORD bulk entry, the RSPEEDi field for the reference rotor must be set to 1.0. The software does not trap other values.

3. For a system with a single rotor, the single rotor is the default reference rotor for the energy calculations in complex eigenvalues. For a system with multiple rotors, there is no default reference rotor.

4. If only SEFILT is specified, modes that pass the SEFILT criterion are identified by “***” in the summary line of the results. If only KEFILT is specified, modes that pass the KEFILT criterion are identified by “***” in the summary line of the

results. If both SEFILT and KEFILT are specified, modes that pass either the SEFILT criterion or KEFILT criterion are identified by “***” in the summary line of the results.

5. When either or both SEFILT and KEFILT are specified, and MODTRK = 4 is specified, only the modes identified by “***” in the summary line of the results are tracked for use in a Campbell diagram.

6. Energy-based filtering of modes is valid for SOL 107 and SOL 110 complex eigenvalue rotor dynamic analysis only.

ROTSE

Supplemental Rotor Superelement Definition

Defines the modal reduction type and additional a-set grids for a rotor superelement. Valid for SOLs 107, 108, and 109.

FORMAT 1: (FORMATS 1 AND 2 CANNOT BE COMBINED ON THE SAME LINE)

1	2	3	4	5	6	7	8	9	10
ROTSE	RSETID	TYPE	EVID						
	G1	G2	G3	G4	G5	G6	G7	G8	

FORMAT 2:

ROTSE	RSETID	TYPE	EVID						
	G1	“THRU”	G2	“BY”	INC				

CONTINUATION FORMAT 1: (CONTINUATION FORMATS 1 AND 2 CANNOT BE COMBINED ON THE SAME LINE)

	G9	G10	G11	G12	–etc.–				

CONTINUATION FORMAT 2:

	G3	“THRU”	G4	“BY”	INC				

EXAMPLE:

ROTSE	5	CX	1001						
	101	THRU	190	BY	5				
	46	23	57	82	9	16			

	201	THRU	255						
	93	94	95	97					

FIELDS:

Field	Contents
RSETID	References an RSETi on the ROTORD bulk entry. See Remark 1. (Integer>0)
TYPE	Modal reduction type. (Character: “RL”, “CX”)
	= “RL”, use real modal reduction.
	= “CX”, use complex modal reduction.
EVID	Eigenvalue extraction data set identification number. (Integer>0)
	For TYPE = “RL”, set identification number of an EIGRL bulk entry.
	For TYPE = “CX”, set identification number of an EIGC bulk entry.
Gi	Grids to remove from the o-set and place in the a-set. See Remark 2. (Integer > 0)
THRU	Specifies a range of grid ID's. (Optional)
BY	Specifies an increment when using THRU option. (Optional)
INC	Increment used with THRU option. (Integer > 0; Default=1)

REMARKS:

1. The RSETID field is referred to by the RSETi field on the ROTORD bulk entry. If a model contains multiple rotors, use separate ROTSE bulk entries for each rotor.
2. In a rotor dynamic analysis, the a-set consists of any grids that are not listed on any ROTORG bulk entry and any grids that are listed on any ROTSE bulk entry.
3. If the a-set does not contain at least one unconstrained DOF, the run terminates.

RROD

Rigid Pin-Ended Element Connection

Defines a pin-ended element that is rigid in translation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RROD	EID	GA	GB	CMA	CMB	ALPHA			

EXAMPLE:

RROD	14	1	2	2					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers of connection points. (Integer > 0)
CMA, CMB	Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank.)
ALPHA	Thermal expansion coefficient.

REMARKS:

1. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. Rigid elements, unlike MPCs, are not selected through the Case Control Section.

4. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

5. Rigid elements are ignored in heat transfer problems.

6. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the element. This implies that the element must have finite length.

7. See R-Type Elements in the*NX Nastran Element Library* for a discussion of rigid elements.

8. The RIGID case control command must be assigned to “LAGRAN” to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.

9. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

   The rigid element thermal strains are calculated by
   $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$.

   If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

RSPLINE

Interpolation Constraint Element

Defines multipoint constraints for the interpolation of displacements at grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

EXAMPLE:

RSPLINE	73	.05	27	28	123456	29		30	
	123	75	123	71					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
D/L	Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default=0.1)
Gi	Grid point identification number. (Integer > 0)
Ci	Components to be constrained. See Remark 2 . (Blank or any combination of the Integers 1 through 6.)

REMARKS:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points.

2. A blank field for Ci indicates that all six degrees-of-freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a Gi, not a Ci. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.

3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.

5. EIDs must be unique.

6. Rigid elements (including RSPLINE), unlike MPCs, are not selected through the Case Control Section.

7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

8. Rigid elements are ignored in heat transfer problems.

9. See R-Type Elements in the *NX Nastran Element Library* for a discussion of rigid elements.

10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

11. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results.

12. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.

13. The RSPLINE has a limit of 100 grid points.

RSSCON

Shell-to-Solid Element Connector

Defines multipoint constraints to model clamped connections of shell-to-solid elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RSSCON	RBID	TYPE	ES1	EA1	EB1	ES2	EA2	EB2	

EXAMPLE:

RSSCON	110	GRID	11	12	13	14	15	16	

RSSCON	111	GRID	31	74	75				

RSSCON	115	ELEM	311	741					

FIELDS:

Field	Contents
RBID	Element identification number. (Integer > 0)
TYPE	Type of connectivity TYPE=“ELEM” connection is described with element identification numbers. TYPE=“GRID” connection is described with grid point identification numbers. (Character: “GRID” or “ELEM”; Default=“ELEM”)
ES1	Shell element identification number if TYPE=“ELEM”. Shell grid point identification number if TYPE=“GRID”. See Figure 17-4. (Integer > 0)
EA1	Solid element identification number if TYPE=“ELEM”. Solid grid point identification number if TYPE=“GRID”. (Integer > 0)

Field	Contents
EB1	Solid grid-point identification number for TYPE="GRID" only. (Integer > 0 or blank)
ES2	Shell grid-point identification number for TYPE="GRID" only. (Integer > 0 or blank)
EA2	Solid grid-point identification number for TYPE="GRID" only. (Integer > 0 or blank)
EB2	Solid grid-point identification number for TYPE="GRID" only. (Integer > 0 or blank)

REMARKS:

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of-freedom are put in the dependent set (m-set). The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower solid edge. The two rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson's ratio effects are considered in the translational degrees-of-freedom.

2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see PARAM,TOLRSC and PARAM,SEPIXOVR.

3. When using the TYPE="ELEM" option:

   - The solid elements are CHEXA, CPENTA, and CTETRA with and without midside nodes. The shell elements are CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8, or CTRIA6.

   - Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using SEELT entry to reassign the downstream boundary element to an upstream superelement.

   - It is not recommended to connect more than one shell element to the same solid using the ELEM option. If attempted, conflicts in the multipoint constraint relations may lead to UFM 6692.

4. When using TYPE="GRID" option:
   - The GRID option does not verify that the grids used are valid shell and/or solid grids.
   - The grids in the GRID option can be in different superelements. The shell grid must be in the upstream superelement.
5. It is recommended that the height of the solid element's face is approximately equal to the shell element's thickness of the shell. The shell edge should then be placed in the middle of the solid face.
6. The shell edge may coincide with the upper or lower edge of the solid face.
7. The RSSCON entry, unlike MPCs, cannot be selected through the Case Control Section.
8. Forces of multipoint constraints may be recovered in the linear structured solution sequences (SOLs 101 through 200) with the MPCFORCES Case Control command.
9. The RSSCON is ignored in heat-transfer problems.
10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See "Degree-of-Freedom Sets" for a list of these entries.

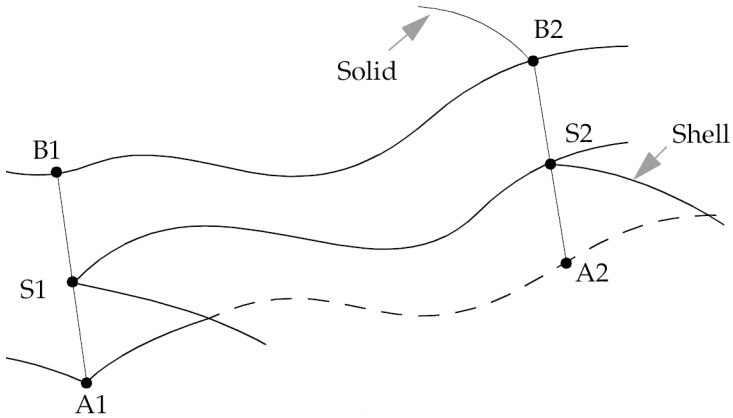


Figure 17-4. Shell Elements Connected to the Faces of Solid Elements

RTRPLT

Rigid Triangular Plate

Defines a rigid triangular plate.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RTRPLT	EID	GA	GB	GC	CNA	CNB	CNC		
	CMA	CMB	CMC	ALPHA					

EXAMPLE:

RTRPLT	7	1	2	3	1236	3	3		

FIELDS:

Field	Contents
EID	Element identification number.
GA, GB, GC	Grid point identification number of connection points.
CNA, CNB, CNC	Independent degrees-of-freedom in the global coordinate system for the element at grid points GA, GB, and GC, indicated by any of the Integers 1 through 6 with no embedded blanks. See Remark 1 . (Integer ≥ 0 or blank)
CMA, CMB, CMC	Component numbers of dependent degrees-of-freedom in the global coordinate system. (Any of the Integers 1 through 6 with no embedded blanks, or 0 or blank.)
ALPHA	Thermal expansion coefficient.

REMARKS:

1. The total number of components in CNA, CNB, and CNC must equal six; for example, CNA=1236, CNB=3, CNC=3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.

2. If CMA, CMB, and CMC are all zero or blank or if the continuation entry is omitted, all of the degrees-of-freedom not in CNA, CNB, and CNC will be made dependent (i.e., they will be made members of the m-set).

3. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

4. Element identification numbers should be unique with respect to all other element identification numbers.

5. Rigid elements, unlike MPCs, are not selected through the Case Control Section.

6. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCES Case Control command.

7. Rigid elements are ignored in heat transfer problems.

8. See R-Type Elements in the *NX Nastran Element Library* for a discussion of rigid elements.

9. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

10. The RIGID case control command must be assigned to “LAGRAN” to use ALPHA for the calculation of thermal expansion. See the RIGID case control command for details.

11. The TEMP(LOAD) and TEMP(INIT) value used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average TEMP(LOAD) and TEMP(INIT) is calculated for each leg of the element using the values on the independent/dependent grid pairs such that each leg can have a different thermal strain if the temperatures vary at the grids.

    The rigid element thermal strains are calculated by
    $\varepsilon_{thermal} = \alpha(\text{AVGTEMP(LOAD)} - \text{AVGTEMP(INIT)})$.

    If TEMP(LOAD) or TEMP(INIT) are not defined, they are assumed to be zero.

RVDOF

Degree-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RVDOF	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

RVDOF	900	1	200	3					

FIELDS:

Field	Contents
IDi	Grid point or scalar point identification number. (Integer > 0)
Ci	Component number. (0 ≤ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

REMARKS:

1. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.
2. The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual

vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVDOF1

Degree-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	

ALTERNATE FORMAT:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	“THRU”	ID2					

EXAMPLE:

RVDOF1	2	101	201	356					

ALTERNATE EXAMPLE:

RVDOF1	123456	7	THRU	109					

FIELDS:

Field	Contents
C	Component number. (0 ≤ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

Field	Contents
IDi	Grid point or scalar point identification number. (Integer > 0)

REMARKS:

1. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.

2. The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVEL

Specification of Elements for Residual Vector Computations

Specifies CBUSH, CBUSH1D, CDAMPi, CELASi, and CVISC elements for which to obtain static solutions for use in residual vector computations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
RVEL	TYPE	ID1	ID2	ID3	ID4	ID5	ID6	ID7	

EXAMPLES:

RVEL	ELEM	3	6	8	9	15	7		

RVEL	ELEM	3	6	THRU	9	15			

FIELDS:

Field	Contents
TYPE	Specifies whether the identification numbers listed correspond to elements or properties. (Character: “ELEM” or “PROP”; No default)
IDi	Element identification number if TYPE = ELEM or property identification number if TYPE = PROP. See Remark 1. (Integer > 0 or “THRU”; No default)

REMARKS:

1. If TYPE = PROP, all elements that reference the listed properties are available for residual vector computations.

2. Elements referenced by an RVEL bulk entry will have residual vectors calculated for opposing unit loads that are applied to the ends of the elements. Elements with stiffness or damping in multiple directions will have separate opposing loads applied for each direction.

3. RVEL bulk entries are only processed if the RESVEC case control command is present with the RVEL describer specified.

4. Continuations are not supported. Although, multiple RVEL bulk entries are permitted. For example, the following input is supported:

```
RVEL    ELEM    5       8       12      34      66      83      23
RVEL    ELEM    103     THRU    123
```

5. The load vectors for elements contained in a superelement and specified by an RVEL bulk entry are carried to the next downstream superelement and used for residual vector computation in that downstream superelement.

SEBNDRY

Superelement Boundary-Point Definition

Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEBNDRY	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

EXAMPLE 1:

SEBNDRY	400	4	10	20	30	40			

EXAMPLE 2:

SEBNDRY	400	ALL	10	20	30	THRU	40		

FIELDS:

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer > 0)
SEIDB	Superelement identification. See Remark 2 . (Integer > 0 or Character “ALL” ; Default=“ALL”)
GIDAi	Identification number of a boundary grid point in superelement SEIDA. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.)

REMARKS:

1. SEBNDRY may only be specified in the main Bulk Data Section and is not recognized after the BEGIN SUPER=n.

2. SEIDB may reference partitioned superelements or superelements in the main Bulk Data Section.

SEBSET

Fixed Boundary Degrees-of-Freedom

Defines boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEBSET	SEID	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

SEBSET	5	2	135	14	6				

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points)
IDi	Grid or scalar point identification numbers. (Integer > 0)

REMARKS:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If only SEBSETi are entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If both SEBSETi and SECSETi entries are present, the c-set degrees-of-freedom are defined by the SECSETi entries and any remaining boundary points are placed in the b-set.

2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBSET1

Fixed Boundary Degrees-of-Freedom, Alternate Form of SEBSET

Defines boundary degrees-of-freedom to be fixed (b-set) during component mode calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEBSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

EXAMPLE:

SEBSET1	5	2	135	14	6	23	24	25	
	122	127							

ALTERNATE FORMAT AND EXAMPLE:

SEBSET1	SEID	C	G1	“THRU”	G2				
SEBSET1	5	3	6	THRU	32				

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points, 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; for THRU option G1 < G2.)

REMARKS:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If there are only SEBSETi entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both SEBSETi and SECSETi entries present, the c-set degrees-of-freedom are defined by the SECSETi entries, and any remaining boundary points are placed in the b-set.

2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBULK

Partitional Superelement Connection

Defines superelement boundary search options and a repeated, mirrored, or collector superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEBULK	SEID	TYPE	RSEID	METHOD	TOL	LOC	UNITNO		

EXAMPLE:

SEBULK	14	REPEAT	4	AUTO	1.0E-3				

FIELDS:

Field	Contents	
SEID	Partitioned superelement identification number. (Integer > 0)	
TYPE	Superelement type. (Character; No Default)	
	PRIMARY	Primary
	REPEAT	Identical
	MIRROR	Mirror
	COLLCTR	Collector
	EXTERNAL	External
	EXTOP2	External using an OUTPUT2 file created in an earlier run.
	EXTOP4	External using an OP4 file created in an earlier run.

Field	Contents
RSEID	Identification number of the reference superelement, used if TYPE="REPEAT" and "MIRROR". (Integer ≥ 0; Default=0)
METHOD	Method to be used when searching for boundary grid points. (Character: "AUTO" or "MANUAL"; Default="AUTO")
TOL	Location tolerance to be used when searching for boundary grid points. (Real; Default=10E-5)
LOC	Coincident location check option for manual connection option. (Character: "YES" or "NO"; Default="YES")
UNITNO	Fortran unit number for the OUTPUT2 or OP4 file (applicable and meaningful only when TYPE = "EXTOP2" or "EXTOP4").

REMARKS:

1. The TYPE="REPEAT" or "MIRROR" does not include superelements upstream of the reference superelement. A repeated or mirrored superelement can have boundaries, loads, constraints, and reduction procedures that are different than the reference superelement.

2. METHOD="MANUAL" requires SECONCT entries. SEBNDRY and SEEXCLD, which reference SEID, will produce a fatal message.

3. SECONCT, SEBNDRY, and SEEXCLD entries can be used to augment the search procedure and/or override the global tolerance.

4. For combined automatic and manual boundary search, the METHOD="AUTO" should be specified and connections should be specified on a SECONCT entry.

5. TOL and LOC are the default values that can be modified between two superelements by providing the required tolerance on the SECONCT entry.

6. TYPE="MIRROR" also requires specification of a SEMPLN entry.

7. TYPE="COLLCTR" indicates a collector superelement, which does not contain any grids or scalar points.

8. For TYPE = "EXTERNAL", "EXTOP2", or "EXTOP4", see discussion under the description of the EXTSEOUT case control entry for employing external superelements using the new two-step procedure. For employing external

superelements using the old three-step procedure, see discussion under the description of the EXTOUT parameter in Chapter 7.

9. This entry will only work if PART superelements (BEGIN SUPER) or external superelements created by employing the EXTSEOUT case control entry exist.

SECONCT

Partitioned Superelement Boundary-Point Connection

Explicitly defines grid and scalar point connection procedures for a partitioned superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	GIDB1	GIDA2	GIDB2	GIDA3	GIDB3	-etc.-		

EXAMPLE:

SECONCT	10	20	1.0E-4	YES					
	1001	4001			2222	4444			

ALTERNATE FORMAT AND EXAMPLE:

SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	‘THRU’	GIDA2	GIDB1	‘THRU’	GIDB2			

SECONCT	10	20							
	101	‘THRU’	110	201	‘THRU’	210			

FIELDS:

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer > 0)

Field	Contents
SEIDB	Identification number of superelement for connection to SEIDA. (Integer ≥ 0)
TOL	Location tolerance to be used when searching for or checking boundary grid points. (Real; Default=10E-5)
LOC	Coincident location check option for manual connection. (Character; “YES” or “NO”; Default=“YES”)
GIDAi	Identification number of a grid or scalar point in superelement SEIDA, which will be connected to GIDBi.
GIDBi	Identification number of a grid or scalar point in superelement SEIDB, which will be connected to GIDAi.

REMARKS:

1. SECONCT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. TOL and LOC can be used to override the default values specified on the SEBULK entries.
3. The continuation entry is optional.
4. The (GIAi, GIBi) pair must both be grids or scalar points.
5. All six degrees-of-freedom of grid points will be defined as boundary degrees-of-freedom.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.
7. Blank fields are allowed after the first GIDA1-GIDB1 pair. Blank fields must also occur in pairs. This remark does not apply to the alternate format.
8. For Alternate Format 1, the thru ranges must be closed sets. That is, all IDs listed between 101 and 110 in the example must exist in the model.

SECSET

Free Boundary Degrees-of-Freedom

Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SECSET	SEID	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

SECSET	3	124	1	5	23	6	15		

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
Ci	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
IDi	Grid or scalar point identification number. (Integer > 0)

REMARKS:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Coordinates listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined by the program and listed in the SEMAP table output.

2. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

3. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SECSET1

Free Boundary Degrees-of-Freedom, Alternate Form of SECSET

Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SECSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

EXAMPLE:

SECSET1	5	2	135	14	6	23	24	25	
	122	127							

ALTERNATE FORMATS AND EXAMPLE:

SECSET1	SEID	C	G1	“THRU”	G2				
SECSET1	5	3	6	THRU	32				

SECSET1	SEID		“ALL”						
SECSET1	SEID		ALL						

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)

Field	Contents
C	Component numbers of degree-of-freedoms. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0)

REMARKS:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Degrees-of-freedom listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined automatically and listed in the SEMAP table output.

2. If the alternate formats are used, the grid points Gi are not required to exist or to be exterior degrees-of-freedom and may be listed on SECSET1 entries. Points of this type will cause one warning message but will otherwise be ignored.

3. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.

5. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

   - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEELT

Superelement Boundary Element Reassignment

Reassigns superelement boundary elements to an upstream superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEELT	SEID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	

EXAMPLE:

SEELT	2	147	562	937					

ALTERNATE FORMAT AND EXAMPLE:

SEELT	SEID	EID1	“THRU”	EID2					
SEELT	5	12006	THRU	12050					

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
EIDi	Element identification numbers. (Integer > 0 or “THRU”; for “THRU” option EID1 < EID2.)

REMARKS:

1. Elements connected entirely to the exterior points of an upstream superelement are called boundary elements and are assigned to the downstream superelement. The SEELT entry provides the means of reassigning the element to the upstream superelement. This entry may be applied to boundary elements only.

2. Open sets are allowed with the “THRU” option.

3. Elements processed with primary superelements will also be contained in any referencing secondary superelement.

4. EIDi may refer to plot elements, general elements, and structural elements.

5. This entry does not change the exterior grid point set of the superelement.

6. SEELT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

7. If the model contains a BEGIN SUPER, the SEELT entry will assign the specified elements to the SEID (not just the boundary elements). This is an alternative to using SESET. In this case, SEID=0 is a valid entry.

SEEXCLD

Partitioned Superelement Exclusion

Defines grids that will be excluded during the attachment of a partitioned superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEEXCLD	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

EXAMPLE1:

SEEXCLD	110	10	45	678	396				

EXAMPLE 2:

SEEXCLD	400	ALL	10	20	30	THRU	40		

FIELDS:

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer ≥ 0)
SEIDB	Superelement identification. (Integer ≥ 0 or Character=“ALL”, Default=“ALL”)
GIDAi	Identification number of a grid in superelement SEIDA to be excluded from connection to superelement SEIDB. (Integer > 0 or “THRU”; for “THRU” option G1 < G2.)

REMARKS:

1. SEEXCLD can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

2. SEIDB may reference partitioned superelements or superelements defined in the main Bulk Data Section.

3. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELABEL

Superelement Output Label

Defines a label or name to be printed in the superelement output headings.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SELABEL	SEID	LABEL							

EXAMPLE:

SELABEL	10	LEFT REAR FENDER, MODEL XYZ2000	

FIELDS:

Field	Contents
SEID	Partitioned superelement identification number. (Integer > 0)
LABEL	Label associated with superelement SEID for output headings. (Character. See Bulk Data Syntax Rules.)

REMARKS:

1. SELABEL can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Only one SELABEL per superelement may be specified.
3. The label will appear in all superelement output headings. However, in some headings the label may be truncated.
4. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELOAD

External Superelement Load Mapping to Residual

Maps loads from an external superelement to a specified load set for the residual structure.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SELOAD	LIDS0	SEID	LIDSE						

EXAMPLE:

SELOAD	10010	100	10						

FIELDS:

Field	Contents
LIDS0	Identification number of the mapped load set to be used in the residual structure. (Integer > 0)
SEID	Partitioned identification number of the partitioned external superelement representing the external superelement. (Integer > 0)
LIDSE	Load set identification number used in the external superelement. (Integer > 0)

REMARKS:

1. SELOAD can only be specified in the main bulk section and is ignored after the BEGIN SUPER = *n* command. Because this load is applied at the boundary grids of the external superelement, the external superelement boundary grids must be connected (via the SECONCT bulk entry for example) directly to a residual grid. If this is not done, the external superelement loading cannot be properly mapped to

the residual and an error will be issued (e.g. user fatal message 2008; load set 2 references undefined grid point 201).

As a best practice, create the external superelement using the component mode reduction method. The component mode reduction method requires the use of q-set scalar DOF to represent the component modes. The q-set DOF must also be connected to the corresponding scalar DOF in the residual via the SECONCT bulk entry. The scalar DOF in the external superelement and the residual are defined using the SPOINT bulk entry.

2. SELOAD only works if PART superelements (BEGIN SUPER) exist that represent external superelements created using the EXTSEOUT case control command.

3. LIDSE is the identification number of a load set defined in the external superelement creation run using the EXTSEOUT case control command.

4. Multiple SELOAD commands can be used for the same SEID, but a specific LIDSE can only be mapped once.

5. LIDS0 must be unique for the residual structure.

6. The mapped load sets LIDS0s can be referenced by other load bulk entries in the system solution. Examples of load bulk entries include LOAD, LSEQ, TLOADi, and RLOADi.

SELOC

Partitioned Superelement Location

Defines a partitioned superelement relocation by listing three non-colinear points in the superelement and three corresponding points not belonging to the superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SELOC	SEID	PA1	PA2	PA3	PB1	PB2	PB3		

EXAMPLE:

SELOC	110	10	100	111	1010	112	30		

FIELDS:

Field	Contents
SEID	Partitioned identification number of the partitioned superelement. (Integer > 0)
PAi	Identification numbers of three non-colinear grids which are in the partitioned superelement. (Integer > 0)
PBi	Identification numbers of three grids (GRID entry) or points (POINT entry) defined in the main Bulk Data Section to which PAi will be aligned. (Integer > 0)

REMARKS:

1. SELOC can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

2. The superelement will be rotated and translated for alignment of the GAi and GBi locations.

3. The PAi can be grid points only; PBi can be either GRIDs or POINTs.

4. PA1, PA2, and PA3 must be contained in superelement SEID.

5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section. If they belong to a superelement that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.

6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.

7. Three grids or points are required even if the superelement connects to only one or two exterior grids.

8. Coordinate systems, global displacement directions, and element coordinate systems for the superelement will rotated and translated.

9. The global coordinate directions of the boundary grid points of the upstream superelement will be transformed internally to the global coordinate directions of the attachment grid points in the downstream superelement. For displacement data recovery, the output will be in the original global coordinate system.

10. The translation and rotation of the superelement to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:

    - The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.

    - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.

    - The z-axis is defined by the cross product of the x-axis into the y-axis.

    - The rotation and translation transformation aligns the local system defined by the superelement grids with the local system defined by the main Bulk Data Section grids.

11. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEMPLN

Superelement Mirror Plane

Defines a mirror plane for mirroring a partitioned superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEMPLN	SEID	“PLANE”	P1	P2	P3				

EXAMPLE:

SEMPLN	110	PLANE	12	45	1125				

FIELDS:

Field	Contents
SEID	Partitioned superelement identification number. (Integer > 0)
“PLANE”	Flag indicating that the plane is defined by three non-colinear points.
Pi	GRID or POINT entry identification numbers of three non-colinear points. (Integer > 0)

REMARKS:

1. SEMPLN can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

2. Grids or points referenced on this entry must be defined in the main Bulk Data Section.

SENQSET

Superelement Internal Generalized Degree-of-Freedom

Defines number of internally generated scalar points for superelement dynamic reduction.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SENQSET	SEID	N							

EXAMPLE:

SENQSET	110	45							

FIELDS:

Field	Contents
SEID	Partitioned superelement identification number. See Remark 3 . (Integer > 0 or Character=“ALL”)
N	Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer > 0; Default=0)

REMARKS:

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID=“ALL” will automatically generate N q-set degrees-of-freedom for all superelements, except the residual structure (SEID=0). Specifying additional

SENQSET entries for specific superelements will override the value of N specified on this entry.

4. If the user manually specifies q-set degrees-of-freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.

5. See PARAM,NQSET for an alternate method of specifying QSET degree-of-freedoms.

6. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEQEP

Grid and Scalar Point Resequencing

Redefines the sequence of extra points to optimize bandwidth.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEQEP	ID1	SEQID1	ID2	SEQID2	ID3	SEQID3	ID4	SEQID4	

EXAMPLE:

SEQEP	5392	15.6	596	0.2	2	1.9	3	2	

FIELDS:

Field	Contents
IDi	Extra point identification number. (Integer > 0)
SEQIDi	Sequenced identification number. (Real or Integer > 0)

REMARKS:

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID.
2. The SEQIDi numbers must be unique and may not be the same as a point IDi that is not being changed. No extra point IDi may be referenced more than once.
3. From one to four extra points may be specified on a single entry.
4. If an extra point IDi is referenced more than once, the last reference will determine its sequence.

SEQGP

Grid and Scalar Point Resequencing

Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEQGP	ID1	SEQID1	ID2	SEQID2	ID3	SEQID3	ID4	SEQID4	

EXAMPLE:

SEQGP	5392	15.6	596	0.2	2	1.9	3	2	

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
SEQIDi	Sequenced identification number. (Real > 0.0 or Integer > 0)

REMARKS:

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID.

2. The SEQIDi numbers must be unique and may not be the same as a point IDi which is not being changed. No grid point IDi may be referenced more than once.

3. From one to four grid or scalar points may be resequenced on a single entry.

4. If a point IDi is referenced more than once, the last reference will determine its sequence.

5. Automatic resequencing is also available. See “OLDSEQ” .

SEQSEP

Superelement Sequences

Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEQSEP	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

EXAMPLE:

SEQSEP	121	21	109	114	124	131			

FIELDS:

Field	Contents
SSID	Identification number for secondary superelement. (Integer > 0)
PSID	Identification number for the primary superelement. (Integer ≥ 0)
GPi	Exterior grid point identification numbers for the primary superelement. (Integer > 0)

REMARKS:

1. This entry is not needed if the grid points listed on the CSUPER entry with the same SSID are in the order of the corresponding exterior grid points of the primary superelement.

2. In Figure 17-5, the exterior grid points of 10, 20, and 30 of SEID=1 correspond to the points 13, 12, and 11, respectively, of image SEID=2. The CSUPER entry may be defined alone or with a SEQSEP entry as shown in Figure 17-5.

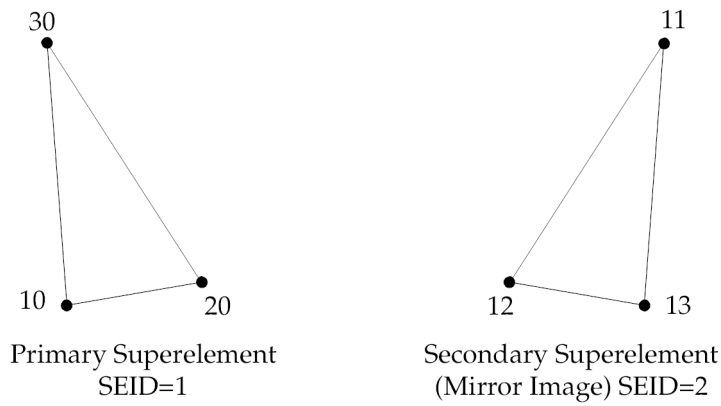


Figure 17-5. Grid Point Correspondence Between Primary and Secondary Superelements

CSUPER Entry Only:

1	2	3	4	5	6	7	8	9	10
CSUPER	2	1	13	12	11				

CSUPER and SEQSEP Entries:

CSUPER	2	I	11	12	13				
SEQSEP	2	1	30	20	10				

SEQSET

Superelement Generalized Degrees-of-Freedom

Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEQSET	SEID	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

SEQSET	15	1	123456	7	5	22	3		

FIELDS:

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points.)
IDi	Grid or scalar point identification numbers. Must be an exterior point. (Integer > 0)

REMARKS:

1. Degrees-of-freedom specified on this entry may not be specified for another superelement.
2. Generalized degrees-of-freedom are interior to the residual structure.

3. Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.

4. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

5. This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user’s responsibility to place them in a downstream superelement.

   Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set to allow the boundary points to participate in the system mode shapes.

   Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user’s responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these degrees-of-freedom for secondary superelements.

SEQSET1

Superelement Generalized Degrees-of-Freedom, Alternate Form

Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEQSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

EXAMPLE:

SEQSET1	15	123456	1	7	9	22	105	6	
	52	53							

ALTERNATE FORMAT AND EXAMPLE:

SEQSET1	SEID	C	G1	“THRU”	G2				
SEQSET1	16	0	101	THRU	110				

FIELDS:

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)

Field	Contents
Gi	Grid or scalar point identification numbers. Must be exterior points. (Integer > 0 or “THRU”; for THRU option G1 < G2.)

REMARKS:

1. Degrees-of-freedom specified on this entry may not be specified for another superelement.

2. Generalized degrees-of-freedom are interior to the residual structure.

3. Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.

4. Degrees-of-freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

5. This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user’s responsibility to place them in a downstream superelement.

   Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set, to allow the boundary points to participate in the system mode shapes.

   Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user’s responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these entries for secondary superelements.

SESET

Superelement Interior Point Definition

Defines interior grid points for a superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SESET	SEID	G1	G2	G3	G4	G5	G6	G7	

EXAMPLE:

SESET	5	2	17	24	25	165			

ALTERNATE FORMAT AND EXAMPLE:

SESET	SEID	G1	“THRU”	G2					
SESET	2	17	THRU	165					

FIELDS:

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer ≥ 0)
Gi	Grid or scalar point identification number. (0 < Integer; G1 < G2)

REMARKS:

1. Interior grid points may also be defined via field 9 of the GRID and GRIDG Bulk Data entries. The SESET entry takes precedence over the SEID field on the GRID on GRIDG entries. SESET defines grid and scalar points to be included as interior to a superelement. SESET may be used as the primary means of defining superelements or it may be used in combination with SEELT entries which define elements interior to a superelement.

2. Gi may appear on an SESET entry only once.

3. Scalar points are ignored.

4. Open sets are allowed with the “THRU” option. Missing grid points (whether in “THRU” range or mentioned explicitly) are not identified.

5. All degrees-of-freedom for Gi are placed in the o-set of the superelement. See “Degree-of-Freedom Sets” .

6. SESET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

7. Continuations are not allowed. Multiple SESET entries with the same ID are permitted when multiple input lines are required. For example,

SESET	5	17	THRU	25	45	76	49	84	
SESET	5	102	107	165					

SESUP

Fictitious Support

Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SESUP	SEID	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

SESUP	5	16	215						

FIELDS:

Field	Contents
SEID	Superelement identification number. Must a primary superelement. (Integer > 0)
IDi	Grid or scalar point identification number. Must be exterior points. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points; Any unique combination of the Integers 1 through 6 for grid points.)

REMARKS:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

2. The Ci degrees-of-freedom must be exterior degrees-of-freedom of the SEID superelement.

3. See “Rigid Body Supports” in the *NX Nastran User’s Guide* for a discussion of supported degrees-of-freedom (members of the r-set).

4. There must be a sufficient number of degrees-of-freedom on SESUP entries to discard any free body modes of the superelement.

5. SESUP Bulk Data entries are not allowed for part (partitioned bulk data) superelements. Use the SUPORT Bulk Data records to identify component rigid body modes.

SET1

Set Definition

Defines a list of structural grid points, elements, or DRESPi (i=1,2,3) responses.

SET1 entries are referenced by SPLINEi entries, PANEL entries, XYOUTPUT, and the P2RSET option on DOPTPRM entries.

GRID OR ELEMENT ID FORMAT:

1	2	3	4	5	6	7	8	9	10
SET1	SID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

RESPONSE ID FORMAT:

1	2	3	4	5	6	7	8	9	10
SET1	SID	R1	R2	R3	R4	R5	R6	R7	
	R8	-etc.-							

EXAMPLE:

SET1	3	31	62	93	124	16	17	18	
	19								

ALTERNATE FORMATS AND EXAMPLE:

SET1	SID	ID1	“THRU”	ID2	ID3	“THRU”	ID4		
– or –									
SET1	SID	R1	“THRU”	R2	R3	“THRU”	R4		
SET1	6	32	THRU	50	60	70	73	78	
	80	THRU	86						

FIELDS:

Field	Contents
SID	Unique identification number. (Integer > 0)
IDi	List of structural grid point or element identification numbers. (Integer > 0 or “THRU”; for the “THRU” option, G1 < G2.)
Ri	List of DRESPi (i=1,2,3) response IDs for the P2RSET option on the DOPTPRM bulk entry. (Integer > 0 or “THRU”; for the “THRU” option, R1 < R2.)

REMARKS:

1. SET1 entries may be referenced by the SPLINEi entries, PANEL entries, XYOUTPUT, and the P2RSET option on the DOPTPRM bulk entry.
2. When using the “THRU” option for SPLINEi or PANEL data entries, all intermediate grid points must exist.
3. When using the “THRU” option for XYOUTPUT requests, missing grid points are ignored.
4. When using the “THRU” option for DRESPi requests, missing response IDs are ignored.
5. The SID must be unique from other SET1 and SET3 SIDs.
6. “THRU” cannot appear in Field 3 or 9. For continuations, “THRU” cannot appear in Fields 2 or 9.

SET2

Grid Point List

Defines a list of structural grid points in terms of aerodynamic macro elements.

SET2 entries are referenced by SPLINEi entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SET2	SID	MACRO	SP1	SP2	CH1	CH2	ZMAX	ZMIN	

EXAMPLE:

SET2	3	111	0.0	0.75	0.0	0.667	3.51		

FIELDS:

Field	Contents
SID	Unique identification number. (Integer > 0)
MACRO	Element identification number of an aerodynamic macro element. (Integer > 0)
SP1, SP2	Lower and higher span division points defining the prism containing the set. (Real)
CH1, CH2	Lower and higher chord division points defining the prism containing the set. (Real)
ZMAX, ZMIN	Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a CAEROi entry) of the prism containing set. (Real)

REMARKS:

1. The SET2 entry is referenced by the SPLINEi entry.

2. Every grid point within the defined prism and within the height range will be in the list. For example:

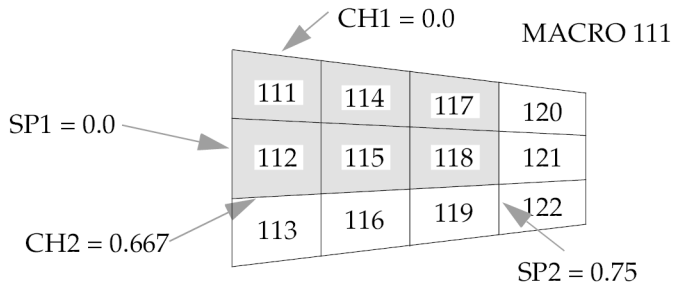


Figure 17-6. SET2 Entry Example

The shaded area in Figure 17-6 defines the cross section of the prism for the sample data given above. Points exactly on the boundary may be missed; therefore, to get all the grid points within the area of the macro element, SP1=-.01, SP2=1.01, etc. should be used.

3. A zero value for ZMAX or ZMIN implies a value of infinity. Usually, ZMAX ≥ 0.0 and ZMIN ≤ 0.0.

4. To print the (internal) grid IDs found, use DIAG 18.

SET3

Set Definition

Defines a list of structural grid points, elements, or physical properties.

SET3 entries are referenced by ACMODL entries and PANEL entries.

GRID ID FORMAT:

1	2	3	4	5	6	7	8	9	10
SET3	SID	TYPE	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	-etc.-							

EXAMPLE:

SET3	1	GRID	11	13	14	15	20	22	
	34	41							

ALTERNATE FORMATS AND EXAMPLE:

SET3	SID	TYPE	ID1	“THRU”	ID2	ID3	“THRU”	ID4	
SET3	2	ELEM	20	THRU	33	36	THRU	44	
	49	THRU	62	91	THRU	110			

FIELDS:

Field	Contents
SID	Unique identification number. (Integer > 0)
TYPE	Set type. (Character: “GRID”, “ELEM”, “PROP”)

Field	Contents
IDi	Identifying numbers of structural grids, elements, or physical properties. (Integer > 0)

REMARKS:

1. The SID must be unique from other SET1 and SET3 SIDs.
2. By specifying the GRID field, the bulk entry referencing the SET3 bulk entry interprets the IDi as structural grid point IDs.
3. By specifying the ELEM field, the bulk entry referencing the SET3 bulk entry interprets the IDi as structural element IDs.
4. By specifying the PROP field, the bulk entry referencing the SET3 bulk entry interprets the IDi as physical property IDs for structural elements.
5. “THRU” cannot appear in Field 4 or 9. For continuations, “THRU” cannot appear in Fields 2 or 9.

SETREE

Superelement Tree Definition (Alternate Form of DTI,SETREE)

Specifies superelement reduction order.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SETREE	SEID	SEUP1	SEUP2	SEUP3	SEUP4	SEUP5	SEUP6	SEUP7	
	SEUP8	SEUP9	-etc.-						

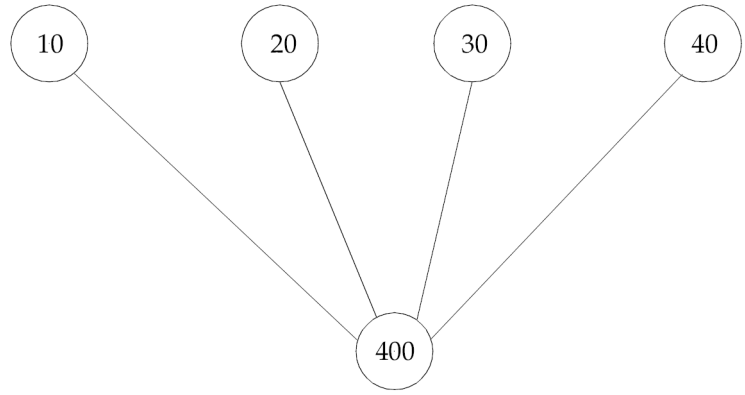
EXAMPLE:

SETREE	400	10	20	30	40				

FIELDS:

Field	Contents
SEID	Partitioned superelement identification number of a downstream superelement. (Integer ≥ 0)

Field	Contents
SEUPi	Identification number of superelements that are upstream of SEID. (Integer > 0)



REMARKS:

1. SETREE entries or DTI,SETREE entry are required for multilevel superelement configurations.
2. At least one SETREE entry is required for each nontip superelement, including the residual structure (SEID=0). Multiple SETREE entries with the same SEID are allowed.
3. A superelement may appear only once in an SEUPi field on all SETREE entries.
4. If an DTI,SETREE entry is provided, then SETREE entries are not required.
5. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
6. Superelements not referenced on the SETREE or DTI,SETREE entry will not be included in the analysis unless they are attached to grid points which belong to the residual structure. These superelements will be placed in the tree immediately above the residual structure.

7. The SETREE entry will only work if PART (BEGIN SUPER) superelements exist in the model. If there are no PARTs in the model, the SETREE entries will be ignored.

SEUSET

Superelement Degree-of-Freedom Set Definition

Defines a degree-of-freedom set for a superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEUSET	SEID	SNAME	ID1	C1	ID2	C2	ID3	C3	

EXAMPLE:

SEUSET	15	U4	1	123456	7	5	22	3	

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string “ZERO”, followed by the set name.)
IDi	Grid or scalar point identification numbers. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blank for grid points; Integer 0 or blank for scalar points.)

REMARKS:

1. SNAME may refer to any of the set names given in “Degree-of-Freedom Sets” or their new names on the DEFUSET entry. However, in the Solution Sequences 100 through 200, it is recommended that SNAME refer only to the set names “U1” through “U9” or their new names on the DEFUSET entry.

2. If SNAME=“ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

3. A maximum of 18 degrees-of-freedom may be designated on a single entry.

4. If degrees-of-freedom defined by SEUSET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the SEUSET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.

5. The SEUSET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if SEUSET entries are used.

6. If a SEUSETi bulk entry lists a standard NX Nastran set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET bulk entries.

7. The user defined degree-of-freedom sets U1 – U9 are created with the SEUSET/SEUSET1 bulk entries. Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases. See “User Defined Degree-of-Freedom Sets” for the list of these special cases.

SEUSET1

Superelement Degree-of-Freedom Set Definition, Alternate Form

Defines a degree-of-freedom set for a superelement.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SEUSET1	SEID	SNAME	C	G1	G2	G3	G4	G5	
	G6	G7	-etc.-						

EXAMPLE:

SEUSET1	15	U4	1	12	15	17	22	25	
	52	53							

ALTERNATE FORMAT AND EXAMPLE:

SEUSET1	SEID	SNAME	C	G1	“THRU”	G2			
SEUSET1	15	U4	1	12	THRU	27			

FIELDS:

Field	Contents
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string “ZERO”, followed by the set name.)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)

Field	Contents
Gi	Grid or scalar point identification number. (Integer > 0)

REMARKS:

1. SNAME may refer to any of the set names given in “Degree-of-Freedom Sets” or their new names on the DEFUSET entry. However, in the Solution Sequences 100 through 200, it is recommended that SNAME refer only to the set names “U1” through “U9” or their new names on the DEFUSET entry.

2. If SNAME= “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

3. If the alternate format is used, all of the points G1 through G2 are assigned to the set.

4. If degrees-of-freedom defined by SEUSET1 entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the SEUSET1 entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.

5. The SEUSET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if SEUSET1 entries are used.

6. If a SEUSETi bulk entry lists a standard NX Nastran set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET bulk entries.

7. The user defined degree-of-freedom sets U1 – U9 are created with the SEUSET/SEUSET1 bulk entries. Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases. See “User Defined Degree-of-Freedom Sets” for the list of these special cases.

SLOAD

Static Scalar Load

Defines concentrated static loads on scalar or grid points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	S1	F1	S2	F2	S3	F3		

EXAMPLE:

SLOAD	16	2	5.9	17	-6.3	14	-2.93		

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
Si	Scalar or grid point identification number. (Integer > 0)
Fi	Load magnitude. (Real)

REMARKS:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. Up to three loads may be defined on a single entry.
3. If Si refers to a grid point, the load is applied to component T1 of the displacement coordinate system (see the CD field on the GRID entry).

SNORM

Surface Normal Vector at Grid Point

Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SNORM	GID	CID	N1	N2	N3				

EXAMPLE:

SNORM	3	2	0.	-1.	0.				

FIELDS:

Field	Contents
GID	Unique grid point identification number. (Integer > 0)
CID	Identification number of coordinate system in which the components of the normal vector are defined. See Remark 3 . (Integer ≥ 0; Default=0 for the basic coordinate system)
Ni	Components of normal vector. The three components of the normal need not define a unit vector. (Real; Default=0.0)

REMARKS:

1. The SNORM Bulk Data entry overrides any unique, internally-generated grid point normals that may have been requested with the user parameter SNORM , described in this guide.

2. The normal is used in CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. For all other elements, the normal is ignored.

3. If CID is a cylindrical or spherical coordinate system, the components Ni are in the local tangent system at grid GID. For example, if CID=10 is a spherical coordinate system and normals must be defined pointing outwards in the radial direction of the sphere, see Figure 17-7, then the SNORM entries for all grids GID on the sphere are simply

   SNORM, GID, 10, 1., 0., 0.

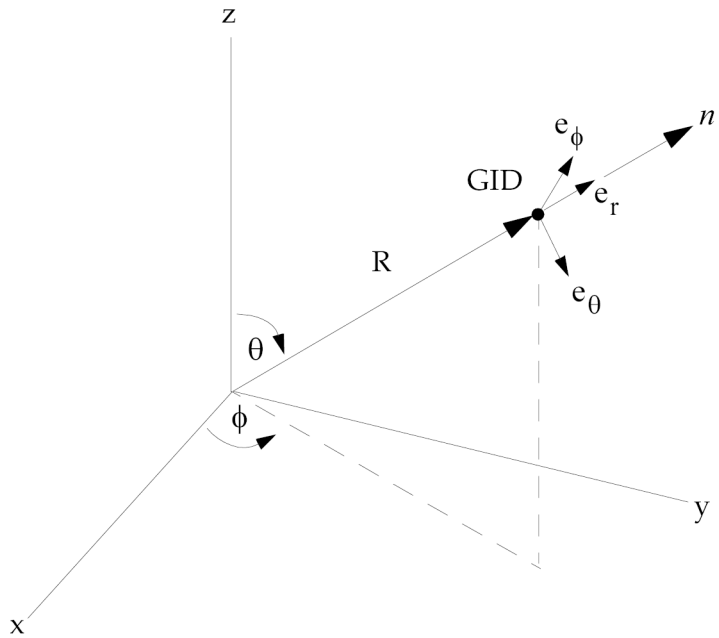


Figure 17-7.

SPC

Single-Point Constraint

Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

EXAMPLE:

SPC	2	32	3	-2.6	5				

FIELDS:

Field	Contents
SID	Identification number of the single-point constraint set. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (0 ≤ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
Di	Value of enforced motion for all degrees-of-freedom designated by Gi and Ci. (Real)

REMARKS:

1. Single-point constraint sets must be selected with the Case Control command SPC=SID.

2. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

3. Single-point forces of constraint are recovered during stress data recovery.

4. From 1 to 12 degrees-of-freedom may be specified on a single entry.

5. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.

6. For reasons of efficiency, the SPCD entry is the preferred method for applying enforced motion rather than the Di field described here.

7. Rotational degrees-of-freedom are in radians.

REMARKS RELATED TO SOLS 601 AND 701:

1. Enforced velocity or acceleration is not supported.

2. To apply enforced displacements with constant magnitude, SPC or SPCD may be used. To apply time-dependent enforced displacements, SPCD must be used. See SPCD entry.

SPC1

Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

EXAMPLE:

SPC1	3	2	1	3	10	9	6	5	
	2	8							

ALTERNATE FORMAT AND EXAMPLE:

SPC1	SID	C	G1	“THRU”	G2				
SPC1	313	12456	6	THRU	32				

FIELDS:

Field	Contents
SID	Identification number of single-point constraint set. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.)

REMARKS:

1. Single-point constraint sets must be selected with the Case Control command SPC=SID.

2. Enforced displacements are available via this entry when used with the recommended SPCD entry.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

4. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.

5. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.

6. Rotational degrees-of-freedom are in radians.

REMARKS RELATED TO SOLS 601 AND 701:

1. A global coordinate (Gi and Ci) referenced on SPCD does not need to be referenced on this entry.

SPCADD

Single-Point Constraint Set Combination

Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

EXAMPLE:

SPCADD	101	3	2	9	1				

FIELDS:

Field	Contents
SID	Single-point constraint set identification number. (Integer > 0)
Si	Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries. (Integer > 0; SID ≠ Si)

REMARKS:

1. Single-point constraint sets must be selected with the Case Control command SPC=SID.
2. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.
3. The Si values must be unique.

4. SPCADD entries take precedence over SPC or SPC1 entries. If both have the same set ID, only the SPCADD entry will be used.

SPCD

Enforced Motion Value

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

EXAMPLE:

SPCD	100	32	436	-2.6	5		2.9		

FIELDS:

Field	Contents
SID	Set identification number of the SPCD entry. (Integer > 0)
Gi	Grid or scalar point identification number. (integer > 0)
Ci	Component numbers. (0 ≤ Integer ≤ 6; up to six unique Integers may be placed in the field with no embedded blanks.)
Di	Value of enforced motion for at Gi and Ci. (Real)

REMARKS:

1. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command.
2. In dynamic analysis, the selection of SID is determined by the presence of the LOADSET request in Case Control as follows:

- There is no LOADSET request in Case Control

  SID is selected by the EXCITEID ID of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field

- There is a LOADSET request in Case Control

  SID is selected by LID in the selected LSEQ entries that correspond to the EXCITEID entry of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field.

3. A global coordinate (Gi and CI) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command.

4. Values of Di will override the values specified on an SPC Bulk Data entry, if the SID is selected as indicated above.

5. The LOAD Bulk Data entry will not combine an SPCD load entry.

6. In static analysis, this method of applying enforced displacements is more efficient than the SPC entry when more than one enforced displacement condition is applied. It provides equivalent answers.

7. In dynamic analysis, this direct method of specifying enforced motion is more accurate, efficient and elegant than the large mass and Lagrange multiplier techniques.

8. Rotational degrees-of-freedom are in radians.

REMARKS RELATED TO SOLS 601 AND 701:

1. Enforced velocity or acceleration is not supported.

2. To enforce displacement with constant magnitude, SID is selected by Case Control command LOAD=SID for both static and transient analysis.

3. To enforce a time dependent displacement, SID is referenced by the field EXCITEID=SID in the TLOAD1 entry with TYPE=1. Time dependent loads are selected by Case Control command DLOAD.

4. A global coordinate (Gi and Ci) referenced on this entry does not need to be referenced on a SPC or SPC1 entry.

SPCOFF

Excludes Degrees-of-Freedom from the AUTOSPC Operation

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See “Constraint and Mechanism Problem Identification” in SubDMAP SEKR in the *NX Nastran User’s Guide* for a description of the AUTOSPC operation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPCOFF	G1	C1	G2	C2	G3	C3	G4	C4	

EXAMPLE:

SPCOFF	32	436	5	1					

FIELDS:

Field	Contents
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points; Integers 1 through 6 with no embedded blanks for grid points.)

REMARKS:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.

2. Degrees-of-freedom that are specified as both SPC and SPCOFF will be considered as SPC.

SPCOFF1

Excludes DOFs from AUTOSPC Processing, Alternate Form

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See “Constraint and Mechanism Problem Identification” in “SubDMAP SEKR” in the *NX Nastran User’s Guide* for a description of the AUTOSPC operation.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPCOFF1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	-etc.-						

EXAMPLE:

SPCOFF1	2	1	3	10	9	6	5	4	
	8								

ALTERNATE FORMAT AND EXAMPLE:

SPCOFF1	C	G1	“THRU”	G2					
SPCOFF1	12456	6	THRU	32					

FIELDS:

Field	Contents
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; for “THRU” option, G1 < G2.)

REMARKS:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.

2. Degrees-of-freedom that are both specified as SPC and SPCOFF will be considered as SPC.

3. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.

SPLINE1

Surface Spline Methods

Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPLINE1	EID	CAERO	BOX1	BOX2	SETG	DZ	METH	USAGE	
	NELEM	MELEM							

EXAMPLE:

SPLINE1	3	111	115	122	14	0.			

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero-element (CAEROi entry ID) that defines the plane of the spline. (Integer > 0)
BOX1, BOX2	First and last box with motions that are interpolated using this spline; see Remark 3 . when using Mach Box method. (Integer > 0, BOX2 ≥ BOX1)
SETG	Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0)
METH	Method for the spline fit. IPS,TPS or FPS. See Remark 1 . (Character, Default=IPS)

Field	Contents
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 5 . (Character, Default=BOTH)
NELEM	The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default=10)
MELEM	The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default=10)

REMARKS:

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.

2. The interpolated points (k-set) will be defined by aero boxes. Figure 17-8 shows the cells for which u_k is interpolated if BOX1=115 and BOX2=118.

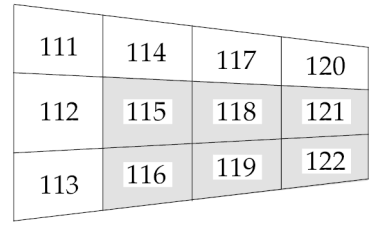


Figure 17-8. SPLINE1 Entry Example

3. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ=0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.

4. When using the Mach Box method, BOX1 and BOX2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. BOX1 and BOX2 do not refer to Mach Boxes.

5. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg=[GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk=[GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPgk]^T=[GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). NX Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

SPLINE2

Linear Spline

Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPLINE2	EID	CAERO	ID1	ID2	SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE					

EXAMPLE:

SPLINE2	5	8	12	24	60	0.	1.0	3	
	1.								

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. (Integer > 0)
ID1, ID2	First and last box or body element whose motions are interpolated using this spline. See Remark 6 when using the Mach Box method. (Integer > 0, ID2 ≥ ID1)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0)
DTOR	Torsional flexibility ratio (*EI*/*GJ*). (Real > 0.0; Default=1.0; use 1.0 for bodies.)

Field	Contents
CID	Rectangular coordinate system for which the y-axis defines the axis of the spline. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (in-plane bending rotations); however, it is not used for bodies. DTHY is for rotation about the spline's y-axis (torsion); however, it is used for slope of bodies. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 9 . (Character, Default=BOTH)

REMARKS:

1. The interpolated points (k-set) will be defined by aero boxes.

2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.

3. The flexibilities DZ, DTHX, and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment). See the *NX Nastran Aeroelastic Analysis User's Guide* for a discussion of special cases.

4. The continuation entry is required.

5. The SPLINE2 EID must be unique with respect to all SPLINEi entries.

6. When using the Mach Box method, ID1 and ID2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. ID1 and ID2 do not refer to Mach Boxes.

7. DTOR is the ratio of rotational to linear deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur.

8. If a SPLINE2 element only references one grid point, the job will fail without a message in the GI Module.

9. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg=[GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk=[GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPgk]^T]=[GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to Version 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). NX Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

SPLINE3

Aeroelastic Constraint Equation

Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPLINE3	EID	CAERO	BOXID	COMP	G1	C1	A1	USAGE	
	G2	C2	A2		-etc.				

EXAMPLE:

SPLINE3	7000	107	109	6	5	3	1.0		
	43	5	-1.0						

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Identification number of the macro-element on which the element to be interpolated lies. (Integer > 0)
BOXID	Identification number of the aerodynamic element; i.e., the box number. (Integer > 0)
COMP	The component of motion to be interpolated. See Remark 4 . (One of the Integers 1, 2, 3, 4, 5, or 6.)
Gi	Grid point identification number of the independent grid point. (Integer > 0)
Ci	Component numbers in the displacement coordinate system. (One of the Integers 1 through 6 for grid points, or 0 for scalar points.)

Field	Contents
Ai	Coefficient of the constraint relationship. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 6 . (Character, Default=BOTH).

REMARKS:

1. The independent grid points and components must refer to degrees-of-freedom in the g-set.

2. The constraint is given by

$$u_d = \sum A_i u_i$$

where

u_d = value of the dependent component of the aerodynamic element
u_i = displacement at grid Gi, component Ci.

3. The SPLINE3 EID must be unique with respect to all SPLINEi entries.

4. The allowable components by CAEROi entry type are indicated by an “X” in the table below:

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2		X	X	X	X
CAERO3			X		
CAERO4			X	X	X
CAERO5			X	X	X
3D Geometry	X	X	X	X	X

COMP=2: lateral displacement

COMP=3 transverse displacement

COMP=5: pitch angle

COMP=6: relative control angle for CAERO4 and CAERO5 yaw angle for CAERO2.

For general 3D aerodynamic geometries the components numbers refer to axes of the Aerodynamic Coordinate System (u_x, u_y, u_z, θ_x, θ_y, θ_z).

5. For Strip theory and Piston theory, the COMP=6 control surface relative angle is positive when the trailing edge has a negative z-deflection in the element coordinate system (see the *NX Nastran Aeroelastic Analysis User's Guide*.)

6. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg=[GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk=[GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $GPgk]^T]=[GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to Version 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). NX Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

SPLINE4

Surface Spline Methods

Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPLINE4	EID	CAERO	AELIST		SETG	DZ	METH	USAGE	
	NELEM	MELEM							

EXAMPLE:

SPLINE4	3	111	115		14	0.	IPS		

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel ID that defines the interpolation surface. (Integer > 0)
AELIST	Identification of an AELIST entry listing boxes with motions that are interpolated using this spline; see Remark 3 when using Mach Box method. (Integer > 0)
SETG	Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0)
METH	Method for the spline fit. One of IPS, TPS or FPS. See Remark 1 . (Character, Default=IPS)

Field	Contents
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4 . (Character, Default=BOTH)
NELEM	The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default=10)
MELEM	The number of FE elements along the local spline y-axis if using the FPS option. (integer > 0; Default=10)

REMARKS:

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.

2. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ=0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.

3. When using the Mach Box method, the AELIST boxes refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. They do not refer to Mach Boxes.

4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg=[GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk=[GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPgk]^T=[GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to Version 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP

option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not.

(In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). NX Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

SPLINE5

Linear Spline

Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points. The interpolating beam supports axial rotation and bending in the yz-plane.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPLINE5	EID	CAERO	AELIST		SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE					

EXAMPLE:

SPLINE5	5	8	12		60			3	
	1.			BOTH					

FIELDS:

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. (Integer > 0)
AELIST	Identification number of an AELIST entry that identifies the aerodynamic boxes whose motions are interpolated using this spline. See Remark 6 when using the Mach Box method. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 8 . (Character, Default=BOTH)

Field	Contents
CID	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (the bending rotations). DTHY is for rotation about the spline's y-axis (torsion); however, it is used for bending of bodies. (Real)
DZ	Linear attachment flexibility. (Real ≥ 0.0, Default=0.0)
DTOR	Torsional flexibility ratio (EI/GJ) for the bending in the zy-plane. This value is ignored for slender bodies since they have no torsion; see Remark 7 . (Real > 0.0; Default=1.0; ignored for CAERO2 bodies.)

REMARKS:

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the interpolating beam (y-axis) is parallel to the x-axis of the aerodynamic coordinate system; the z-axis is taken from the referenced CID and x is made orthogonal.
3. The flexibilities DZ, DTHX and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment.) See the *NX Nastran Aeroelastic Analysis User's Guide* for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE5 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, the AELIST entries refer to the ID numbers of aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. They do not refer to Mach Boxes.
7. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.

8. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg=[GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk=[GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPgk]^T=[GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to Version 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). NX Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

SPOINT

Scalar Point Definition

Defines scalar points.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

EXAMPLE:

SPOINT	3	18	1	4	16	2			

ALTERNATE FORMAT AND EXAMPLE:

SPOINT	ID1	“THRU”	ID2						
SPOINT	5	THRU	649						

FIELDS:

Field	Contents
IDi	Scalar point identification number. (0 < Integer; for “THRU” option, ID1 < ID2)

REMARKS:

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASi, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.

2. All scalar point identification numbers must be unique with respect to all other structural, scalar, and fluid points. However, duplicate scalar point identification numbers are allowed in the input.

3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.

4. If the alternate format is used, all scalar points ID1 through ID2 are defined.

5. For a discussion of scalar points, see “Understanding Scalar Points” in the *NX Nastran User’s Guide*.

SUPORT

Fictitious Support

Defines determinate reaction degrees-of-freedom in a free body.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SUPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

EXAMPLE:

SUPORT	16	215							

FIELDS:

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. The SUPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint (i.e., SPCi entry or PS on the GRID entry).

2. SUPORT and/or SUPORT1 entries are required to perform inertia relief in static analysis (SOL 101).

   In SOL 101, PARAM,INREL,-1 must also be specified or the SUPORTi entries will be treated as constraints.

3. Be careful not to spell SUPORT with two Ps.

4. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

5. From 1 to 24 support degrees-of-freedom may be defined on a single entry.

6. See “Rigid Body Supports” in the *NX Nastran User’s Guide* for a discussion of supported degrees-of-freedom (members of the r-set).

7. An alternative to SUPORT is the SUPORT1 entry, which is requested by the SUPORT1 Case Control command.

8. SUPORT entries are not supported for SOL 106, 114–118, and 129.

SUPORT1

Fictitious Support, Alternate Form

Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPORT1 must be requested by the SUPORT1 Case Control command.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SUPORT1	SID	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

SUPORT1	5	16	215						

FIELDS:

Field	Contents
SID	Identification number of the support set. See Remark 1. (Integer > 0)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. The SUPORT and SUPORT1 entries specify reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint; (i.e., SPCi entry or PS on the GRID entry).
2. SUPORT and/or SUPORT1 entries are required to perform inertia relief in static analysis; i.e., SOL 101.

In SOL 101, PARAM,INREL,-1 must also be specified or the SUPORTi entries will be treated as constraints.

3. SUPORT1 must be requested by the SUPORT1 Case Control command. The degrees-of-freedom specified on SUPORT1 will be combined with those on the SUPORT entry.

4. Be careful not to spell SUPORT with two Ps.

5. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See “Degree-of-Freedom Sets” for a list of these entries.

6. From 1 to 18 support degrees-of-freedom may be defined on a single entry.

7. See “Rigid Body Supports” in the *NX Nastran User’s Guide* for a discussion of supported degrees-of-freedom (members of the r-set).

8. In superelement analysis, SUPORT1 may be specified for points belonging to the residual structure only.

9. SUPORT1 entries are not supported for SOL 106, 114–118, and 129.

SWLDPRM

Parameters for CWELD/CFAST Connectors

Define parameters for CWELD/CFAST connectors.

FORMAT:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

EXAMPLE:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	CHKRUN	1	GSPROJ	17.0					

FIELDS:

Field	Contents
PARAMi	Parameter name. Parameters are listed below.
VALi	Value assigned to parameter.

Parameter Descriptions

CHKRUN Integer ≥ 0; Default=0

Determines if solution continues or ends after connector element generation.

= 0 Solutions runs to completion.

= 1 Solution stops after connector elements are generated.

Field	Contents
GSMOVE	Integer ≥ 0; Default=0 Number of times GS (or XS,YS,ZS) is relocated if the projection to patches A and/or B is not found. GSMOVE only applies to the ELPAT and PARTPAT methods.
GSPROJ	Real; Default=20.0 Allowable angle between the normal projection from GS (or XS,YS,ZS) to patch A, and the normal projection from GS (or XS,YS,ZS) to patch B. Connectors are only created when this angle is less than GSPROJ. The angle check is skipped when GSPROJ=-1.0.
GSTOL	Real ≥ 0.0; Default=Ignored Allowable distance from GS (or XS,YS,ZS) to the intersection of patches A and B. Connectors are only created when the distance is less than GSTOL. Only applies to ELPAT and PARTPAT methods.
NREDIA	0 ≤ Integer ≤ 4; Default=0 Number of times the diameter is reduced in half if the diameter fails to project entirely onto patches A and/or B. The reduced diameter only applies to connection point creation; the CWELD stiffness is calculated with the original value. NREDIA only applies to the ELPAT and PARTPAT methods.
PROJTOL	0.0 ≤ Real ≤ 0.2; Default=0.0 Used to calculate the Tolerance = (PROJTOL) * (Size of shell element), which determines if a projected point located outside of the shell elements in patch A or B is used.
PRTSW	Integer ≥ 0; Default=0 Determines if CWELD/CFAST diagnostic information is written to the .f06 file. = 0 No diagnostic output. = 1 Diagnostic output is written to the .f06 file.

Field	Contents
ANGPAT	Real ≥ 0.0; Default=18 degrees. The maximum angle in degrees between the normal of the master weld element and the normal of any adjacent elements to be considered part of the A and B side patchs. Increasing the angle may generate patches which are very irregular and may not represent a good weld connection.
GHOFF	Integer ≥ 0; Default=0 Determines if a CWELD/CFAST can be created when GHA or GHB points do not project on to an element. = 0 CWELD/CFAST creation will fail if any of the GHA or GHB points do not project on to any element in the patch. = 1 If at least 1 of the GHA or GHB points project onto an element in the patch, the GHA or GHB points that do not project onto the patch will still be associated with the master element.
DISPRT	Integer; Default=2 Determines if displacements are calculated at GA/GB. = 0 Displacements calculated for GA/GB in all cases. If GA/GB are not explicitly defined, they are only written to the .f06 file. = 1 Displacements will not be calculated for GA/GB for the case when GA/GB are not grid points (coordinates generated internally). = 2 Displacements will not be calculated for GA/GB in all cases.

Chapter 18: Bulk Data Entries T—V

TABDMP1

Modal Damping Table

Defines modal damping as a tabular function of natural frequency for modal solutions.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABDMP1	TID	TYPE							
	f1	g1	f2	g2	f3	g3	-etc.-		

EXAMPLE:

TABDMP1	2								
	2.5	0.01057	2.6	0.01362	ENDT				

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
TYPE	Specifies interpretation of damping values. (Character: “G”, “CRIT”, or “Q”; Default is “G”)
	“G” gi interpreted as structural damping values.
	“CRIT” gi interpreted as fraction of critical damping values.
	“Q” gi interpreted as quality or amplification factor values.
fi	Frequency value in cycles per unit time. (Real ≥ 0.0)
gi	Damping value corresponding to frequency fi. (Real)

REMARKS:

1. TABDMP1 bulk entries are used to define modal damping for modal complex eigenvalue analysis (SOL 110), modal frequency response analysis (SOL 111), and modal transient response analysis (SOL 112).

2. TABDMP1 bulk entries are selected with the SDAMPING case control command.

3. Damping values defined on a TABDMP1 bulk entry are interpolated or extrapolated to obtain the damping at the undamped natural frequency of each mode. Depending on the parameter KDAMP, the damping at each undamped natural frequency is used to calculate either the modal viscous damping or the modal structural damping for each mode. The modal viscous damping or modal structural damping is used in the equations of motion for response calculations. For modal transient response analysis (SOL 112), only modal viscous damping is valid. See Remark 10 for additional information on the KDAMP parameter.

4. The frequency values, fi, must be specified in either ascending or descending order, but not both.

5. Any fi or gi entry may be ignored by placing “SKIP” in either of the two fields used for that entry.

6. At least one continuation entry must be specified.

7. The damping at a specific frequency, g(*f*), is either linearly interpolated or linearly extrapolated from the damping values listed in the TABDMP1 bulk entry. To extrapolate, the last two end points are used. No warning messages are issued if table data is input incorrectly.

8. Except at the lowest and highest frequencies in the range of the table, damping values can be discontinuous. At a discontinuity, the damping value used is the average of the discontinuous damping values. For example, in Figure 18-1, the damping value used at f = f3 = f4 is g = (g3 + g4) / 2.

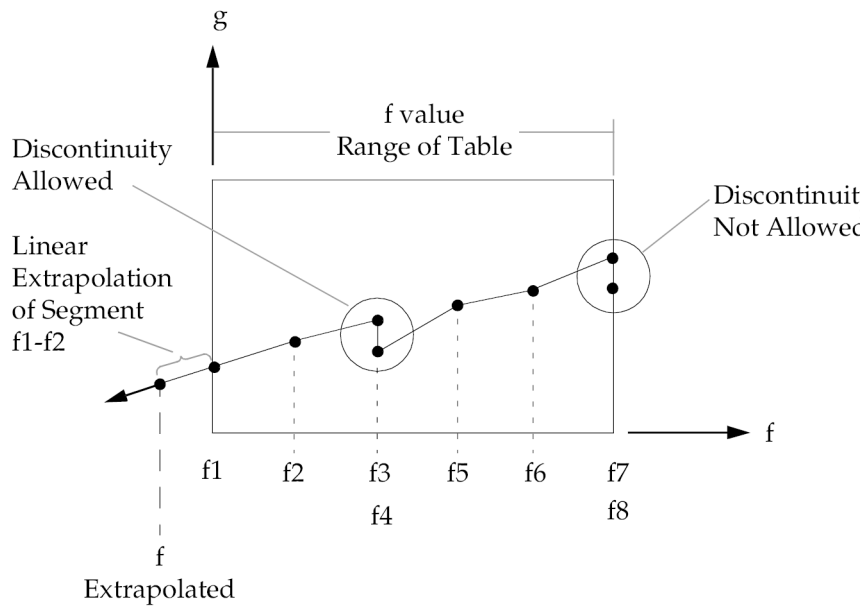


Figure 18-1. Extrapolation of and Discontinuity in Damping Values

9. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

10. PARAM,KDAMP is used to specify whether the damping values listed in the TABDMP1 bulk entry are used to calculate modal viscous damping or modal structural damping. See Table 18-1 for more information on the use of PARAM,KDAMP.

Table 18-1. Use of PARAM,KDAMP

	PARAM,KDAMP,1 (Default)	PARAM,KDAMP,-1
Damping values used to calculate →	Modal viscous damping, b_j	Modal structural damping, G_j
If TYPE = G	$b_j = g(f_j)\ b_{cr} / 2$	$G_j = g(f_j)$
If TYPE = CRIT	$b_j = g(f_j)\ b_{cr}$	$G_j = 2\ g(f_j)$
If TYPE = Q	$b_j = b_{cr} / (2\ g(f_j))$	$G_j = 1 / g(f_j)$
Modal damping force added to equation of motion →	$b_j\ d\zeta_j/dt$	$iG_j\ \zeta_j$

where:
b_j = Modal viscous damping for the jth mode
G_j = Modal structural damping for the jth mode
f_j = Undamped natural frequency for the jth mode
$g(f_j)$ = Damping value at f_j interpolated or extrapolated from gi damping values
ζ_j = Modal coordinate for the jth mode
$b_{cr} = 2\ m_j\ \omega_j$ = Critical damping for the jth mode with $\omega_j = 2\pi\ f_j$
m_j = Modal mass for the jth mode

For additional information, see “Formulation of Dynamic Equations in SubDMAP GMA” in the *NX Nastran User’s Guide*.

TABLED1

Dynamic Load Tabular Function, Form 1

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID	XAXIS	YAXIS	EXTRAP					
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

EXAMPLE:

TABLED1	32			1					
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. See Remark 6. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. See Remark 6. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)

Field	Contents
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate the two starting data points to obtain the table look-up when x < xi, and extrapolate the two ending data points to obtain the table look-up when x > xi. • If EXTRAP = 1, use the value of the table field at the starting data point for the table look-up when x < xi, and use the value of the table field at the ending data point for the table look-up when x > xi.
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-2, discontinuities are allowed only between points x2 through x7. If the y-axis is a LINEAR axis, the value of y at x3 in Figure 18-2 is:

   $y = (y3+y4) / 2$

   If the y-axis is a LOG axis, the value of y at x3 in Figure 18-2 is:

   $$y = \sqrt{y_3 y_4}$$

3. At least one continuation must be specified.

4. Any xi-yi pair may be ignored by placing the character string “SKIP” in either of the two fields.

5. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLED1 uses the algorithm

$$y = y_T(x)$$

where the table lookup at x returns $y_T(x)$.

The table lookup is performed using linear interpolation within the table and, when EXTRAP = 0, linear extrapolation outside the table using the two starting or end points. See Figure 18-2. The algorithms used for interpolation or extrapolation are:

XAXIS	YAXIS	$y_T(x)$
LINEAR	LINEAR	$\frac{xj - x}{xj - xi} yi + \frac{x - xi}{xj - xi} yj$
LOG	LINEAR	$\frac{In(xj / x)}{In(xj / xi)} yi + \frac{In(x / xi)}{In(xj / xi)} yj$
LINEAR	LOG	$\exp\left[\frac{xj - x}{xj - xi} In\ yi + \frac{x - xi}{xj - xi} In\ yj\right]$
LOG	LOG	$\exp\left[\frac{In(xj / x)}{In(xj / xi)} In\ yi + \frac{In(x / xi)}{In(xj / xi)} In\ yj\right]$

where xj and yj follow xi and yi.

No warning messages are issued if table data is input incorrectly.

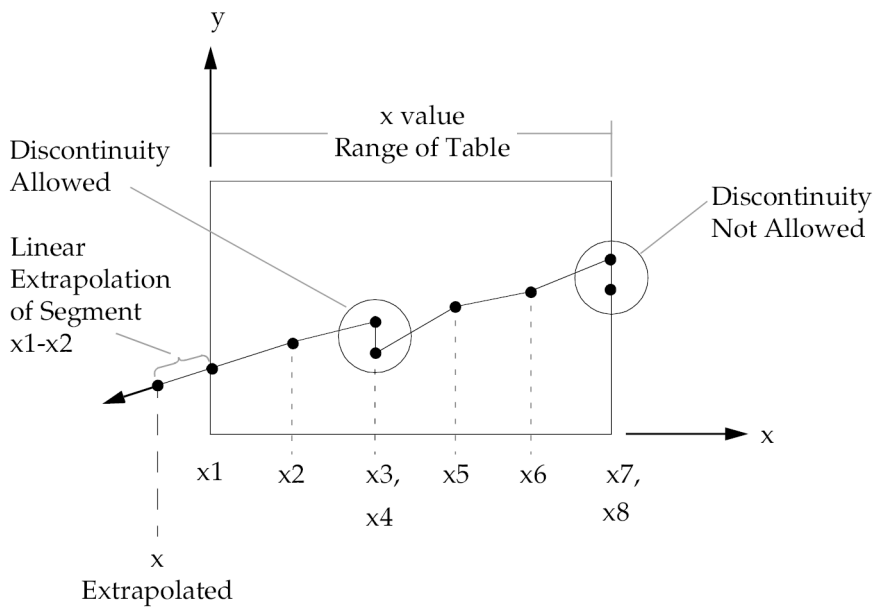


Figure 18-2. Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.

8. For frequency-dependent loads, xi is measured in cycles per unit time.

9. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0.

10. With designed frequency dependent properties in SOL 200, currently only TABLED1 can be used, and the use of TABLED1 for this case is limited to the LINEAR, LINEAR default options for XAXIS and YAXIS. For frequency dependent properties not associated with design variables, other options and other TABLEDi can be used.

REMARKS RELATED TO SOLS 601 AND 701:

1. TABLED1 is referenced by the TID field in TLOAD1 entry to model time-dependent loading. xi is therefore the time value and yi is the multiplier factor for the load.

2. XAXIS and YAXIS are ignored. Both are assumed to be “LINEAR”.

3. Discontinuities are not allowed.

4. No table extrapolation is done beyond the range of specified xi values. Hence, the range of xi values should at least span the solution start and end times. In most cases, x1 = 0.0 should be specified since the solution start time is usually 0.0. The solution end time depends on the time steps defined in the TSTEP entry.

TABLED2

Dynamic Load Tabular Function, Form 2

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1	EXTRAP						
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

EXAMPLE:

TABLED2	15	-10.5							
	1.0	-4.5	2.0	-4.2	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
X1	Table parameter. See Remark 6. (Real)
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0)

- If EXTRAP = 0, extrapolate the two starting data points to obtain the table look-up when x < xi, and extrapolate the two ending data points to obtain the table look-up when x > xi.
- If EXTRAP = 1, use the value of the table field at the starting data point for the table look-up when x < xi, and use the value of the table field at the ending data point for the table look-up when x > xi.

Field	Contents
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-3 discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In Figure 18-3, the value of y at x = x3 is $y = (y3+y4) / 2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLED2 uses the algorithm

$$y = y_T(x - \mathrm{X1})$$

where the table lookup at $(x - \mathrm{X1})$ returns $y_T(x - \mathrm{X1})$.

The table lookup is performed using linear interpolation within the table and, when EXTRAP = 0, linear extrapolation outside the table using the two starting or end points. See Figure 18-3. No warning messages are issued if table data is input incorrectly.

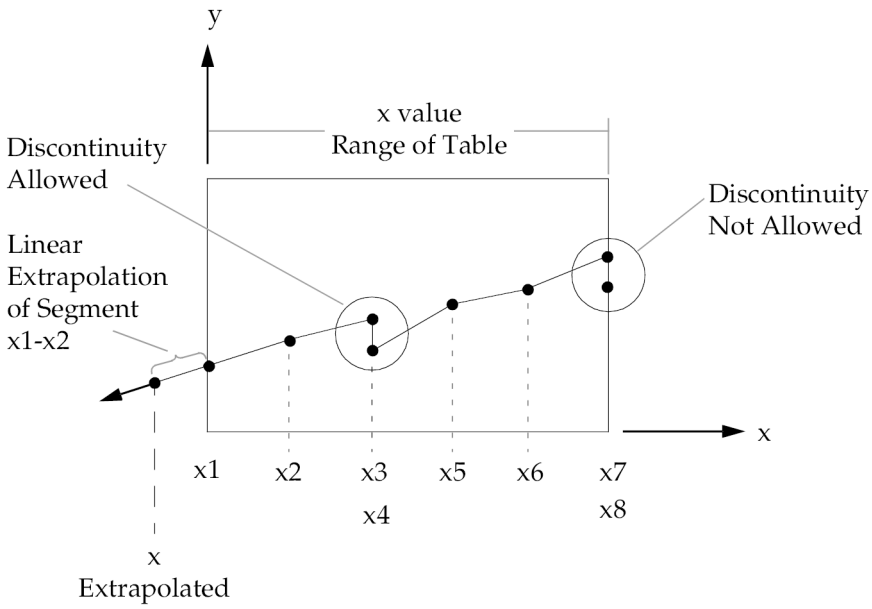


Figure 18-3. Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.

8. For frequency-dependent loads, X1 and xi are measured in cycles per unit time.

REMARKS RELATED TO SOLS 601 AND 701:

1. TABLED2 is referenced by the TID field in TLOAD1 entry to model time-dependent loading. xi is therefore the time value and yi is the multiplier factor for the load.

2. X1 is the delay time (or arrival time of the load), i.e., at time < X1, the load is not active. When using X1 to control the arrival time of an enforced displacement, it is often appropriate to apply it to the deformed configuration. See the DISPOPT parameter on the NXSTRAT bulk entry for information.

3. Discontinuities are not allowed.

4. No table extrapolation is done beyond the range of specified xi values. Hence, the range of xi values should at least span the solution start and end times. In most cases, x1 = 0.0 should be specified since the solution start time is usually 0.0. The solution end time depends on the time steps defined in the TSTEP entry.

TABLED3

Dynamic Load Tabular Function, Form 3

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2	EXTRAP					
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

EXAMPLE:

TABLED3	62	126.9	30.0	1					
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. (Real; X2 ≠ 0.0)
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate the two starting data points to obtain the table look-up when x < xi, and extrapolate the two ending data points to obtain the table look-up when x > xi. • If EXTRAP = 1, use the value of the table field at the starting data point for the table look-up when x < xi, and use the value of the table field at the ending data point for the table look-up when x > xi.
xi, yi	Tabular values. (Real)

Field	Contents
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-4 discontinuities are allowed only between points x2 and x7. Also if y is evaluated at a discontinuity, then the average value of y is used. In Figure 18-4, the value of y at x =x3 is y=(y3+y4) / 2.

3. At least one continuation entry must be present.

4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLED3 uses the algorithm

$$y = y_T\left(\frac{x - \mathrm{X1}}{\mathrm{X2}}\right)$$

where the table lookup at (x – X1)/X2 returns y_T((x – X1)/X2).

The table lookup is performed using linear interpolation within the table and, when EXTRAP = 0, linear extrapolation outside the table using the two starting or end points. See Figure 18-4. No warning messages are issued if table data is input incorrectly.

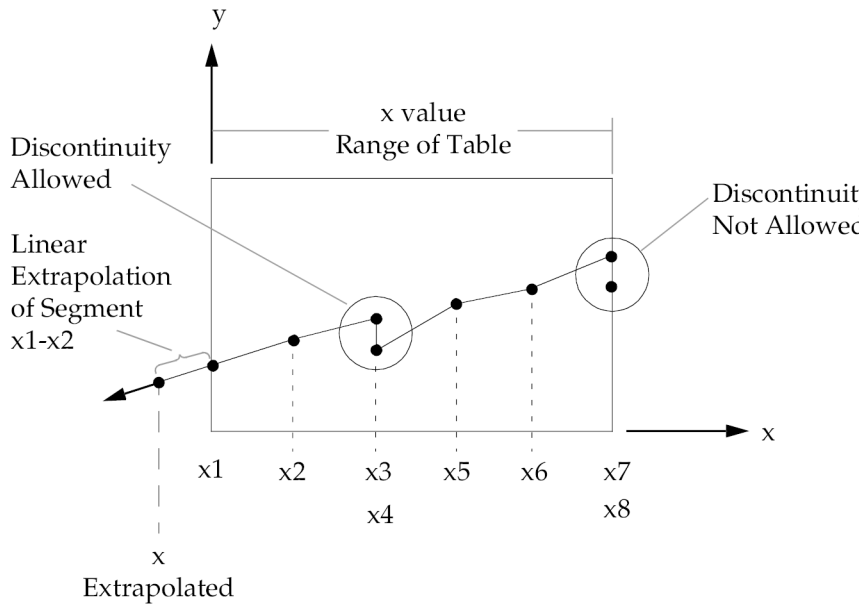


Figure 18-4. Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.

8. For frequency-dependent loads, X1 and xi are measured in cycles per unit time. X2 is dimensionless.

TABLED4

Dynamic Load Tabular Function, Form 4

Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-	“ENDT”	

EXAMPLE:

TABLED4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3<X4)
Ai	Coefficients. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. At least one continuation entry must be specified.

2. The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

3. TABLED4 uses the algorithm

$$y = \sum_{i=0}^{N} \mathrm{Ai}\left(\frac{x - \mathrm{X1}}{\mathrm{X2}}\right)^{i}$$

where *x* is input and *y* is returned. N is the number of pairs.

Whenever *x*< X3, use X3 for *x*; whenever *x*> X4, use X4 for *x*. There are N+1 entries in the table. There are no error returns from this procedure.

4. For frequency-dependent loads, xi and X1 are measured in cycles per unit time. Specify the units for Ai and X2 so that the units of all terms in the summation are consistent.

TABLEM1

Material Property Table, Form 1

Uses tabular data to define temperature-dependent material properties. The value for the material property at a specific temperature is interpolated or extrapolated from the tabular data.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLEM1	TID	XAXIS	YAXIS	EXTRAP					
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

EXAMPLE:

TABLEM1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. See Remark 6. (Character: “LINEAR” or “LOG”; Default=“LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. See Remark 6. (Character: “LINEAR” or “LOG”; Default=“LINEAR”)

Field	Contents
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate the two starting data points to obtain the table look up when x < xi, and extrapolate the two ending data points to obtain the table look up when x > xi. • If EXTRAP = 1, use the value of the table field at the starting data point for the table look up when x < xi, and use the value of the table field at the ending data point for the table look up when x > xi.
xi, yi	Tabular values. xi are values for temperature. yi are values for the material property. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both. No warning message is issued if tabular data is input incorrectly.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-5, discontinuities are allowed only between points x2 through x7.

   If the y-axis is a LINEAR axis, the value of y at x3 in Figure 18-5 is:

   $y = (y_3 + y_4) / 2$

   If the y-axis is a LOG axis, the value of y at x3 in Figure 18-5 is:

   $$y = \sqrt{y_3 y_4}$$

3. At least one continuation entry must be specified.

4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. The tabular data is interpolated for values of *x* that lie within the range of the tabular data and, when EXTRAP = 0, the tabular data is extrapolated from the two starting or end points for values of *x* that lie outside the range of the tabular data.

   The value *y* that the software returns for the material property at temperature *x* is:

$$y = y_T(x)$$

   where $y_T(x)$ is the table look up at *x*.

   The equations that the software uses to interpolate or extrapolate the tabular data for each combination of XAXIS and YAXIS options are:

XAXIS option	YAXIS option	Table look up value, $y_T(x)$
LINEAR	LINEAR	$\frac{\mathrm{xj} - x}{\mathrm{xj} - \mathrm{xi}}\,\mathrm{yi} + \frac{x - \mathrm{xi}}{\mathrm{xj} - \mathrm{xi}}\,\mathrm{yj}$
LOG	LINEAR	$\frac{\mathrm{In}(\mathrm{xj}/x)}{\mathrm{In}(\mathrm{xj}/\mathrm{xi})}\mathrm{yi} + \frac{\mathrm{In}(x/\mathrm{xi})}{\mathrm{In}(\mathrm{xj}/\mathrm{xi})}\mathrm{yj}$
LINEAR	LOG	$\exp\left[\frac{\mathrm{xj} - x}{\mathrm{xj} - \mathrm{xi}}\mathrm{In\ yi} + \frac{x - \mathrm{xi}}{\mathrm{xj} - \mathrm{xi}}\mathrm{In\ yj}\right]$
LOG	LOG	$\exp\left[\frac{\mathrm{In}(\mathrm{xj}/x)}{\mathrm{In}(\mathrm{xj}/\mathrm{xi})}\mathrm{In\ yi} + \frac{\mathrm{In}(x/\mathrm{xi})}{\mathrm{In}(\mathrm{xj}/\mathrm{xi})}\mathrm{In\ yj}\right]$

   where xj and yj follow xi and yi.

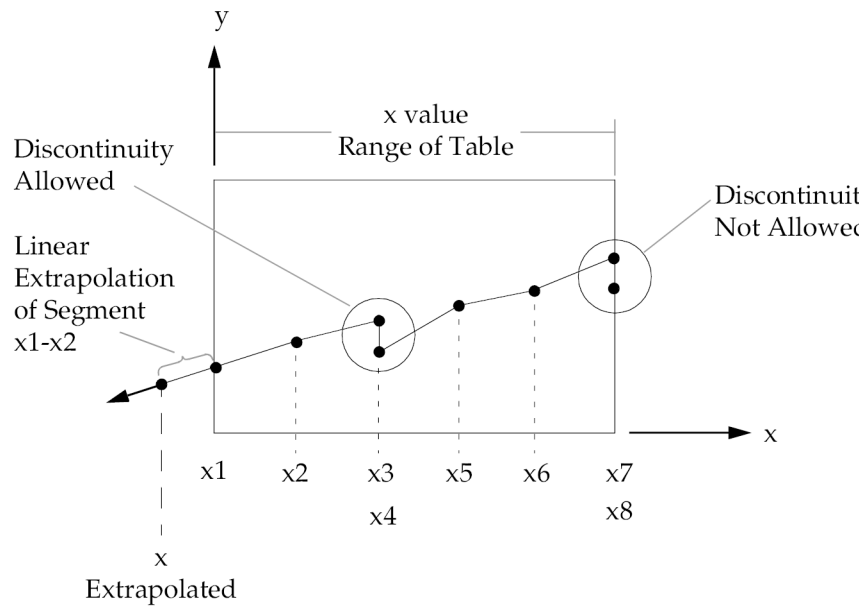


Figure 18-5. Example of Table Extrapolation and Discontinuity

7. When you define a material property using a TABLEM1 entry, the software ignores the value of that material property on the corresponding MATi entry.

8. If an axis is logarithmic, the tabular values on the axis must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0.

REMARKS RELATED TO SOLS 601 AND 701:

1. YAXIS is only used when TABLEM1 is referenced by the MATTC entry. Otherwise, XAXIS and YAXIS are ignored and both are assumed to be “LINEAR”.

2. Discontinuities are not allowed.

3. No table extrapolation is done beyond the range of specified xi values. Hence, the xi values should at least span the range of applied temperatures.

TABLEM2

Material Property Table, Form 2

Uses shifted tabular data to define temperature-dependent material properties. The value for the material property at a specific temperature is the product of the interpolated or extrapolated table look up value and the value for the material property as specified on the corresponding MATi entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLEM2	TID	X1	EXTRAP						
	x1	y1	x2	y2	x3	y3	-etc.-		

EXAMPLE:

TABLEM2	15	-10.5							
	1.0	-4.5	2.0	-4.5	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
X1	Shift parameter. (Real)

Field	Contents
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate the two starting data points to obtain the table look up when x < xi, and extrapolate the two ending data points to obtain the table look up when x > xi. • If EXTRAP = 1, use the value of the table field at the starting data point for the table look up when x < xi, and use the value of the table field at the ending data point for the table look up when x > xi.
xi, yi	Tabular values. xi are values for temperature. yi are values for the material property. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both. No warning message is issued if the tabular data is input incorrectly.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-6 discontinuities are allowed only between points x2 through x7.

   If y is evaluated at a discontinuity, then the average value of y is used. For example, the value of y at x3 = (x – X1) in Figure 18-6 is:

   $y = (y_3 + y_4) / 2$

3. At least one continuation entry must be specified.

4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. The tabular data is linearly interpolated for values of (x – X1) that lie within the range of the tabular data and, when EXTRAP = 0, the tabular data is linearly extrapolated from the two starting or end points for values of (x – X1) that lie outside the range of the tabular data.

   The value y that the software returns for the material property at temperature x is:

$$y = zy_T(x - X1)$$

where $y_T(x - X1)$ is the table look up at $(x - X1)$, and z is the value of the material property on the corresponding MATi entry.

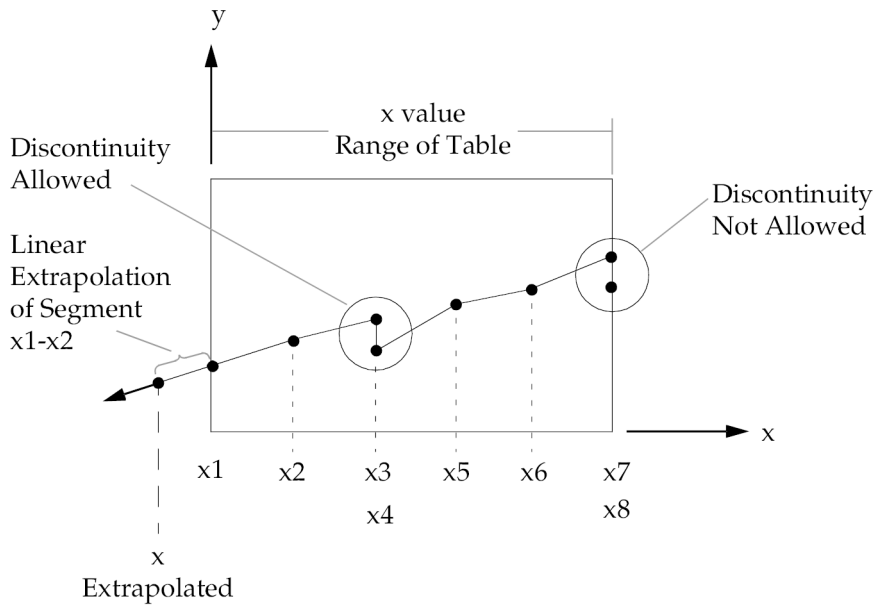


Figure 18-6. Example of Table Extrapolation and Discontinuity

TABLEM3

Material Property Table, Form 3

Uses shifted and scaled tabular data to define temperature-dependent material properties. The value for the material property at a specific temperature is the product of the interpolated or extrapolated table look up value and the value for the material property as specified on the corresponding MATi entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLEM3	TID	X1	X2	EXTRAP					
	x1	y1	x2	y2	x3	y3	-etc.-		

EXAMPLE:

TABLEM3	62	126.9	30.0	1					
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
X1	Shift parameter. See Remark 6. (Real; X2 ≠ 0.0)
X2	Scaling parameter. See Remark 6. (Real; X2 ≠ 0.0)

Field	Contents
EXTRAP	Extrapolation option. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate the two starting data points to obtain the table look up when x < xi, and extrapolate the two ending data points to obtain the table look up when x > xi. • If EXTRAP = 1, use the value of the table field at the starting data point for the table look up when x < xi, and use the value of the table field at the ending data point for the table look up when x > xi.
xi, yi	Tabular values. xi are values for temperature. yi are values for the material property. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both. No warning message is issued if the tabular data is input incorrectly.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-7 discontinuities are allowed only between points x2 through x7.

   If y is evaluated at a discontinuity, then the average value of y is used. For example, the value of y at x3 = (x – X1) / X2 in Figure 18-7 is:

   $y = (y_3 + y_4) / 2$

3. At least one continuation entry must be specified.

4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. The tabular data is linearly interpolated for values of (x – X1) / X2 that lie within the range of the tabular data and, when EXTRAP = 0, the tabular data is linearly extrapolated from the two starting or end points for values of (x – X1) / X2 that lie outside the range of the tabular data.

   The value y that the software returns for the material property at temperature x is:

$$y = zy_T\left(\frac{x - X1}{X2}\right)$$

where $y_T((x - X1) / X2)$ is the table look up at $(x - X1) / X2$, and z is the value of the material property on the corresponding MATi entry.

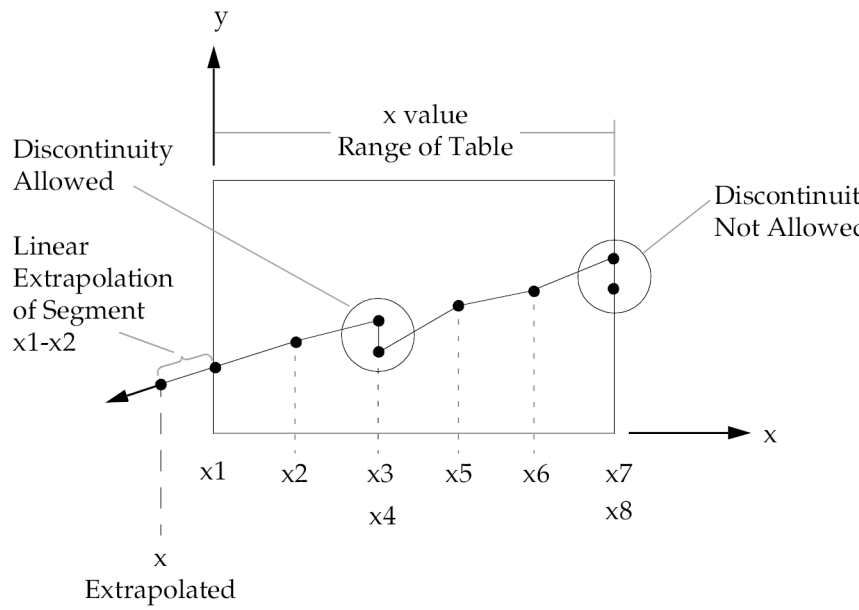


Figure 18-7. Example of Table Extrapolation and Discontinuity

TABLEM4

Material Property Table, Form 4

Uses a power series to define temperature-dependent material properties. The value for the material property at a specific temperature is the product of the power series sum evaluated at that temperature, and the value for the material property as specified on the corresponding MATi entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLEM4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

EXAMPLE:

TABLEM4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
X1, X2, X3, X4	Table parameters. (Real; X2 ≠ 0.0; X3 < X4)
Ai	Coefficients. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. At least one continuation entry must be specified.

2. The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

3. The value *y* that the software returns for the material property at temperature *x* is:

$$y = z \sum_{i=0}^{N} \mathrm{Ai}\left(\frac{x - \mathrm{X1}}{\mathrm{X2}}\right)^{i}$$

where *N* is the number of coefficients, and *z* is the value of the material property on the corresponding MATi entry.

Whenever $x < \mathrm{X3}$, use X3 for *x*; whenever $x > \mathrm{X4}$, use X4 for *x*.

There are (N + 1) entries in the table.

There are no error returns from this procedure.

TABLES1

Material Property Table, Form 1

Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLES1	TID								
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

EXAMPLE:

TABLES1	32								
	0.0	0.0	.01	10000.	.02	15000.	ENDT		

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

REMARKS:

1. xi must be in either ascending or descending order, but not both.
2. At least one continuation entry must be present.

3. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.

4. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.

5. TABLES1 is used to input a curve in the form of

$$y = y_T(x)$$

where *x* is input to the table and *y* is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See Figure 18-8. No warning messages are issued if table data is input incorrectly.

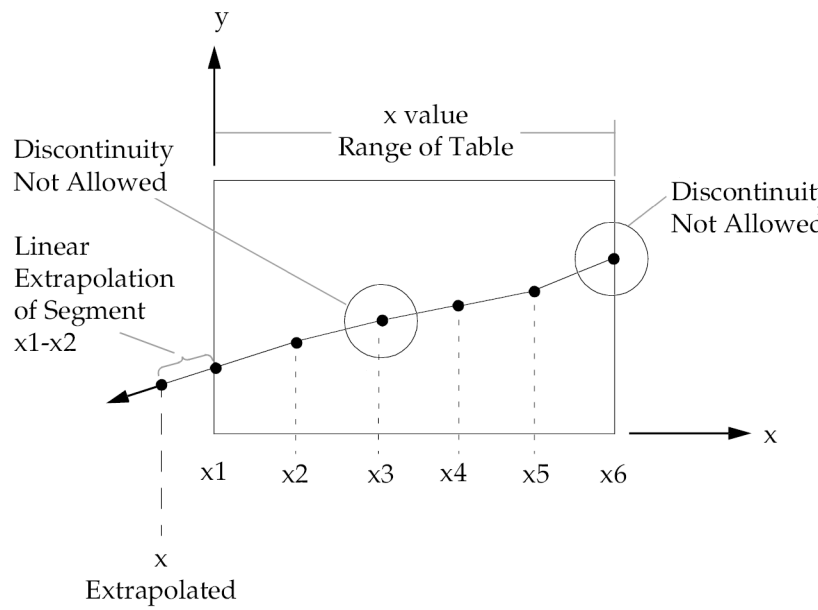


Figure 18-8. Example of Table Extrapolation

6. Discontinuities are not allowed.

REMARKS RELATED TO SOLS 601 AND 701:

1. By default, the table is extended by linear extrapolation using the last two end points. XTCURVE=0 may be specified in NXSTRAT to choose not to extend the table to allow for element rupture at the last xi value which defines the rupture plastic strain when the multilinear plastic material is used (i.e., TABLES1 is referenced by TID field in MATS1 with TYPE = “PLASTIC”), Upon rupture, an element is removed from the model. If DTDELAY is specified in NXSTRAT entry, the contribution from the element stiffness is gradually reduced to zero over time DTDELAY.

2. By default, xi, yi are assumed to be true stress-strain values. In case the input is in engineering stress-strain values, specifying CVSSVAL=1 in NXSTRAT provides a convenient way to convert the engineering stress-strain values to true stress-strain values for use in the analysis.

3. The stress and strain measures for both input and output is described in section 3.1 of the *Advanced Nonlinear Theory and Modeling Guide*.

TABLEST

Table of tables

Uses a series of tables to represent a functional relationship that has two independent variables.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABLEST	TID	EXTRAP							
	T1	TID1	T2	TID2	T3	-etc.-			

EXAMPLE:

TABLEST	101	1							
	150.0	10	175.0	20	ENDT				

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
EXTRAP	Extrapolation option. See Remarks 7 and 8. (Integer: 0 or 1; Default = 0) • If EXTRAP = 0, extrapolate for value when temperature or displacement or force is outside the range of Ti. • If EXTRAP = 1, use the value of the nearest table field when temperature or displacement or force is outside the range of Ti.
Ti	Temperature or displacement or force that corresponds to the table referenced in field TIDi. (Real)
TIDi	Table identification numbers of TABLES1 or TABLEDi bulk entries. (Integer > 0)

REMARKS:

1. TIDi must be unique with respect to all TABLES1 or TABLEDi and TABLEST table identification numbers.

2. Temperature or displacement or force values must be listed in ascending order.

3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.

4. TABLEST bulk entries are referenced by:

   - MATS1 bulk entries to define nonlinear elastic (TYPE = “NLELAST”) materials.

   - PBEAR bulk entries to define bearing stiffness or viscous damping (TYPE = ”KD”,”KF”,BD”,”BF”) as a function of rotor speed and displacement or rotor speed and force. For this usage, the number of TIDi entries is limited to 100.

5. A function of two variables such as $z = f(x,y)$ can be approximated by a series of N single-variable functions:

   $z_1 = f_1(x) = f(x, Y_1)$
   $z_2 = f_2(x) = f(x, Y_2)$
   .....
   $z_i = f_i(x) = f(x, Y_i)$
   .....
   $z_N = f_N(x) = f(x, Y_N)$

   where Y_i are constants.

   The single-variable functions, $f_i(x)$, are approximated by the tabular data listed on the TABLES1 and TABLEDi bulk entries that are referenced in the TIDi fields of the TABLEST bulk entry. The constants, Y_i, are given by the values listed in the Ti fields of the TABLEST bulk entry.

6. The table lookup value for $f(X_0, Y_0)$ when Y_0 is within the range of data is obtained as follows:

   a. The bounding table fields are identified. The bounding table fields are the table fields, $f_j(x)$ and $f_k(x)$, whose fixed variables, Y_j and Y_k, are in closest proximity to Y_0 such that $Y_j < Y_0 < Y_k$.

   b. The tabular data for each bounding table field is interpolated to obtain $f_j(X_0)$ and $f_k(X_0)$.

   c. Points $(f_j(X_0), Y_j)$ and $(f_k(X_0), Y_k)$ are interpolated to find the value corresponding to Y_0.

7. The table lookup value for $f(X_0,Y_0)$ when Y_0 is outside the range of data is obtained as follows:

   - If EXTRAP = 0

     a. The two table fields in closest proximity are identified. These table fields are the table fields, $f_j(x)$ and $f_k(x)$, whose fixed variables, Y_j and Y_k, are in closest proximity to Y_0 such that $Y_0 > Y_j > Y_k$ or $Y_0 < Y_j < Y_k$.

     b. The tabular data for these table fields is interpolated to obtain $f_j(X_0)$ and $f_k(X_0)$.

     c. Points $(f_j(X_0),Y_j)$ and $(f_k(X_0),Y_k)$ are extrapolated to find the value corresponding to Y_0.

   - If EXTRAP = 1

     a. The table field in closest proximity is identified. This table field is the table field, $f_j(x)$, whose fixed variable, Y_j is in closest proximity to Y_0.

     b. The tabular data for the table field is interpolated to obtain $f_j(X_0)$.

     c. $f_j(X_0)$ is used as the lookup value.

8. When the TABLEST bulk entry is referenced by a PBEAR bulk entry, the software behaves as if EXTRAP = 1, regardless of whether the EXTRAP field is left blank or EXTRAP = 0 is specified.

9. SOL 401 always uses EXTRAP=1 on the TABLEST entry. EXTRAP=0 is not supported.

REMARKS RELATED TO SOLS 601 AND 701:

1. TABLEST may be referenced by MATS1 entries that define elasto-plastic (TYPE=”PLASTIC”) materials, but not nonlinear elastic (TYPE=“NLELAST”) materials.

TABRND1

Power Spectral Density Table

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABRND1	TID	XAXIS	YAXIS						
	f1	g1	f2	g2	f3	g3	-etc.-		

EXAMPLE:

TABRND1	3								
	2.5	.01057	2.6	.01362	ENDT				

FIELDS:

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: “LINEAR” or “LOG”; Default=“LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: “LINEAR” or “LOG”; Default=“LINEAR”)
fi	Frequency value in cycles per unit time. (Real ≥ 0.0)
gi	Power spectral density. (Real)

REMARKS:

1. The fi must be in either ascending or descending order, but not both.

2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 18-9 discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In Figure 18-9, the value of g at f=f3 is g=(g3+g4) / 2. If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as

   $$y = \sqrt{y3y4}.$$

3. At least two entries must be present.

4. Any fi-gi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABRND1 uses the algorithm

   $$g = g_T(f)$$

   where *f* is input to the table and *g* is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See Figure 18-9. No warning messages are issued if table data is input incorrectly.

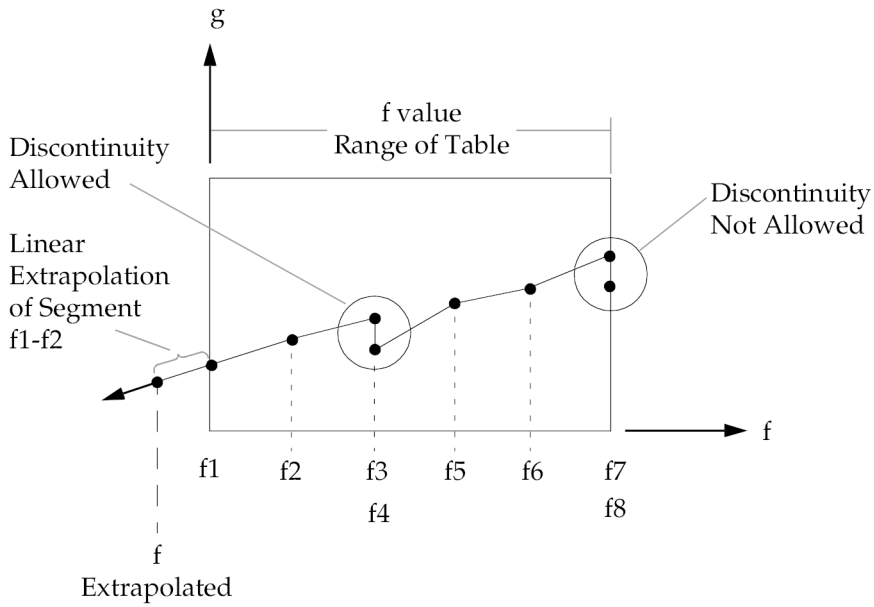


Figure 18-9. Example of Table Extrapolation and Discontinuity

7. For auto spectral density, the value of *g* returned must be greater than or equal to zero, as shown in Remark 6 .

8. Tabular values on an axis if XAXIS or YAXIS=LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0.

9. The algorithms used are:

XAXIS	YAXIS	f(x)
LINEAR	LINEAR	$\frac{f_{i+1}-f}{f_{i+1}-f_i}g_i + \frac{f-f_i}{f_{i+1}-f_i}g_{i+1}$
LOG	LINEAR	$\frac{\mathrm{In}(f_{i+1}/f)}{\mathrm{In}(f_{i+1}/f_i)}g_i + \frac{\mathrm{In}(f/f_i)}{\mathrm{In}(f_{i+1}/f_i)}g_{i+1}$

XAXIS	YAXIS	f(x)
LINEAR	LOG	$\exp\left[\frac{f_{i+1}-f}{f_{i+1}-f_i}\text{In } g_i + \frac{f-f_i}{f_{i+1}-f_i}\text{In } g_{i+1}\right]$
LOG	LOG	$\exp\left[\frac{\text{In}(f_{i+1}/f)}{\text{In}(f_{i+1}/f_i)}\text{In } g_i + \frac{\text{In}(f/f_i)}{\text{In}(f_{i+1}/f_i)}\text{In } g_{i+1}\right]$

where $f_i < f < f_{i+1}$

TABRNDG

Gust Power Spectral Density

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TABRNDG	TID	TYPE	L/U	WG					

EXAMPLE:

TABRNDG	1020	1	1.3	.1					

FIELDS:

Field	Contents
TID	Table identification number. (Integer >0)
TYPE	PSD type: von Karman (TYPE=1) or Dryden model (TYPE=2). (Integer=1 or 2)
L/U	Scale of turbulence divided by velocity (units of time). See L/U in Remark 2 . (Real)
WG	Root-mean-square gust velocity. (Real)

REMARKS:

1. This entry must be referenced by a RANDPS entry.
2. The power spectral density is given by

$$S_q(\omega) = 2(\mathrm{WG})^2(\mathrm{L}/\mathrm{U})\frac{1 + 2(p+1)k^2(\mathrm{L}/\mathrm{U})^2\omega^2}{[1 + k^2(\mathrm{L}/\mathrm{U})^2\omega^2]^{p+3/2}}$$

where

Type	p	k
1=von Karman	1/3	1.339
2=Dryden	1/2	1.0

and $\omega=2\pi f$. The units of $S_q(\omega)$ are velocity squared per frequency (f).

3. Other power spectral density functions may be defined using the TABRND1 entry.

TABVE

Define Coefficients for Viscoelastic Material (SOL 601 only)

Defines series of decay coefficients and modulii used for viscoelastic material definition.

FORMAT

1	2	3	4	5	6	7	8	9	10
TABVE	TID	MOD0							
	decay1	mod1	decay2	mod2	decay3	mod3	-etc-		

EXAMPLE

TABVE	3	1.5E-6							
	0.5	90.0	1.0	150.0	ENDT				

FIELDS:

Field	Contents
TID	Table identification number referenced by MATVE entry. (Integer > 0).
MOD0	The long-time modulus term. (Real; Default=0.0)
decay(i)	The optional i^{th} term of the decay coefficient in the Prony series. (Real)
mod(i)	The optional i^{th} term of the modulus in the Prony series. (Real)

REMARKS:

1. TABVE entry may have only one line with non-zero MOD0.

2. The maximum number of terms allowed in the Prony series is 15.

TEMP

Grid Point Temperature Field

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

EXAMPLE:

TEMP	3	94	316.2	49	219.8				

FIELDS:

Field	Contents
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ti	Temperature. (Real)

REMARKS:

1. In the static solution sequences, the temperature set ID(SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. From one to three grid point temperatures may be defined on a single entry.

4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.

5. If the element material is temperature dependent, its properties are evaluated at the average temperature.

6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined. Gauss point temperatures are averaged for solid elements instead of grid point temperature.

7. For steady state heat transfer analysis, this entry together with the TEMPD entry supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT)=SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMPD entry supplies the initial condition temperatures. The Case Control command IC=SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.

9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See “Buckling Analysis in SubDMAP MODERS” in the *NX Nastran User’s Guide* and the *NX Nastran Basic Nonlinear Analysis User’s Guide*.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply constant temperature loads, SID is selected by Case Control command TEMP=SID for both static and transient analyses. TEMP(INIT)=SID is used to specify initial temperatures.

2. To apply time-dependent temperature loads, SID is referenced by the field EXCITEID=SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

TEMPBC

Grid Point Temperatures

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	GID1	TEMP2	GID2	TEMP3	GID3	

EXAMPLE:

TEMPBC	10	STAT	100.0	1	100.0	2	100.0	3	

ALTERNATE FORMAT AND EXAMPLE:

TEMPBC	SID	TYPE	TEMP1	GID1	“THRU”	GID2	“BY”	INC	

TEMPBC	20	STAT	100.0	4	THRU	50	BY	2	

FIELDS:

Field	Contents
SID	Temperature set identification number. (Integer > 0)
TYPE	Type of temperature boundary condition. See Remarks. (Character; Default=“STAT”): STAT – Constant temperature boundary condition TRAN – Time-varying temperature boundary condition

Field	Contents
TEMPi	Temperature (Real)
GIDi	Grid point identification number. (Integer > 0 or “THRU” or “BY”)
INC	Grid point number increment. (Integer)

REMARKS:

1. For a constant Boundary Condition (TYPE=“STAT”), the temperature boundary load set (SID) is selected in the Case Control Section (SPC=SID). TYPE=“STAT” may be used in both steady-state (SOL 153) and transient analysis (SOL 159).

2. For transient analysis (SOL 159), a constant boundary condition should be specified using the SPC Bulk Data entry.

3. For a time-varying boundary condition (TYPE=“TRAN”), SID is referenced by a TLOADi Bulk Data entry through the DAREA specification. TYPE=“TRAN” is permitted only in transient analysis (SOL 159). A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD=SID) for use in transient analysis.

4. In the alternate format, TEMP1 is the nodal temperature for the grid points GID1,GID1+INC,...,GID2. If “BY” and INC are not specified, then the grid point number increment is unity.

5. If TYPE=“STAT”, then no SPCi Bulk Data entries may be specified.

6. If TYPE=“TRAN”, then no CELAS2 or DAREA Bulk Data entries may be specified. Also, “U” must be specified in the CONV field on the TSTEPNL entry to obtain accurate results.

7. All TEMPBC entries in the Bulk Data Section must indicate either TYPE=“STAT” or TYPE=“TRAN” but not both.

REMARKS RELATED TO SOL 601:

1. TYPE=”TRAN” can be used in both steady-state (SOL 601,153) and transient analysis (SOL 601,159).

2. Both TYPE=”STAT” and TYPE=”TRAN” may be specified in one model.

TEMPD

Grid Point Temperature Field Default

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

EXAMPLE:

TEMPD	1	216.3							

FIELDS:

Field	Contents
SIDi	Temperature set identification number. (Integer > 0)
Ti	Default temperature value. (Real)

REMARKS:

1. For structural analysis in the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. From one to four default temperatures may be defined on a single entry.

4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.

5. If the element material is temperature dependent, its properties are evaluated at the average temperature.

6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data is defined.

7. For steady-state heat transfer analysis, this entry together with the TEMP entry supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT)=SID requests selection of this entry. The temperature values specified here must be coincident with any temperatures boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMP entry supplies the initial condition temperatures. The Case Control command IC=SID request selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.

9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See “Buckling Analysis in SubDMAP MODERS” in the *NX Nastran User’s Guide* and the *NX Nastran Basic Nonlinear Analysis User’s Guide*.

10. For partitioned Bulk Data superelements and auxiliary models, TEMPD must be specified in all partitioned Bulk Data Sections.

REMARKS RELATED TO SOLS 601 AND 701:

1. To apply constant temperature loads, SID is selected by Case Control command TEMP=SID for both static and transient analyses. TEMP(INIT)=SID is used to specify initial temperatures.

2. To apply time-dependent temperature loads, SID is referenced by the field EXCITEID=SID in the TLOAD1 entry. Time-dependent loads are selected by Case Control command DLOAD.

TEMPEX

Time independent temperature set defined in a .bun file for SOL 401.

Defines a time independent temperature set using a .bun file.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMPEX	SID	UNID							

EXAMPLE:

TEMPEX	200	21							

FIELDS:

Field	Contents
SID	Unique identification number specified in a TEMP(LOAD) or TEMP(INIT) case control command. (Integer>0)
UNID	Fortran unit number of an external .bun file included on an ASSIGN statement. For example, ASSIGN BUN='temperature0.bun' UNIT=22. (Integer>0)

REMARKS:

1. SID must be unique to all other TEMPEX entries.
2. The SID of the TEMPEX bulk entry can only be selected with a TEMP(LOAD) or TEMP(INIT) case control command. The DTEMP case control command cannot be used to select the TEMPEX bulk entry.

TEMPP1

Plate Element Temperature Field, Form 1

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading, temperature-dependent material properties, or stress recovery.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	TBAR	TPRIME	T1	T2			
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

EXAMPLE:

TEMPP1	2	24	62.0	10.0	57.0	67.0			
	26	21	19	30					

ALTERNATE FORMAT AND EXAMPLE OF CONTINUATION ENTRY:

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	1	THRU	10	30	THRU	61			

FIELDS:

Field	Contents
SID	Temperature set identification number. (Integer > 0)
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer > 0 or the continuation entries may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi and EIDj < EIDk.)

Field	Contents
TBAR	Temperature at the element's reference plane as defined by ZOFFS on the connection entry. (Real)
TPRIME	Effective linear thermal gradient. Not used for membranes. (Real)
T1, T2	Temperatures for an additional membrane stress calculation at points defined on the element property entry (*Z*1 and *Z*2 field on PSHELL entry). See Remark 9. T1 may be specified on the lower surface and T2 on the upper surface for the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR elements. These data are not used for membrane elements. See Remark 11. If T1 and T2 are blank, they are ignored, and the additional stress calculation does not occur. (Real)

REMARKS:

1. In the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. If continuation entries are present, EID1 and elements specified on the continuation entry are used. Elements must not be specified more than once.

4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.

5. For temperature field other than a constant gradient, the "effective gradient" for a homogeneous plate is

$$\text{TPRIME} = \frac{1}{I}\int_z T(z)z\,dz$$

where *I* is the bending inertia and *z* is the distance from the neutral surface in the positive normal direction.

6. The “average” temperature for a homogeneous plate is

$$\mathrm{TBAR} = \frac{1}{Volume} \int_{Volume} T dVolume$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature TBAR.

8. Large “THRU” ranges will lead to System Fatal Message 3008 (“Insufficient Core”) and should be avoided, particularly for open sets.

9. If the element material is nonlinear then T1 and T2 should be left blank (see the MATS1 entry).

10. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See “Buckling Analysis in SubDMAP MODERS” in the *NX Nastran User’s Guide* and the *NX Nastran Basic Nonlinear Analysis User’s Guide*.

11. If T1 or T2 are defined, an additional membrane stress is calculated by

    $\{\Delta\sigma\} = -[G_e]\{\alpha_e\}(T - T_{fiber})$

    where T is the explicitly entered temperatures T1,T2 and T_{fiber} is the outer fiber temperatures Tfiber1,Tfiber2 computed by Tfiber=TBAR + z·TPRIME (z is the distance from the center fiber). $[G_e]$ and $\{\alpha_e\}$ are evaluated at TBAR. The thermal expansion due to T1,T2 is assumed to be completely restrained by elastic stiffness.

TEMPP3

Plate Element Temperature Field, Form 3

TEMPP3 is no longer available. Use TEMPP1.

TEMPRB

One-Dimensional Element Temperature Field

Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	TP1A	TP1B	TP2A	TP2B	
	TCA	TDA	TEA	TFA	TCB	TDB	TEB	TFB	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

EXAMPLE:

TEMPRB	200	1	68.0	23.0	0.0	28.0		2.5	
	68.0	91.0	45.0		48.0	80.0	20.0		
	9	10							

ALTERNATE FORMAT AND EXAMPLE OF CONTINUATION ENTRY:

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	2	THRU	4	10	THRU	14			

FIELDS:

Field	Contents
SID	Temperature set identification number. (Integer > 0)

Field	Contents
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer > 0 or the second continuation entry may have “THRU” in fields 3 and/or 6 in which case EID2 < EIDi and EIDj < EIDk.)
TA, TB	Temperature at end A and end B on the neutral axis. (Real)
TPij	Effective linear gradient in direction i on end j; used with CBAR, CBEAM, and CBEND only. (Real)
Tij	Temperature at point i as defined on the PBAR, PBEAM, and PBEND entries at end j. This data is used for stress recovery only with CBAR, CBEAM, and CBEND exclusively. (Real)

REMARKS:

1. In the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.

3. If at least one nonzero or nonblank Tij is present, the point temperatures given are used for stress recovery. If no Tij values are given, linear temperature gradients are assumed for stress recovery. The Tij values are not used in the calculation of differential stiffness.

4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used. Elements must not be specified more than once.

5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.

6. The effective thermal gradients in the element coordinate system for the CBAR element are defined by the following integrals over the cross section. For end “A” (end “B” is similar).

$$\mathrm{TA} = \frac{1}{A}\int_A TA(y, z)dA$$

$$\mathrm{TP1A} = \frac{I_2}{\Delta}\int_A (y - y_n)TA(y, z)dA - \frac{I_{12}}{\Delta}\int_A (z - z_n)TA(y, z)dA$$

$$\mathrm{TP2A} = \frac{I_1}{\Delta}\int_A (z - z_n)TA(y, z)dA - \frac{I_{12}}{\Delta}\int_A (y - y_n)TA(y, z)dA$$

$$\Delta = I_1 I_2 - I_{12}^2$$

if I_{12}=0

$$\mathrm{TP1A} = \frac{1}{I_1}\int_A (y - y_n)TA(y, z)dA$$

$$\mathrm{TP2A} = \frac{1}{I_2}\int_A (z - z_n)TA(y, z)dA$$

where *TA*(*y*, *z* is the temperature at point y,z (in the element coordinate system) at end “A” of the bar. See the CBAR entry description for the element coordinate system: I_1, I_2, and$I_{12}$are the moments of inertia about the z and y axes, respectively. The temperatures are assumed to vary linearly along the length (x-axis). Note that if the temperature varies linearly over the cross section, then TP1A, TP1B, TP2A and TP2B are the actual gradients.

7. If the element material is temperature-dependent, the material properties are evaluated at the average temperature (*TA*+*TB*) / 2.

8. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See “Buckling Analysis in SubDMAP MODERS” in the *NX Nastran User’s Guide* and the *NX Nastran Basic Nonlinear Analysis User’s Guide*.

9. If any T_y is specified the stresses computed by the effective gradient are corrected by Δσ such that:

$$\sigma = \sigma\Big|_{T_A + y^{TP1A} + z^{TP2A}} + \Delta\sigma$$

where Δσ is in the form

$$\Delta\sigma = -\alpha E[T_{CA} - T_o - C_1 \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBAR and CBEAM

$$\Delta\sigma = -\alpha E[T_{CA} - T_o - (C_1 + \Delta N) \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBEND.

TF

Dynamic Transfer Function

Defines a dynamic transfer function of the form:

$$(\mathrm{B0} + \mathrm{B1} \cdot p + \mathrm{B2} \cdot p^2)u_d + \sum_i (\mathrm{A0}(i) + \mathrm{A1}(i)p + \mathrm{A2}(i)p^2)u_i = 0$$

Equation 18-1.

Can also be used as a means of direct matrix input. See Remark 4 .

FORMAT:

1	2	3	4	5	6	7	8	9	10
TF	SID	GD	CD	B0	B1	B2			
	G(1)	C(1)	A0(1)	A1(1)	A2(1)	-etc.-			

EXAMPLE:

TF	1	2	3	4.0	5.0	6.0			
	3	4	5.0	6.0	7.0				

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
GD, G(i)	Grid, scalar, or extra point identification numbers. (Integer > 0)
CD, C(i)	Component numbers. (Integer zero or blank for scalar or extra points, any one of the Integers 1 through 6 for a grid point.)

Field	Contents
B0, B1, B2A0(i), A1(i), A2(i)	Transfer function coefficients. (Real)

REMARKS:

1. Transfer function sets must be selected with the Case Control command TFL=SID.
2. Continuation entries are optional.
3. The matrix elements defined by this entry are added to the dynamic matrices for the problem.
4. The constraint relation given in Equation 18-1 will hold only if no structural elements or other matrix elements are connected to the dependent coordinate u_d. In fact, the terms on the left side of Equation 18-1 are simply added to the terms from all other sources in the row for u_d.
5. See the *NX Nastran Advanced Dynamic Analysis User's Guide* for a discussion of transfer functions.
6. For each SID, only one logical entry is allowed for each GD, CD combination.
7. For heat transfer analysis, the initial conditions must satisfy Equation 18-1.

TIC

Transient Initial Condition

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

EXAMPLE:

TIC	1	3	2	5.0	-6.0				

FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
G	Grid, scalar, or modal coordinate identification number (Integer>0). See Remark 4.
C	Component numbers. (Any one of the integers 1 through 6 for grid points, integer 0 or blank for scalar, and integer -1 for modal coordinates.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real) See Remark 4.

REMARKS:

1. Transient initial condition sets must be selected with the case control command. Note the use of IC in the case control command versus TIC on the bulk entry. For heat transfer, the IC case control command selects TEMP or TEMPD bulk entries for initial conditions and not the TIC entry.

2. If no TIC set is selected in the case control section, all initial conditions are assumed to be zero.

3. Initial conditions for coordinates not specified on TIC bulk entries will be assumed zero.

4. In direct transient analysis (SOL 109 and 129) wherein the TIC bulk entry is selected by an IC or IC(PHYSICAL) case control command, G may reference only grid or scalar points. In modal transient analysis (SOL 112) wherein the TIC bulk entry is selected by an IC or IC(PHYSICAL) case control command, G may reference only grid or scalar points. In modal transient analysis (SOL 112) wherein the TIC bulk entry is selected by an IC(MODAL) case control command, G may reference only modal coordinates.

5. The initial conditions for the independent degrees-of-freedom specified by this bulk entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and/or TLOAD2 bulk entries.

6. Input at extra points will be ignored.

TLOAD1

Transient Response Dynamic Excitation, Form 1

Defines a time-dependent dynamic load or enforced motion of the form:

$$\{P(t)\} = \{A \cdot F(t - \tau)\}$$

for use in transient response analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAY	TYPE	TID				

EXAMPLES:

TLOAD1	5	7	101	LOAD	13				

TLOAD1	5	7	0.005		13				

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of a load set (for example, the DAREA entry), SPCD entry, SELOAD entry, or thermal load set (for heat transfer analysis) that lists each degree of freedom to apply the excitation and the corresponding scale factor, *A*, for the excitation. See Remark 2 and Remark 3. (Integer > 0)
DELAY	Time delay, *τ*. (Real or Integer ≥ 0 or blank; for default behavior, see Remark 10) If real entry, value of *τ* for all degrees of freedom in EXCITEID entry.

Field	Contents
	If integer entry, identification number of a DELAY entry that contains values of τ for all degrees of freedom in EXCITEID entry. See Remark 11.
TYPE	Defines the type of the dynamic excitation. See Remark 2 and Remark 3. (Integer, character or blank; Default = 0)
TID	Identification number of TABLEDi entry that defines *F(t)* for all degrees of freedom in EXCITEID entry. (Integer > 0)

REMARKS:

1. Dynamic excitation sets must be selected with DLOAD = SID in the case control section.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data

3. TYPE determines the manner in which EXCITEID is used by the program.
   - TYPE = 0 : Applied load excitation.
   - TYPE = 1, 2, or 3 : Enforced motion excitation.
     - o If EXCITEID references SPC/SPCD entries, the software will use the SPCD method of enforced motion. You directly specify displacements, velocities, or accelerations with SPC/SPCD entries.
     - o If EXCITEID references a load entry (DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entries), the software will use the large

mass method of enforced motion. You create a large mass and load at the grid and degree-of-freedom where the enforced motion is desired. The software then computes the enforced motion the same for TYPE = 1, 2, and 3.

Type = 1, Enforced displacement = Dynamic load input / Large mass
Type = 2, Enforced velocity = Dynamic load input / Large mass
Type = 3, Enforced acceleration = Dynamic load input / Large mass

See the Enforced Motion chapter in the *NX Nastran Basic Dynamic Analysis User's Guide* for details on both the SPCD and large mass methods of enforced motion.

4. The legacy method of selecting a DAREA, FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entry for a dynamic loading required the LOADSET case control command which selected the LSEQ bulk entry, which in turn selected the load entry. Since the FORCEi, MOMENTi, PLOAD, PLOAD1, and SPCD entries are now selected directly with the EXCITEID on the dynamic load entries DLOADi, TLOADi, and RLOADi, the LOADSET case control and LSEQ bulk entry are no longer required. The legacy method is still supported, and is described on the LSEQ entry remarks.

5. EXCITEID may reference sets containing QHBDY, QBDYi, QVECT, and QVOL entries when using the heat transfer option.

6. TLOAD1 loads may be combined with TLOAD2 loads using a DLOAD bulk entry.

7. SID must be unique for all TLOAD1, TLOAD2, RLOAD1, RLOAD2, ACSRCE, and SELOAD entries.

8. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore *A* may be a function of time.

9. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. Please refer to the "Fourier Transform" section of the *NX Nastran Advanced Dynamic Analysis User's Guide*.

10. If the DELAY field is blank or zero (either integer zero or real zero), the corresponding value for τ used by the software is real zero.

11. For degrees of freedom in the EXCITEID entry that are not specified on the DELAY entry, the software uses real zero as the value for τ.

12. In SOL 401, when RFORCE or RFORCE1 entries are referenced by the EXCITEID field on a TLOAD1 entry, the data on the associated TABLEDi, along

with the scale factors S and Si on a DLOAD entry (if defined), scale the angular velocity (ω) and acceleration (α), which are used to compute an inertia force in the equation F = [m] [ω x (ω x r)) + α x r]. Since ω is squared in the force computation, the resulting scaling is not linearly related to the computed force (F). All other solutions scale the computed force (F).

13. In SOL 401, thermal loads, defined, for example, with the TEMPD, TEMP, or TEMPP1 bulk entries, cannot be referenced by the EXCITEID field in order to define a time-stepped thermal load. If you attempt to reference a thermal load with the EXCITEID field, the software will ignore the definition without issuing a warning or an error. Thermal loads can be selected in a subcase with the TEMP(LOAD) case control command.

REMARKS RELATED TO SOLS 601 AND 701:

1. TLOAD1 may be used in a static or transient analysis to specify time-dependent loads. In a nonlinear static analysis, TLOAD1 is frequently used with TSTEP to apply incremental loads to achieve better convergence in the solution.

2. SID may be selected directly with the Case Control command DLOAD = SID, or be combined with other TLOAD1 entries in a DLOAD entry. For the latter case, the SID in the DLOAD entry is then selected by the Case Control command DLOAD.

3. Only TYPE = 0 or 1 is supported.

4. For TYPE = 0, EXCITEID may reference the SID in a FORCE, FORCE1, FORCE2, MOMENT, MOMENT1, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, GRAV, RFORCE, TEMP, or TEMPD entry. SELOAD entries are not supported.

5. For TYPE = 1, EXCITED must reference the SID in an SPCD entry.

6. Delay is ignored.

TLOAD2

Transient Response Dynamic Excitation, Form 2

Defines a time-dependent dynamic excitation or enforced motion of the form:

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (\mathrm{T1} + \tau) \text{ or } t > (\mathrm{T2} + \tau) \\ A\tilde{t}^{B} e^{C\tilde{t}} \cos\left(2\pi F\tilde{t} + \frac{P\pi}{180}\right), & (\mathrm{T1} + \tau) \le t \le (\mathrm{T2} + \tau) \end{cases}$$

for use in a transient response problem, where:

$$\tilde{t} = t - \mathrm{T1} - \tau$$

FORMAT:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAY	TYPE	T1	T2	F	P	
	C	B							

EXAMPLES:

TLOAD2	4	10	106		2.1	4.7	12.0		
	2.0								

TLOAD2	4	10	0.01		2.1	4.7	12.0		
	2.0	0.4							

FIELDS:

Field	Contents
SID	Load set identification number. (Integer > 0)

Field	Contents
EXCITEID	Identification number of a load set (for example, the DAREA entry), SPCD entry, SELOAD entry, or a thermal load set (for heat transfer analysis) that lists each degree of freedom to apply the excitation and the corresponding scale factor, *A*, for the excitation. See Remark 2 and Remark 3. (Integer > 0)
DELAY	Time delay, τ. (Real or Integer ≥ 0 blank; for default behavior, see Remark 6) If real entry, value of τ for all degrees of freedom in EXCITEID entry. If integer entry, identification number of a DELAY entry that contains values of τ for all degrees of freedom in EXCITEID entry. See Remark 7.
TYPE	Defines the type of the dynamic excitation. See Remark 2 and Remark 3. (Integer, character or blank; Default = 0)
T1	Time constant. (Real ≥ 0.0)
T2	Time constant. (Real; T2 > T1)
F	Frequency in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
P	Phase angle in degrees. (Real; Default = 0.0)
C	Exponential coefficient. (Real; Default = 0.0)
B	Growth coefficient. (Real; Default = 0.0)

REMARKS:

1. Dynamic excitation sets must be selected with DLOAD = SID in the case control section.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)

TYPE	TYPE of Dynamic Excitation
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data

3. TYPE determines the manner in which EXCITEID is used by the program.

   - TYPE = 0 : Applied load excitation.

   - TYPE = 1, 2, or 3 : Enforced motion excitation.

     - o If EXCITEID references SPC/SPCD entries, the software will use the SPCD method of enforced motion. You directly specify displacements, velocities, or accelerations with SPC/SPCD entries.

     - o If EXCITEID references a load entry (DAREA, FORCEi, MOMENTi, PLOAD, RFORCEi, and PLOAD1 entries), the software will use the large mass method of enforced motion. You create a large mass and load at the grid and degree-of-freedom where the enforced motion is desired. The software then computes the enforced motion the same for TYPE = 1, 2, and 3.

       Type = 1, Enforced displacement = Dynamic load input / Large mass
       Type = 2, Enforced velocity = Dynamic load input / Large mass
       Type = 3, Enforced acceleration = Dynamic load input / Large mass

   See the “Enforced Motion” chapter in the *NX Nastran Basic Dynamic Analysis User’s Guide* for details on both the SPCD and large mass methods of enforced motion.

4. The legacy method of selecting a FORCEi, MOMENTi, RFORCEi, PLOAD, and PLOAD1 entry for a dynamic loading required the LOADSET case control command which selected the LSEQ bulk entry, which in turn selected the load entry. Since the FORCEi, MOMENTi, PLOAD, PLOAD1, and SPCD entries are now selected directly with the EXCITEID on the dynamic load entries DLOADi, TLOADi, and RLOADi, the LOADSET case control and LSEQ bulk entry are no longer required. The legacy method is still supported, and is described on the LSEQ entry remarks.

5. EXCITEID (field 3) may reference sets containing QHBDY, QBDYi, QVECT, and QVOL entries when using the heat transfer option.

6. If the DELAY field is blank or zero (either integer zero or real zero), the corresponding value for τ used by the software is real zero.

7. For degrees of freedom in the EXCITEID entry that are not specified on the DELAY entry, the software uses real zero as the value for τ.

8. TLOAD1 loads may be combined with TLOAD2 loads using a DLOAD bulk entry.

9. SID must be unique for all TLOAD1, TLOAD2, RLOAD1, RLOAD2, ACSRCE, and SELOAD entries.

10. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore *A* may be a function of time.

11. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and them combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. In this case, *B* will be rounded to the nearest integer. Please refer to “Fourier Transform” section of the *NX Nastran Advanced Dynamic Analysis User’s Guide*.

12. The continuation entry is optional.

TMCPARA

Parameters for SOL 601 Thermo-Mechanical Coupling Analysis

Defines parameters for SOL 601,153 and SOL 601,159 thermo-mechanical coupling (TMC) analysis.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TMCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	-etc.-				

EXAMPLE:

TMCPARA	1	COUP	1	MAXITE	30	TRELAX	0.9		
	AUTO	1							

FIELDS:

Field	Contents
ID	Identification number. Currently not used. (Integer ≥ 0)
PARAMi	Name of the TMCPARA parameter. Allowable names are given in the parameter listing below. (Character)
VALUEi	Value of the parameter. (Real or Integer)

Table 18-2. TMCPARA parameters

Name	Description
	General Parameters
COUP	Specifies the type of TMC coupling for SOL 601,153 or SOL 601,159. (Integer, Default = 0) 0 – Iterative coupling 1 – One-way sequential coupling 2 – No coupling Two subcases are required for TMC analysis, one with ANALYSIS=HEAT and one with ANALYSIS=STRUC (default). If the first two subcases do not satisfy the above, the program will issue error and exit. COUP=0 will run structural analysis first followed by thermal analysis with TMC iterations. This type of coupling is applicable when the structural deformations affect the thermal analysis, e.g., contact, heat generation due to plastic deformation or frictional contact sliding. COUP=1 will run thermal analysis first followed by structural analysis with no TMC iterations. This type of coupling is applicable when the temperature results affect the structural analysis (e.g., thermal expansion and temperature-dependent material properties), but the structural deformations do not have significant effect on the thermal analysis. COUP=2 will run the first subcase (either thermal or structural).
TRANOPT	Specifies transient or static analysis options for SOL 601,159. (Integer, Default = 0) 0 – Transient thermal and structural 1 – Transient thermal, static structural 2 – Steady-state thermal, transient structural
	Automatic Time Stepping

Table 18-2. TMCPARA parameters

Name	Description
AUTO	Indicates whether automatic time stepping scheme is used for the thermal analysis solution (Integer, Default = 0) 0 – No automatic incrementation scheme is used 1 – Automatic Time Stepping (ATS) scheme is used
ATSSUBD	Number that limits the smallest time step size when the automatic time stepping (ATS) scheme is used. For a time step size of DT, the program will stop if convergence is not achieved and the next subdivided time step size is less than DT/ATSSUBD. (Integer ≥ 1; Default = 10)
	Transient Analysis
TINTEG	Selects the time integration method to be used for transient thermal analysis. (Integer; Default = 0) 0 – Euler backward method 2 – Trapezoidal rule 3 – Composite method
HEATCAP	Selects the type of heat capacity matrix to be used in transient thermal analysis. (Integer; Default = 1) 0 – Lumped heat capacity matrix 1 – Consistent heat capacity matrix
	Equilibrium Iteration and Convergence
ITSCHEM	Selects the equilibrium iteration scheme used for nonlinear thermal analysis. (Integer; Default = 1) 0 – Modified Newton iteration 1 – Full Newton iteration
LSEARCH	Flag to indicate the use of line searches within the iteration scheme for nonlinear thermal analysis. (Integer; Default = 0) 0 – Line search is not used 1 – Line search is used

Table 18-2. TMCPARA parameters

Name	Description
MAXITE	Maximum number of iterations within a thermal solution time step. If the maximum number of iterations is reached without achieving convergence, the program will stop unless the automatic time stepping (ATS) scheme is selected (AUTO=1). (1 ≤ Integer ≤ 999; Default = 15)
TTOL	Relative temperature tolerance used to determine if the thermal solution has converged. (Real; Default = 0.001)
TMCTOL	Relative tolerance used to determine if the TMC iteration has converged. Applicable only for iterative TMC coupling solution, i.e., COUP=0. (Real; Default = TTOL)
TRELAX	Temperature relaxation factor that may be used to help in the convergence of thermal solution. (0.0 ≤ Real ≤ 1.0; Default = 1.0)
Miscellaneous	
HGENPL	Proportion of internal heat generated in shell, 2D and 3D solid elements due to plastic deformation. (0.0 ≤ Real ≤ 1.0; Default = 1.0)

REMARKS:

1. The TMCPARA bulk entry is not required. When it is not present, the default values are used. At least one parameter should be defined when a TMCPARA entry exists.

TRIM

Trim Variable Constraint

Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TRIM	ID	MACH	Q	LABEL1	UX1	LABEL2	UX2	AEQR	
	LABEL3	UX3	-etc.-						

EXAMPLE:

TRIM	1	0.9	100.	URDD3	1.0	ANGLEA	7.0	0.0	
	ELEV	0.2							

FIELDS:

Field	Contents
SID	Trim set identification number. (Integer > 0)
MACH	Mach number. (Real ≥ 0.0 and ≠ 1.0)
Q	Dynamic pressure. (Real > 0.0)
LABELi	The label identifying aerodynamic trim variables defined on an AESTAT or AESURF entry. (Character. See Bulk Data Syntax Rules.)
UXi	The magnitude of the aerodynamic extra point degree-of-freedom. (Real)
AEQR	Flag to request a rigid trim analysis (Real ≥ 0.0 and ≤ 1.0, Default1.0. A value of 0.0 provides a rigid trim analysis, see Remark 4.

REMARKS:

1. The TRIM entry must be selected with the Case Control command TRIM=SID.

2. The selected TRIM entry specifies the constrained values of the listed extra point degrees-of-freedom (“trim variables”) for a particular loading condition. These variables are listed on AESTAT and/or AESURF entries.

3. If MACH is less than 1.0, the Doublet-Lattice theory is used. If MACH is greater than 1.0, the ZONA51 theory can be used if licensed to do so.

4. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

TSTEP

Transient Time Step

Defines time step intervals at which a solution will be generated and output in transient analysis.

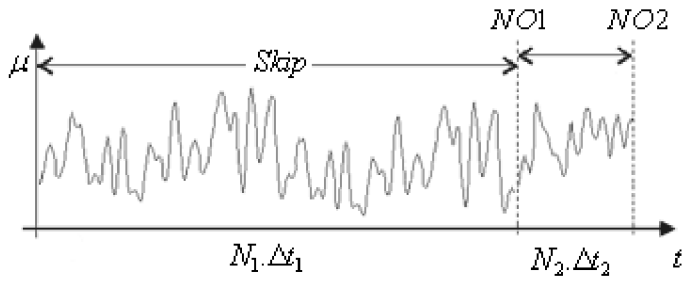
FORMAT:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1					
		N2	DT2	NO2					
		-etc.-							

EXAMPLE:

TSTEP	101	9000	.001	9000					
		1000	.001	1					

In the above example, 10000 time steps with increment .001 seconds are computed, the first 9000 time steps are skipped, and time steps 9001 thru 10000 are output.



FIELDS:

Field	Contents
SID	Set identification number. (Integer > 0)
Ni	Number of time steps of value DTi. (Integer ≥ 1)
DTi	Time increment. (Real > 0.0)
NOi	Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0; Default=1)

REMARKS:

1. TSTEP entries must be selected with the Case Control command TSTEP=SID.
2. Note that TSTEP permits changes in the size of the time step during the course of the solution. In the above example, 10000 time steps with increment .001 seconds are computed, the first 9000 time steps are skipped, and time steps 9001 thru 10000 are output.
3. See “Guidelines for Effective Dynamic Analysis ” in the *NX Nastran Basic Dynamic Analysis User’s Guide* for a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.
5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the TSTEPNL entry.

REMARKS RELATED TO SOLS 601 AND 701:

1. TSTEP may be used in a static or transient analysis. In a nonlinear static analysis, TSTEP is frequently used with TLOAD1 to apply incremental loads to achieve better convergence in the solution.

2. If no TSTEP is selected with the Case Control command TSTEP, the program assumes 1 time step with time increment of 1.0.

3. For SOL 701, the actual time step size used may be determined by the program based on the critical time step size for stability. To request that the program use the time step size DTi here, specify XSTEP=1 in NXSTRAT entry. In all cases, the total solution time is used, i.e., total solution time is the sum of all Ni * DTi

4. For SOL 701, please see Section 7.1 in the Advanced Nonlinear Theory and Modeling Guide on the critical time step size and when results are output.

5. Note that TSTEPNL is not used in SOLs 601 or 701.

TSTEP1

Time step intervals for SOL 401.

Defines the time step intervals at which a solution will be generated and output in SOL 401.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TSTEP1	SID	Tend1	Ninc1	Nout1					
		Tend2	Ninc2	Nout2					
		Tend3	Ninc3	Nout3					
		-etc.-							

EXAMPLE:

TSTEP1	1	10.0	5	2					
		50.0	4	3					
		100	2	ALL					

In this example, assuming a start time*=0.0 for the subcase, the resulting time steps are as follows. The time steps in which output occurs are highlighted.

The 1st row has an end time of 10.0, 5 increments, and output at every 2nd time step.

Time Step 1	Time Step 2	Time Step 3	Time Step 4	Time Step 5
2.0	4.0	6.0	8.0	10.0

The 2nd row has an end time of 50.0, 4 increments, and output frequency at every 3rd time step.

Time Step 1	Time Step 2	Time Step 3	Time Step 4
20.0	30.0	40.0	50.0

The 3rd row has an end time of 100.0, 2 increments, and output at all time steps.

Time Step 1	Time Step 2
75.0	100.0

In the same example, assuming a start time*=5.0 for the subcase, the resulting time steps for the first row are as follows.

Time Step 1	Time Step 2	Time Step 3	Time Step 4	Time Step 5
6.0	7.0	8.0	9.0	10.0

The 2nd and 3rd row are the same:

Time Step 1	Time Step 2	Time Step 3	Time Step 4
20.0	30.0	40.0	50.0

Time Step 1	Time Step 2
75.0	100.0

*See Remark 4.

FIELDS:

Field	Contents
SID	Identification number. (Integer > 0)
Tendi	End time (REAL≥ 0.0; $Tend_i < Tend_{i+1}$; Default = 1.0 for Tend1). See Remark 5.
Ninci	Number of increments (Integer > 0; Default = 1).
Nouti	Controls the frequency of output. See Example. (Character = “YES”, ”END”, ”ALL”, “CPLD”, or Integer ≥ 0; Default = “END”). See Remarks 3 and 6.

REMARKS:

1. The TSTEP1 entry must be selected with the TSTEP=SID case control command.

2. No output occurs when Nouti=0.

3. Output occurs at Tendi when Nouti≠0.

4. The start time (T_{start}) for a static subcase is determined as follows:

   - If a static subcase definition in the case control includes SEQDEP=NO, that subcase is not sequentially dependent (NSD). The start time for an NSD subcase is 0.0.

   - For a sequentially dependent (SD) static subcase (default), the final Tendi from a previous SD or NSD static subcase is the start time (T_{start}) for the current SD subcase. If an SD subcase has no previous SD or NSD static subcases, the start time is 0.0 for that SD subcase, and Tend1=0.0 is permitted. Otherwise, Tend1 ≥ T_{start} for all other SD subcases.

5. A constant time subcase is defined by either setting Tend=0.0, or by setting Tend=T_{start}. See the previous remark. The following conditions apply to a constant time subcase:

   - Only a single Tendi is permitted.

   - Ninc can be defined to divide loads, temperatures, and contact offsets into increments. This is useful to help convergence.

   - Output occurs one time. It occurs at the final increment if Ninc>0. Nouti≠0 for output to occur.

6. Nouti controls the frequency of results output. The table below summarizes the input options.

Nout	Output frequency
YES	Output occurs at all increments defined on TSTEP1.
END (Default)	Output occurs at the end time.
ALL	Output occurs at all increments on TSTEP1 and any software subincrements. Note: When Nouti=ALL in the context of the Simcenter Multiphysics solution, the result output time steps will be a combination of the structural output steps as well as the coupled time steps.
Integer = 0	No output occurs.
Integer > 0	Output is computed at every Nout increment specified with TSTEP1. See Example.
CPLD	Output occurs only at coupling times. This option can only be defined by the Simcenter Multiphysics environment.

TSTEPNL

Parameters for Nonlinear Transient Analysis

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129 and 159.

FORMAT:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO		KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	KDAMP	
	KUPDATE								

EXAMPLE:

TSTEPNL	250	100	.01	1		2	10	PW	
	1.E-2	1.E-3	1.E-6	2	10	2	.02		
	5	5	0	0.75	16.0	0.1	20.		

FIELDS:

Field	Contents
ID	Identification number. (Integer > 0)
NDT	Number of time steps of value DT. See Remark 2 . (Integer ≥ 4)
DT	Time increment. See Remark 2 . (Real > 0.0)
NO	Time step interval for output. Every NO-th step will be saved for output. See Remark 3 . (Integer > 0; Default=1)
KSTEP	KSTEP is the number of converged bisection solutions between stiffness updates. (Integer > 0; Default=2)

Field	Contents
MAXITER	Limit on number of iterations for each time step. See Remark 4 . (Integer ≠ 0; Default=10)
CONV	Flags to select convergence criteria. See Remark 5 . (Character: “U”, “P”, “W”, or any combination; Default=“PW.”)
EPSU	Error tolerance for displacement (U) criterion. (Real > 0.0; Default=1.0E-2)
EPSP	Error tolerance for load (P) criterion. (Real > 0.0; Default=1.0E-3)
EPSW	Error tolerance for work (W) criterion. (Real > 0.0; Default=1.0E-6)
MAXDIV	Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark 6 . (Integer > 0; Default=2)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 7 . (Integer ≥ 0; Default=10)
MAXLS	Maximum number of line searches allowed per iteration. See Remark 7 . (Integer ≥ 0; Default=2)
FSTRESS	Fraction of effective stress $(\bar{\sigma})$ used to limit the subincrement size in the material routines. See Remark 8 . (0.0 < Real < 1.0; Default=0.2)
MAXBIS	Maximum number of bisections allowed for each time step. See Remark 9 . (-9 ≤ Integer ≤ 9; Default=5)
ADJUST	Time step skip factor for automatic time step adjustment. See Remark 10 . (Integer ≥ 0; Default=5)
MSTEP	Number of steps to obtain the dominant period response. See Remark 11 . (10 ≤ Integer ≤ 200; Default=variable between 20 and 40.)
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 11 . (0.1 ≤ Real ≤ 1.0; Default=0.75)
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 12 . (1.0 ≤ Real ≤ 32.0; Default=16.0)

Field	Contents
UTOL	Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 13 . (0.001 < Real ≤ 1.0; Default=0.1)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 14. (Real > 2.0; Default=20.0)
KDAMP	Option to include differential stiffness in calculating structural damping. (Character= “YES”, “NO”; Default = “NO”)
KUPDATE	Method for controlling stiffness updates. See Remark 15. (Character = “AUTO”, “FULL”; Default = “AUTO”)

REMARKS:

1. The TSTEPNL Bulk Data entry is selected by the Case Control command TSTEP=ID. Each residual structure subcase requires a TSTEP entry and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase defined by the end conditions of the previous subcase.

2. NDT is used to define the total duration for analysis, which is NDT * DT. Since DT is adjusted during the analysis , the actual number of time steps, in general, will not be equal to NDT). Also, DT is used only as an initial value for the time increment.

3. For printing and plotting the solution, data recovery is performed at time steps 0, NO, 2 * NO, ..., and the last converged step. The Case Control command OTIME may also be used to control the output times.

4. The number of iterations for a time step is limited to MAXITER. If MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process. See Remark 6 .

5. The convergence test flags (U=displacement error test, P=load equilibrium error test, W=work error test) and the error tolerances (EPSU, EPSP, and EPSW)

define the convergence criteria. All requested criteria (combination of U, P, and/or W) are satisfied upon convergence. Note that at least two iterations are necessary to check the displacement convergence criterion.

When KUPDATE=AUTO, EPSP is used to detemine if the stiffness is updated, even when P is not included in the CONV field. This can have an effect on convergence or results. See “Update Principles” in Section 3.3.1 of the *NX Nastran Handbook of Nonlinear Analysis*.

6. MAXDIV provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions (NDIV) is incremented by 1 or 2. The solution is assumed to diverge when NDIV reaches MAXDIV during the iteration. If the bisection option is used (allowed MAXBIS times) the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new stiffness based on the converged state at the beginning of the time step. If NDIV reaches MAXDIV again within the same time step, the analysis is terminated.

7. Nonzero values of MAXQN and MAXLS will activate the quasi-Newton update and the line search process, respectively.

   EPSW is used to detemine if line search is performed, even when W is not included in the CONV field. This can have an effect on convergence or results. See “Other Provisions for Line Search” in section 3.4.5 of the *NX Nastran Handbook of Nonlinear Analysis*.

8. The number of subincrements in the material routines is determined such that the subincrement size is approximately FSTRESS. FSTRESS is also used to establish a tolerance for error correction in elastoplastic material, i.e.,

$$\text{error in yield function} < \text{FSTRESS} \cdot \text{yield stress}$$

   If the limit is exceeded at the converging state, the program will terminate with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface, resulting in $\delta=0$.

9. The bisection process is activated when divergence occurs and MAXBIS $\neq 0$. The number of bisections for a time increment is limited to |MAXBIS|. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in |MAXBIS| bisection, the analysis is terminated.

10. ADJUST controls the automatic time stepping. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is the user’s responsibility to ensure that the loading history is properly traced with the ADJUST option. The ADJUST option should be suppressed for

the duration of short pulse loading. If unsure, start with a value for DT that is much smaller than the pulse duration in order to properly represent the loading pattern.

- If ADJUST=0, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
- If ADJUST > 0, the time increment is continually adjusted for the first few steps until a good value of Δt is obtained. After this initial adjustment, the time increment is adjusted every ADJUST-th time step only.
- If ADJUST is one order greater than NDT, then automatic adjustment is deactivated after the initial adjustment.

11. MSTEP and RB are used to adjust the time increment during analysis. The recommended value of MSTEP for nearly linear problems is 20. A larger value (e.g., 40) is required for highly nonlinear problems. By default, the program automatically computes the value of MSTEP based on the changes in the stiffness.

    The time increment adjustment is based on the number of time steps desired to capture the dominant frequency response accurately. The time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r)\Delta t_n$$

where

$$r = \frac{1}{\mathrm{MSTEP}}\left(\frac{2\pi}{\omega_n}\right)\left(\frac{1}{\Delta t_n}\right)$$

with

f	=	0.25	for	$r < 0.5 \cdot \mathrm{RB}$
f	=	0.5	for	$0.5 \cdot \mathrm{RB} \le r < \mathrm{RB}$
f	=	1.0	for	$\mathrm{RB} \le r < 2.0$
f	=	2.0	for	$2.0 \le r < 3.0/\mathrm{RB}$
f	=	4.0	for	$r \ge 3.0/\mathrm{RB}$

12. MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\mathrm{MIN}\left(\frac{\mathrm{DT}}{2^{\mathrm{MAXBIS}}}, \frac{\mathrm{DT}}{\mathrm{MAXR}}\right) \leq \Delta t \leq \mathrm{MAXR} \cdot \mathrm{DT}$$

13. UTOL is a tolerance used to filter undesirable time step adjustments; i.e.,

$$\frac{\left\|\dot{U}_n\right\|}{\left\|\dot{U}\right\|_{max}} < \mathrm{UTOL}$$

Under this condition no time step adjustment is performed in a structural analysis (SOL 129). In a heat transfer analysis (SOL 159) the time step is doubled.

14. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$, $\Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.

15. The following stiffness update methods can be selected in the KUPDATE field.

    - If AUTO is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step, the number of iterations required to converge is estimated. Stiffness is updated, if
      (i) the estimated number of iterations to converge exceeds MAXITER,
      (ii) the estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and
      (iii) the solution diverges.

      See Remark 5.

    - If the FULL option is selected, the program updates the stiffness matrix at every Newton-Raphson iteration. For large deflection problems (PARAM,LGDISP > 0), the FULL option is recommended.

USET

Degree-of-Freedom Set Definition

Defines a degree-of-freedom set.

FORMAT:

1	2	3	4	5	6	7	8	9	10
USET	SNAME	ID1	C1	ID2	C2	ID3	C3		

EXAMPLE:

USET	U4	333	26	17	0				

FIELDS:

Field	Contents
SNAME	Set name. (One to four characters, or the string “ZERO” followed by the set name.)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

REMARKS:

1. SNAME may refer to any of the set names given in “Degree-of-Freedom Sets” or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U9 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.

2. If SNAME=“ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

3. A maximum of 18 degrees-of-freedom may be designated on a single entry.

4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.

5. The USET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.

6. If a USETi Bulk Data entry lists a standard NX Nastran set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

7. The user defined degree-of-freedom sets U1 – U9 are created with the USET and USET1 bulk entries. Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases. See “User Defined Degree-of-Freedom Sets” for the list of these special cases.

USET1

Degree-of-Freedom Set Definition, Alternate Form

Defines a degrees-of-freedom set.

FORMAT:

1	2	3	4	5	6	7	8	9	10
USET1	SNAME	C	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc.-						

EXAMPLE:

USET1	SB	345	2	1	36	5	9	7	
	40								

ALTERNATE FORMAT AND EXAMPLE:

USET1	SNAME	C	ID1	“THRU”	ID2				
USET1	SB	123	170	THRU	180				

FIELDS:

Field	Contents
SNAME	Set name. (One to four characters or the word “ZERO” followed by the set name.)
C	Component numbers. (Integer zero or blank for scalar points or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, ID1 < ID2.)

REMARKS:

1. SNAME may refer to any of the set names given in "Degree-of-Freedom Sets" or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U9 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.

2. If SNAME= "ZEROi", where i is a set name, then the degrees-of-freedom are omitted from set i.

3. If the alternate format is used, all of the points ID1 through ID2 are assigned to the set.

4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.

5. The USET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.

6. If a USETi Bulk Data entry lists a standard NX Nastran set, such as S or M, the program may fail in the PARTN module with the message "SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE". This entry should only reference new sets defined on DEFUSET Bulk Data entries.

7. The user defined degree-of-freedom sets U1 – U9 are created with the USET and USET1 bulk entries. Be aware that the user defined sets U2 - U8 are used by NX Nastran in some special cases. You may use U2 – U8 as long as it doesn't conflict with these cases. See "User Defined Degree-of-Freedom Sets" for the list of these special cases.

UXVEC

Control Parameter State

Specification of a vector of aerodynamic control point (extra point) values. These data define the control positions corresponding to user defined nonlinear control forces that have been defined by AEDW, AEPRESS and AEFORCE entries. Only nonzero values need to be defined.

FORMAT:

1	2	3	4	5	6	7	8	9	10
UXVEC	ID								
	LABEL1	UX1	LABEL2	UX2	-etc.-				

EXAMPLE:

UXVEC	1001								
	THRUST	1.E4	ANGLEA	.015					

FIELDS:

Field	Contents
ID	Control vector identification number, see Remark 1. (Integer > 0)
LABELi	Controller name. This must correspond to an existing AESURF, AESTAT or AEPARM label. (Character. See Bulk Data Syntax Rules.)
UXi	The magnitude of the aerodynamic extra point degree of freedom (Real)

For ETOL, when the error estimate exceeds the value input for the ETOL entry

1. The contour method is employed to develop an improved view
2. The units of the user defined AEPARM controllers are implied by their use on this entry and the corresponding values on the force vector definition. The user

must be self-consistent in all uses of the same controller. AESURF controllers are expressed in radians as are the rigid body angles ANGLEA and BETA. The rigid body rates, ROLL, PITCH and YAW are nondimensional rates pb/2V, qc/2V, rb/2V; respectively. V is the velocity and b and c are the reference span and chord lengths, respectively.

3. LABELs that are part of the UX vector that are omitted in the UXVEC specification are assigned a value of 0.0.

VCEV

Vitual Crack Extension Vector

Defines virtual crack tip extension vectors.

FORMAT:

1	2	3	4	5	6	7	8	9	10
VCEV	SID	CID							
	N11	N12	N13						
	N21	N22	N23						
	N31	N32	N33						
	N41	N42	N43						
	N51	N52	N53						
	N61	N62	N63						
	N71	N72	N73						

EXAMPLE:

CRAKTP	1	5							
	1.0	0.0	0.0						
	0.707	0.707	0.0						
	0.5	0.866	0.0						
	0.0	1.0	0.0						
	–0.5	0.866	0.0						
	–0.707	0.707	0.0						
	–1.0	0.0	0.0						

VCEV

FIELDS:

Field	Contents
SID	Virtual crack extension vector identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
Ni1, Ni2, Ni3	Components of the i-th virtual crack extension vector in the CID coordinate system. (Real)

REMARKS:

1. Up to seven virtual crack extension vectors can be defined.
2. Vectors do not need to be normalized when they are entered on a VCEV bulk entry. The software will normalize them when computing the j-integral.

VIEW

View Factor Definition

Defines radiation cavity and shadowing for radiation view factor calculations.

FORMAT:

1	2	3	4	5	6	7	8	9	10
VIEW	IVIEW	ICAVITY	SHADE	NB	NG	DISLIN			

EXAMPLE:

VIEW	1	1	BOTH	2	3	0.25			

FIELDS:

Field	Contents
IVIEW	Identification number. (Integer > 0)
ICAVITY	Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)
SHADE	Shadowing flag for the face of CHBDYi element. (Character, Default=“BOTH”)
	NONE means the face can neither shade nor be shaded by other faces.
	KSHD means the face can shade other faces.
	KBSHD means the face can be shaded by other faces.
	BOTH means the face can both shade and be shaded by other faces. (Default)
NB	Subelement mesh size in the beta direction. (Integer > 0; Default=1)
NG	Subelement mesh size in the gamma direction. (Integer > 0; Default=1)

Field	Contents
DISLIN	The displacement of a surface perpendicular to the surface. See Figure 18-10. (Real; Default=0.0)

REMARKS:

1. VIEW must be referenced by CHBDYE, CHBDYG, or CHBDYP elements to be used.
2. ICAVITY references the cavity to which the face of the CHBDYi element belongs; a zero or blank value indicates this face does not participate in a cavity.
3. NB, NG, and DISLIN are used in the calculation of view factors by finite difference or contour integration techniques. They are not used with the VIEW3D entry.
4. A summary of the shadowing conditions can be requested by the PARAM,MESH,YES Bulk Data entry.
5. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.
6. DISLIN should only be used with LINE type CHBDYE and CHBDYP surface elements. DISLIN > 0.0 means into the cavity. See Figure 18-10.

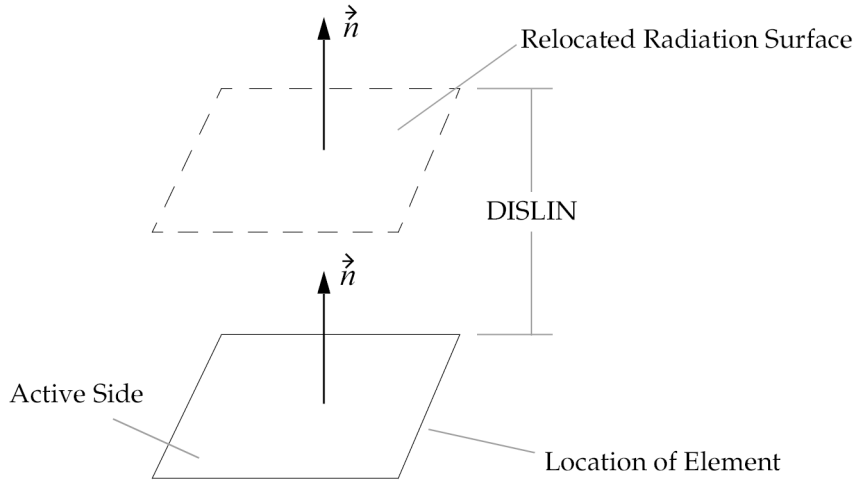


Figure 18-10. DISLIN Convention

7. NB and NG define the subelement mesh refinement when using the VIEW module (as opposed to the VIEW3D module) for the calculation of view factors.

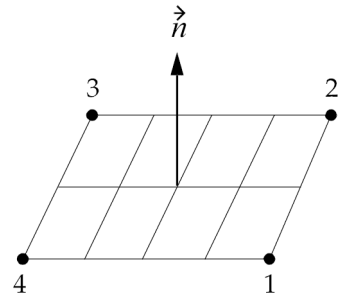


Figure 18-11. Typical AREA4 surface element where NB=2 and NG=4

VIEW3D

View Factor Definition - Gaussian Integration Method

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

FORMAT:

1	2	3	4	5	6	7	8	9	10
VIEW3D	ICAVITY	GITB	GIPS	CIER	ETOL	ZTOL	WTOL	RADCHK	

EXAMPLE:

VIEW3D	1	2	2	4		1.0E-6			

FIELDS:

Field	Contents
ICAVITY	Radiant cavity identification number on RADCAV entry. (Integer > 0)
GITB	Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default=4)
GIPS	Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default=4)
CIER	Discretization level used in the semi-analytic contour integration method. (1 ≤ Integer ≤ 20; Default=4)
ETOL	Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real ≥ 0.0; Default=0.1)
ZTOL	Assumed level of calculation below which the numbers are considered to be zero. (Real ≥ 0.0; Default=1.E-10)

Field	Contents
WTOL	Assumed degree of warpage above which the actual value of F_{ij} will be calculated. (0.0 ≤ Real ≤ 1.0; Default=0.01)
RADCHK	Type of diagnostic output desired for the radiation exchange surfaces. (Integer; Default=3) RADCHK=-1, No diagnostic output requested RADCHK=1, Grid table and element connectivity RADCHK=2, Surface Diagnostics - Surface type, area, skewness, taper, warpage, grid point sequencing, aspect ratio, and shading flags. RADCHK=3, Area, view factor, area-view factor product with error estimate, existence flags for partial self-shadowing, third-body shadowing with error estimate, and enclosure summations for view factor. (Default) RADCHK=0, Same as RADCHK=1, 2, and 3 RADCHK=12, Same as RADCHK=1 and 2 RADCHK=13, Same as RADCHK=1 and 3 RADCHK=23, Same as RADCHK=2 and 3

REMARKS:

1. For ETOL, when the error estimate exceeds the value input for the ETOL entry, the contour method is employed to develop an improved view factor.

2. For ZTOL, the use of a geometry scale that results in small numerical values of *AiFij* should be avoided.

3. When WTOL is exceeded, the actual value of *Fii* will be calculated when using the adaptive view module. Warpage will not be considered in the calculation of factor *Fij*.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

© 2016 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.