SIEMENS

Element Library Reference

Contents

Proprietary & Restricted Rights Notice

Chapter 1: Overview of the Element Library

- *Overview of the NX Nastran Elements*
- *Summary of Small Strain Elements*

1.1 Overview of NX Nastran Elements

The *NX Nastran Element Library* describes the elements supported by NX Nastran. The elements are generally divided into major categories according to their topology:

- scalar (0-D)
- one-dimensional (1-D)
- two-dimensional (2-D)
- three-dimensional (3-D)
- special elements
- R-type elements

Note

See the following guides for information on elements supported by the nonlinear solutions:

For SOLs 106 and 129, see the *Basic Nonlinear Analysis User's Guide* and the *Handbook of Nonlinear Analysis*.

For SOL 401, see the *Multi-Step Nonlinear User's Guide*.

For SOLs 601 and 701, see the *Advanced Nonlinear Theory and Modeling Guide* .

Several general notes apply to all NX Nastran elements:

- All elements in your model should have unique element ID numbers. Do not reuse element IDs on different element types.
- The formulation of an element's stiffness matrix is independent of how you number the element's grid points.
- Each element has its own element coordinate system defined by connectivity order or by other element data. Element information (such as element force or stress) is output in the element coordinate system.

- The performance of elements in NX Nastran's library is constantly being improved. Consequently, you may observe changes in numerical results (for equivalent models) in subsequent versions of the program.

Additional details concerning the features and use of each of NX Nastran's elements can be found in the *NX Nastran Quick Reference Guide.*

Element and Property Definition in NX Nastran

Structural elements are defined on Bulk Data connection entries that identify the grid points to which the element is connected. The mnemonics for all such entries have a prefix of the letter "C", followed by an indication of the type of element, such as CBAR and CROD. The order of the grid point identification defines the positive direction of the axis of a one-dimensional element and the positive surface of a plate element. The connection entries include additional orientation information when required. Some elements allow for offsets between its connecting grid points and the reference plane of the element. The coordinate systems associated with element offsets are defined in terms of the grid point coordinate systems. For most elements, each connection entry references a property definition entry. If many elements have the same properties, this system of referencing eliminates a large number of duplicate entries.

The property definition Bulk Data entries define geometric properties such as thicknesses, cross-sectional areas, and moments of inertia. The mnemonics for all such entries have a prefix of the letter "P", followed by some or all of the characters used on the associated connection entry, such as PBAR and PROD. Other included items are the nonstructural mass and the location of points where stresses will be calculated. For most elements, each property definition entry will reference a material property entry.

In some cases, the same finite element can be defined by using different Bulk Data entries. These alternate entries have been provided for user convenience. In the case of a rod element, the normal definition is accomplished with a connection entry (CROD) which references a property entry (PROD). However, an alternate definition uses a CONROD entry which combines connection and property information on a single entry. This is more convenient if a large number of rod elements all have different properties.

Most of the elements may be used with elements of other types within the limitations of good modeling practice. Exceptions are the axisymmetric elements, which are designed to be used by themselves. There are two types at present, linear and nonlinear. The conical shell element ("CCONEAX") describes a thin shell by sweeping a line defined on a plane by two end points through a circular arc. Loads may be varied with azimuth angle through use of harmonic analysis techniques. This element has a unique set of input entries, which may be used with a limited set of other entries. These unique entries, if mixed with entries for other types of elements, cause a preface error.

Material Properties

The material property definition entries, such as MAT1 and MAT2 are used to define the properties for each of the materials used in the structural model.

In linear analysis, temperature-dependent material properties are computed once only at the beginning of the analysis. In nonlinear analysis, temperature-dependent material properties may be updated many times during the analysis.

See Also

Material Properties in the *NX Nastran User's Guide*.

For SOLs 106 and 129, see the *Basic Nonlinear Analysis User's Guide* and the *Handbook of Nonlinear Analysis*.

For SOL 401, see the *Multi-Step Nonlinear User's Guide*.

For SOLs 601 and 701, see the *Advanced Nonlinear Theory and Modeling Guide* .

1.2 Summary of Small Strain Elements

A summary of the finite elements and their characteristics is given in Table 1-1, Table 1-2, and Table 1-3. An X in the table indicates the existence of an item. Element identification numbers must be unique across all element types.

Element characteristics for the nonlinear solutions are documented in the specific Guides for these solutions:

For SOLs 106 and 129, see the *Basic Nonlinear Analysis User's Guide* and the *Handbook of Nonlinear Analysis*.

For SOL 401, see the *Multi-Step Nonlinear User's Guide*.

For SOLs 601 and 701, see the *Advanced Nonlinear Theory and Modeling Guide*.

Table 1-1. Element Summary – Small Strain Elements, Structural Matrices

	Structural Matrices					
Element Type	**Stiffness**	**Mass**	**Differential Stiffness**	**Viscous Damping**	**Axisymmetric**	**p-Adaptivity**
CAXIFi					X	
CBAR	X	LC	X			
CBEAM	X	LC	X			X
CBUSH	FD			FD		
CBUSH1D	X	X	X			
CBEND	X	C	X			
CCONEAX	X	L			X	
CONMi		LC				
CONROD	CS	LC	X			
CRAC2D	I	LC				
CRAC3D	I	LC				
CDAMPi				X		
CELASi	X					
CFLUIDi					X	
CGAP	X					
CHBDYi						
CHEXA	I	LC	X			X
CMASSi		L				
CPENTA	I	LC	X			X
CQUAD4	I	LC	X			X

Table 1-1. Element Summary – Small Strain Elements, Structural Matrices

	Structural Matrices					
Element Type	**Stiffness**	**Mass**	**Differential Stiffness**	**Viscous Damping**	**Axisymmetric**	**p-Adaptivity**
CQUAD8	I	LC	X			
CQUADR	I	LC				
CROD	CS	LC	X			
CSHEAR	CS	L	X			
CSLOTi					X	
CTETRA	I	LC	X			X
CTRIA3	I	LC	X			X
CTRIA6	I	LC	X			
CTRIAR	I	LC				
CTRIAX6	I	LC			X	
CTUBE	CS	LC	X			
CVISC				X		
CWELD	X	X				

Table 1-2. Element Summary – Small Strain Elements, Materials

	Materials		
Element Type	**Isotropic**	**Anisotropic**	**Orthotropic**
CAXIFi			
CBAR	X		
CBEAM	X		
CBUSH			
CBUSH1D	X		
CBEND	X		
CCONEAX	X	X	X
CONMi			
CONROD	X		
CRAC2D	X	X	
CRAC3D	X	X	
CDAMPi			
CELASi	X		
CFLUIDi			
CGAP			
CHBDYi			
CHEXA	X	X	
CMASSi			
CPENTA	X	X	
CQUAD4	X	X	X
CQUAD8	X	X	X

Table 1-2. Element Summary – Small Strain Elements, Materials

	Materials		
Element Type	**Isotropic**	**Anisotropic**	**Orthotropic**
CQUADR	X	X	X
CROD	X		
CSHEAR	X		
CSLOTi			
CTETRA	X	X	
CTRIA3	X	X	X
CTRIA6	X	X	X
CTRIAR	X	X	X
CTRIAX6	X		X
CTUBE	X		
CVISC			
CWELD	X	X	

Table 1-3. Element Summary – Small Strain Elements, Static Load and Heat Transfer

	Static Load				Heat Transfer		
Element Type	**Thermal**	**Pressure**	**Gravity**	**Element Deformation**	**Heat Conduction**	**Heat Capacity**	**Thermal Load**
CAXIFi							
CBAR	EB	X	X	X	X	X	X
CBEAM	EB	X	X	X	X	X	X
CBUSH							
CBUSH1D							
CBEND	EB	X	X		X	X	X
CCONEAX	E	X	X				
CONMi			X				
CONROD	E		X	X	X	X	X
CRAC2D	E		X				
CRAC3D	E		X				
CDAMPi						X	
CELASi					X		
CFLUIDi							
CGAP							
CHBDYi					X	X	X
CHEXA	E	X	X		X	X	X
CMASSi			X				
CPENTA	E	X	X		X	X	X
CQUAD4	EB				X	X	X
CQUAD8	EB	X	X		X	X	X
CQUADR	EB	X	X				

Table 1-3. Element Summary – Small Strain Elements, Static Load and Heat Transfer

	Static Load				Heat Transfer		
Element Type	**Thermal**	**Pressure**	**Gravity**	**Element Deformation**	**Heat Conduction**	**Heat Capacity**	**Thermal Load**
CROD	E		X	X	X	X	X
CSHEAR	E	X	X				
CSLOTi							
CTETRA	E	X	X		X	X	X
CTRIA3	EB	X	X		X	X	X
CTRIA6	EB	X	X		X	X	X
CTRIAR	EB	X	X				
CTRIAX6	E	X	X		X	X	X
CTUBE	E		X	X	X	X	X
CVISC							
CWELD							

Table 1-4. Element Summary – Data Recovery

	Data Recovery								
Element Type	**Stress Real***	**Stress Complex***	**Force Real***	**Force Complex***	**Force Sum Global**	**Force Sum Element Edges**	**Structure Plot**	**Contour Plot**	**Grid Point Stresses**
CAXIFi	12	23							
CBAR	16	19	9	17	X	X	X		
CBEAM	89	111	89	177	X		X		
CBEND	23	23	17	31	X		X		
CBUSH	7	13	7	13	X				
CBUSH1D	8				X				
CCONEAX**	18		7				X		
CONMi									
CONROD	5	5	3	5	X	X	X		
CRAC2D	8						X		
CRAC3D	10								
CDAMPi			2	3					
CELASi	2	3	2	3	X				
CFLUIDi							X		
CGAP	8						X		
CHBDYP, CHBDYG							X		
CHEXA	193	121			X	X	X		X
CMASSi									
CPENTA	151	95			X	X	X		X
CQUAD4	17	15	9	17	X	X	X	X	X
CQUAD8	87	77	47	87	X		X		X
CQUADR	87	77	47	87	X		X		X

Table 1-4. Element Summary – Data Recovery

	Data Recovery								
Element Type	**Stress Real***	**Stress Complex***	**Force Real***	**Force Complex***	**Force Sum Global**	**Force Sum Element Edges**	**Structure Plot**	**Contour Plot**	**Grid Point Stresses**
CROD	5	5	3	5	X	X	X		
CSHEAR	8	5	17	33	X	X	X	X	
CSLOTi	7	13					X		
CTETRA	109	69			X		X		X
CTRIA3	17	15	9	17	X	X	X	X	X
CTRIA6	70	62	38	70	X		X		X
CTRIAR	70	62	38	70	X	X	X		X
CTRIAX6	30	34			X		X	X	
CTUBE	5	5	3	5	X	X	X	X	
CVISC				5			X		
CWELD			9	17					

Note

* The integers represent the number of output words per element, useful for storage requirement calculations.

** This requires the presence of an AXISYM Case Control command.

Table 1-5. Element Summary – Data Recovery(continued)

	Data Recovery									
Element Type	**Strain Energy Density**	**Strain Energy**	**SORT2**	**MSGSTRESS**	**Heat Transfer**	**MSGMESH**	**Composite* Failure Indices & Strength Ratios**	**Composite* Stresses**	**Strain Real***	**Strain Complex***
CAXIFi			X							
CBAR	X	X	X		X	X				
CBEAM	X	X	X		X	X				
CBEND	X	X	X	X	X					
CCONEAX**										
CBUSH		X	X							
CBUSH1D		X	X						8	
CONMi										
CONROD	X	X	X		X		X			
CRAC2D										
CRAC3D										
CDAMPi			X							
CELASi		X	X		X					
CFLUIDi							X			

Table 1-5. Element Summary – Data Recovery(continued)

	Data Recovery									
Element Type	**Strain Energy Density**	**Strain Energy**	**SORT2**	**MSGSTRESS**	**Heat Transfer**	**MSGMESH**	**Composite* Failure Indices & Strength Ratios**	**Composite* Stresses**	**Strain Real***	**Strain Complex***
CGAP		X	X							
CHBDYi			X		X	X				
CHEXA	X	X	X	X	X	X			193	121
CMASSi										
CPENTA	X	X	X	X	X	X			151	95
CQUAD4	X	X	X	X	X	X	9	11	17	15
CQUAD8	X	X	X		X	X	9	11	87	77
CQUADR	X	X	X			X			87	77
CROD	X	X	X		X	X				
CSHEAR	X	X	X			X				
CSLOTi			X							
CTETRA	X	X	X		X				109	69
CTRIA3	X	X	X	X	X	X	9	11	17	15
CTRIA6	X	X	X		X	X	9	11	70	62
CTRIAR	X	X	X			X			70	62
CTRIAX6	X	X	X	X	X	X				
CTUBE	X	X	X		X	X				
CVISC										
CWELD	X	X	X							

Note

* The integers represent the number of output words per element, useful for storage requirement calculations.

** This requires the presence of an AXISYM Case Control command.

Legend for Tables 1–1 through 1–5		
Stiffness Matrix	**Mass Matrix**	**Thermal Load**
CS – Constant strain element	L – Lumped mass only	E – Extension load only
LS – Linear strain element	C – Coupled mass only	EB – Both extension and bending load
I – Modified isoparametric element	LC – Lumped mass or coupled mass	
FD – Frequency dependent		

Integer values indicate number of items output.

Chapter 2: 0D Elements

- *Overview of 0D (Scalar) Elements*
- *Spring Elements*
- *Damping Elements*
- *Mass Elements*

2.1 Overview of 0D (Scalar) Elements

A 0D or scalar element is an element that connects two degrees of freedom in the structure or one degree of freedom and a ground. The degrees of freedom may be any of the six components of a grid point or the single component of a scalar point. Scalar elements lack geometric definition and therefore do not have an element coordinate system.

Scalar elements are available as springs, masses, and viscous dampers.

- Scalar spring elements are useful for representing elastic properties that cannot be conveniently modeled with the usual structural elements (elements whose stiffnesses are derived from geometric properties).
- Scalar masses are useful for the selective representation of inertia properties, such as occurs when a concentrated mass is effectively isolated for motion in one direction only.
- Scalar dampers are used to provide viscous damping between two selected degrees-of-freedom or between one degree-of-freedom and ground.

It is possible, using only scalar elements and constraints, to construct a model for the linear behavior of any structure. However, using scalar elements when offsets are defined on beam and shell elements will cause incorrect results in buckling analysis and differential stiffness because the large displacement effects are not calculated.

Offsets will also cause internal constraints in linear analysis, i.e., hidden constraints to ground. Therefore, you should only use scalar elements when the usual structural elements aren't satisfactory.

Scalar elements are useful for modeling part of a structure with its vibration modes or when trying to consider electrical or heat transfer properties as part of an overall structural analysis. Scalar elements are commonly used in conjunction with structural elements where the details of the physical structure are not known or required. Typical examples include shock absorbers, joint stiffness between linkages, isolation pads, and many others.

See *The NASTRAN Theoretical Manual* for further discussions on the use of scalar elements.

Whenever you define scalar elements between grid points, the grid points should be coincident. If the grid points aren't coincident, any forces applied to the grid point by the scalar element may induce moments on the structure. This can cause inaccurate results.

There are two types of scalar elements.

- Scalar elements whose properties are defined on the PELAS and PMASS entries. This includes CELAS1, CELAS3, CMASS1, and CMASS3.
    - The CELAS1 and CMASS1 can be defined on either grid and scalar points.
    - The CELAS3 and CMASS3 can only be defined on scalar points.
- Scalar elements whose properties are defined on the connection entries directly. This includes CELAS2, CELAS4, CMASS2, and CMASS4.
    - The CELAS2 and CMASS2 can be defined on either grid and scalar points.
    - The CELAS4 and CMASS4 can only be defined on scalar points.

As an example of the NX Nastran model consisting of only scalar points, consider the structure shown in Figure 2-1. In this example, the CROD elements are replaced with equivalent springs.

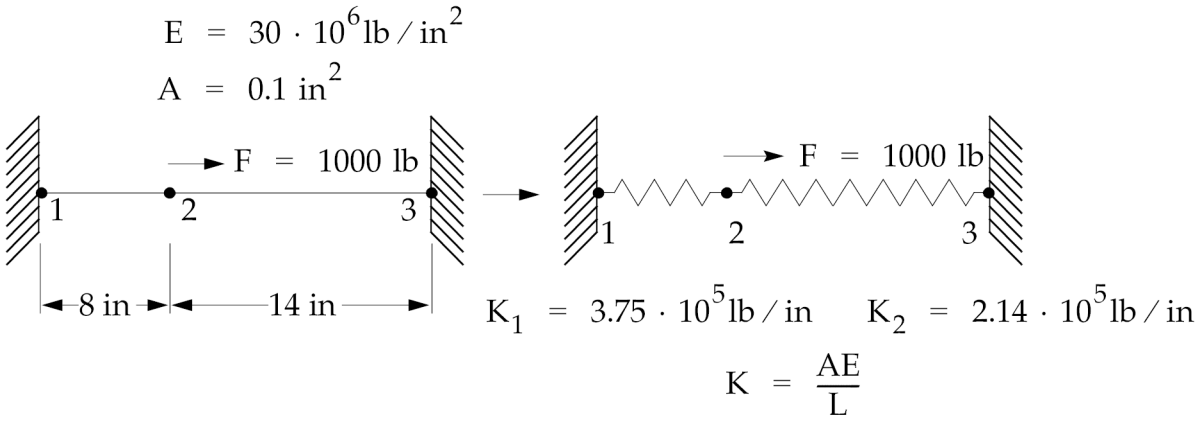


Figure 2-1. Equivalent Spring Model

The input file for the structure shown is given in Listing 2-1.

```
$ FILENAME - SPRING1.DAT
ID LINEAR,SPRING1
SOL 101
TIME 2
CEND
TITLE = NX NASTRAN USER'S GUIDE PROBLEM 4.1
SUBTITLE = ROD STRUCTURE MODELED USING SCALAR ELEMENTS
LABEL = POINT LOAD AT SCALAR POINT 2
LOAD = 1
SPC = 2
DISP = ALL
FORCE = ALL
BEGIN BULK
$
$THE RESEQUENCER IN NX NASTRAN REQUIRES AT LEAST ONE GRID POINT
$IN THE MODEL. IT IS FULLY CONSTRAINED AND WILL NOT AFFECT THE RESULTS
$
GRID    99              0.      0.      0.              123456
$
$ THE SCALAR POINTS DO NOT HAVE GEOMETRY
$
SPOINT  1       2       3
$
$ MEMBERS ARE MODELED SPRING ELEMENTS
$
CELAS4  1       3.75E5  1       2
CELAS4  2       2.14E5  2       3
$
$ POINT LOAD
$
SLOAD   1       2       1000.
$
SPC1    2       0       1       3
$
ENDDATA
```

Listing 2-1. Equivalent Spring Model

This example is used only to demonstrate the use of the scalar element. In general, you wouldn't replace structural elements with scalar springs. If this were an actual structure being analyzed, the preferred method would be to use CROD elements. The resulting output for this model is given in Figure 2-2.

```
                                    D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      S      0.0            1.697793E-03   0.0
           99      G      0.0            0.0            0.0            0.0            0.0            0.0


                              F O R C E S   I N   S C A L A R   S P R I N G S       ( C E L A S 4 )
       ELEMENT          FORCE          ELEMENT          FORCE          ELEMENT          FORCE          ELEMENT          FORCE
         ID.                             ID.                             ID.                             ID.
            1     -6.366724E+02              2      3.633277E+02
```

Figure 2-2. The Equivalent Spring Model Output

The displacement vector output also includes the displacement results for the scalar points. Since a scalar point has only one degree of freedom, NX Nastran displays up to six scalar points displacements per line. For the output shown in Figure 2-2, the displacement of scalar points 1, 2, and 3 are labeled T1, T2, and T3, respectively. The POINT ID of 1 is the ID of the first scalar point in that row. The TYPE "S" indicates that all the output in that row is scalar point output.

The sign convention for the scalar force and stress results is determined by the order of the scalar point IDs on the element connectivity entry. The force in the scalar element is computed by Eq. 2-1.

$$F_{\text{SPRING}} = K(U_1 - U_2)$$

Equation 2-1.

For the equivalent spring model, the force in element 1 is found to be

$$3.75 \cdot 10^5 \cdot (0.0 - 0.001698) = -636.7 \text{ lb}$$

This result agrees with the force shown in Figure 2-2. If you reverse the order of SPOINT ID 1 and 2 in the CELAS2 entry for element 1, the force in the spring will be

$$3.75 \cdot 10^5 \cdot (0.001698 - 0.0) = 636.7 \text{ lb}$$

Neither answer is wrong-they simply follow the convention given by Eq. 2-1.

The input file "spring1.dat" is included on the NX Nastran delivery CD. The same structure modeled with two CROD elements is available in the "rod1.dat" file in the Test Problem Library.

2.2 Spring Elements

Spring elements connect two degrees of freedom at two different grid point. They behave like simple extension/compression or rotational (e.g. clock) springs, carrying either force or moment loads. Forces result in translational (axial) displacement and moments result in rotational displacement.

You can create the most general definition of a scalar spring with a CELAS1 entry. The associated properties are given on the PELAS entry. The properties include the magnitude of the elastic spring, a damping coefficient, and a stress coefficient to be used in stress recovery. The CELAS2 defines a scalar spring without reference to a property entry. The CELAS3 entry defines a scalar spring that is connected only to scalar points and the properties are given on a PELAS entry. The CELAS4 entry defines a scalar spring that is connected only to scalar points and without reference to a property entry. No damping coefficient or stress coefficient is available with the CELAS4 entry. Element force *F* is calculated from the equation

$$F = k(u_1 - u_2)$$

Equation 2-2.

where *k* is the stiffness coefficient for the scalar element and u_1 is the displacement of the first degree-of-freedom listed on its connection entry. Element stresses are calculated from the equation

$$\sigma = S \cdot F$$

Equation 2-3.

where *S* is the stress coefficient on the connection or property entry and *F* is as defined above.

Scalar elements may be connected to ground without the use of constraint entries. Grounded connections are indicated on the connection entry by leaving the appropriate scalar identification number blank. Since the values for scalar elements are not functions of material properties, no references to such entries are needed.

CELASi Formats

For static analysis, the linear scalar springs (CELASi, i = 1-4) and concentrated masses (CMASSi, i = 1-4) are useful. There are four types of scalar springs and mass definitions. The formats of the CELASi entries (elastic springs) are as follows:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Field	Contents
EID	Unique element identification number.
PID	Property identification number of a PELAS entry (CELAS1 and CELAS3).
G1, G2	Geometric grid point or scalar identification number (CELAS1 and CELAS2).

Field	Contents
C1, C2	Component number (CELAS1 and CELAS2).
S1, S2	Scalar point identification numbers (CELAS 3 and CELAS4).
K	Stiffness of the scalar spring (CELAS2 and CELAS4).
S	Stress coefficient (CELAS2).
GE	Damping coefficient (CELAS2).

The CELAS2 element, whose format is shown above, defines a spring and includes spring property data directly on the element entry.

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. (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. (Real)
S	Stress coefficient. (Real)

Entering a zero or blank for either (Gi, Ci) pair indicates a grounded spring. A grounded point is a point whose displacement is constrained to zero. Also, G1 and G2 cannot be the same GRID point (same ID), but they should be modeled coincident. Thus, spring elements do not have physical geometry in the same sense that beams, plates, and solids do, and that is why they are called zero dimensional.

PELAS Entry

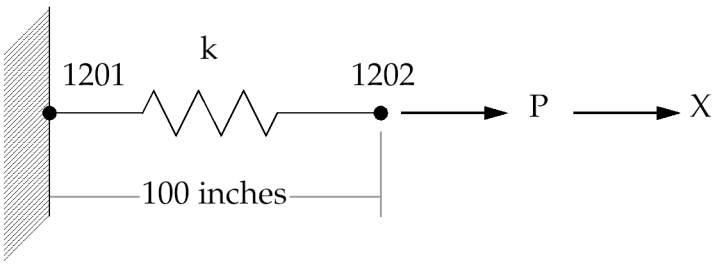
You use the PELAS bulk data entry to define properties for CELASi elements, such as their elastic property values, and damping and stress coefficients.

See Also

- PELAS in the *NX Nastran Quick Reference Guide*

CELAS2 Example

Consider the simple extensional spring shown in the following figure. One end is fixed and the other is subjected to a 10 lb_f axial load. The axial stiffness of the spring (k) is 100 lb_f/inch. What is the displacement of GRID 1202?



The required Bulk Data entries are specified as follows:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	
CELAS2	1200	100.	1201	1	1202	1			

GRID	1201		0.	0.	0.		123456		
GRID	1202		100.	0.	0.		23456		

GRID 1201 at the fixed wall is constrained in all 6 DOFs. GRID 1202 is constrained in DOFs 2 through 6 since the element it is connected to only uses DOF 1 (translation in the X-direction). Recall that a grid point is free in all six DOFs until it is told otherwise. Leaving any DOF of any GRID point “unattached”—either unconnected to an element’s stiffness or unconstrained by other means—results in a rigid body motion singularity failure in static analysis. The PARAM,AUTOSPC feature of Solution 101 automatically constrains these unconnected DOFs.

Note also that damping (GE in field 8) is not relevant to a static analysis and is therefore not included on this entry. The stress coefficient (S) in field 9 is an optional user-specified quantity. Supplying S directs NX Nastran to compute the spring stress using the relation P_s = S · P, where P is the applied load.

The grid point displacement and element force output is shown in Figure 2-3.

```
                                    D I S P L A C E M E N T   V E C T O R
POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
    1201      G      0.0            0.0            0.0            0.0            0.0            0.0
    1202      G      1.000000E-01   0.0            0.0            0.0            0.0            0.0


                          F O R C E S   I N   S C A L A R   S P R I N G S       ( C E L A S 2 )
ELEMENT       FORCE          ELEMENT       FORCE          ELEMENT       FORCE          ELEMENT       FORCE
  ID.                          ID.                          ID.                          ID.
   1200   -1.000000E+01
```

Figure 2-3. CELAS2 Spring Element Output

The displacement of GRID 1202 is 0.1 inches in the positive X-direction (the spring is in tension). The hand calculation is as follows:

$$u_x = \frac{P}{k} = \frac{10.lb_f}{100.lb_f/\text{inch}} = 0.1 \text{ inch}$$

The force in the spring element is calculated by NX Nastran as

$$f = k(u_x^1 - u_x^2) = 100.\ (0. - 0.1) = -10.0\ lb_f$$

where:

$u^1{}_x$ = displacement of G1
$u^2{}_x$ = displacement of G2

Reversing the order of G1 and G2 on the CELAS2 entry reverses the sign of the element force. The sign of force and stress output for scalar elements depends on how the grid points are listed (ordered) when you define an element, and not on a physical sense of tension or compression. This is not the case when you use line (one-dimensional) elements such as rods and beams. Therefore, you should be careful how you interpret signs when you use scalar elements.

See Also

- CELAS1 in the *NX Nastran Quick Reference Guide*
- CELAS2 in the *NX Nastran Quick Reference Guide*
- CELAS3 in the *NX Nastran Quick Reference Guide*
- CELAS4 in the *NX Nastran Quick Reference Guide*

2.3 Damping Elements

The CDAMP1, CDAMP2, CDAMP3, CDAMP4, and CDAMP5 entries define scalar dampers in a manner similar to the scalar spring definitions. The associated PDAMP entry contains only a value for the scalar damper. You must select the mode displacement method (PARAM,DDRMM,-1) for element force output.

See Also

- CDAMP1 in the *NX Nastran Quick Reference Guide*
- CDAMP2 in the *NX Nastran Quick Reference Guide*
- CDAMP3 in the *NX Nastran Quick Reference Guide*
- CDAMP4 in the *NX Nastran Quick Reference Guide*
- CDAMP5 in the *NX Nastran Quick Reference Guide*
- DDRMM in the *NX Nastran Quick Reference Guide*

2.4 Mass Elements

The CMASS1, CMASS2, CMASS3, and CMASS4 entries define scalar masses in a manner similar to the scalar spring definitions. The associated PMASS entry contains only the magnitude of the scalar mass.

See Also

- CMASS1 in the *NX Nastran Quick Reference Guide*
- CMASS2 in the *NX Nastran Quick Reference Guide*
- CMASS3 in the *NX Nastran Quick Reference Guide*
- CMASS4 in the *NX Nastran Quick Reference Guide*

- PMASS in the *NX Nastran Quick Reference Guide*

Chapter 3: 1D Elements

- *Overview of 1D (Line) Elements*
- *CBAR Element*
- *CBEAM Element*
- *CBEND Element*
- *CONROD Rod Element*
- *CROD Element*
- *CTUBE Element*
- *CVISC Element*

3.1 Overview of 1D (Line) Elements

Line elements, also called one-dimensional elements, are used to represent rod and beam behavior. One-dimensional elements are used to represent structural members that have stiffness along a line or curve between two grid points. Typical applications include beam type structures, stiffeners, tie-down members, supports, mesh transitions, and many others.

The one-dimensional elements in NX Nastran include:

- CBAR
- CBEAM
- CBEND
- CONROD
- CROD
- CTUBE
- CVISC

A rod element supports tension, compression, and axial torsion, but not bending. A beam element includes bending. NX Nastran makes an additional distinction between “simple” beams and “complex” beams.

- Simple beams are modeled with the CBAR element and require that beam properties do not vary with cross section. The CBAR element also requires that the shear center and neutral axis coincide and is therefore not useful for modeling beams that warp, such as open channel sections.
- Complex beams are modeled with the CBEAM element, which has all of the CBAR’s capabilities plus a variety of additional features. CBEAM elements permit tapered cross-sectional properties, a noncoincident neutral axis and shear center, and cross-sectional warping.

3.2 CBAR Element

In NX Nastran, a bar element is known as a CBAR. The CBAR element is a general purpose beam that supports tension and compression, torsion, bending in two perpendicular planes, and shear in two perpendicular planes. The CBAR uses two grid points and can provide stiffness to all six DOFs of each grid point. With the CBAR, its elastic axis, gravity axis, and shear center all coincide. The displacement components of the grid points are three translations and three rotations.

You define a CBAR element using the CBAR bulk data entry and define its properties using the PBAR bulk data entry.

CBAR Characteristics and Limitations

- Its formulation is derived from classical beam theory (plane cross sections remain plane during deformation).

- It must be straight and prismatic. The properties must be constant along the length of the CBAR element. This limitation is not present in the CBEAM element.
- The shear center and neutral axis must coincide (the CBAR element cannot model warping of open sections). This limitation is not present in the CBEAM element.
- Extensional stiffness along the neutral axis and torsional stiffness about the neutral axis may be defined.
- Torsional stiffening of out-of-plane cross-sectional warping is neglected. This limitation is not present in the CBEAM element
- You can define both bending and transverse shear stiffness in the two perpendicular directions to the CBAR element's axial direction.
- The principal axis of inertia doesn't need to coincide with the element axis.
- The neutral axis may be offset from the grid points (an internal rigid link is created). This is useful for modeling stiffened plates or gridworks.
- A pin flag capability is available to provide a moment or force release at either end of the element (this permits the modeling of linkages or mechanisms).
- You can compute the stress at up to four locations on the cross section at each end. Additionally, you can use the CBARAO bulk data entry to request output for intermediate locations along the length of the CBAR.

CBAR Format

Two formats of the CBAR entry are available, as shown below:

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	

Alternate Format:

CBAR	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PBAR entry. (Integer > 0 or blank; Default is EID unless BAROR entry has nonzero entry in field 3)
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. (Integer > 0; G0 ≠ GA or GB)
PA, PB	Pin flags for bar ends A and B, respectively. Removes 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. 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, W3AW1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$, respectively in displacement coordinate systems at points GA and GB, respectively. (Real or blank).

See Also

- CBAR in the *NX Nastran Quick Reference Guide*

CBAR Element Coordinate System and Orientation

With a CBAR (or CBEAM) element, you must define an orientation vector to orient the element in space. This vector also specifies the local element coordinate system. Since you enter the element's geometric properties in the element coordinate system, this orientation vector specifies the orientation of the element.

This example illustrates the importance of the orientation vector. Consider the two I-beams shown below. The I-beams have the same properties because they have the same dimensions. However, since they have different orientations in space, their stiffness contribution to the structure is different. Therefore, it's critical to orient beam elements correctly.

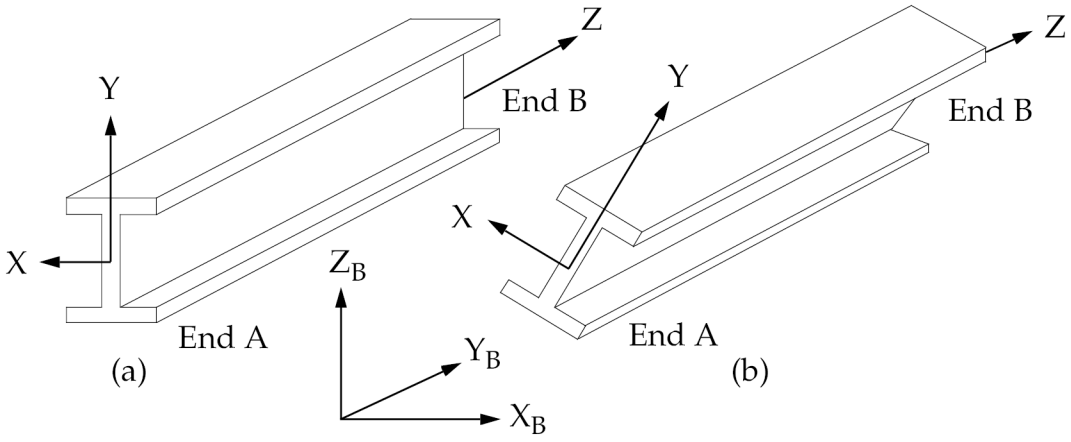


Figure 3-1. Demonstration of Beam Orientation

The orientation vector $\vec{v}$ as it is related to the CBAR element coordinate system is shown in Figure 3-2. A vector defines plane 1, which contains the elemental x- and y-axes.

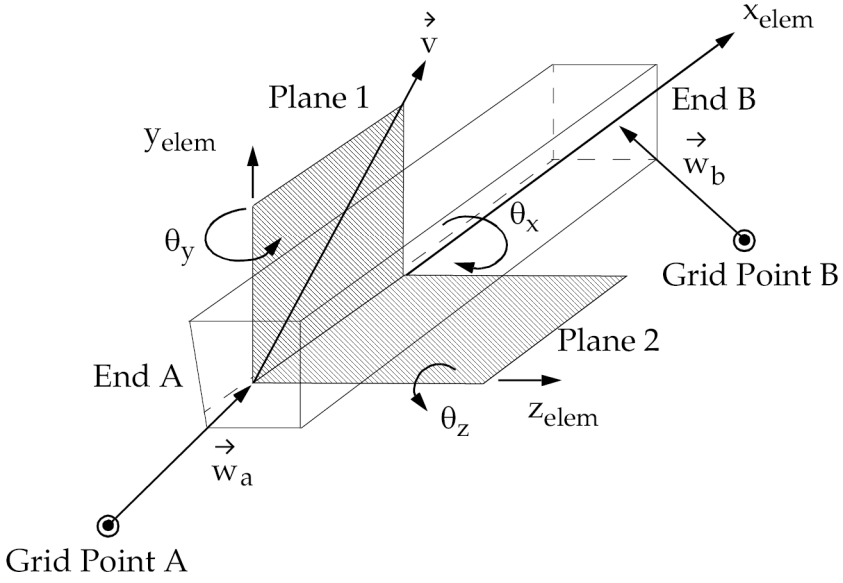


Figure 3-2. CBAR Element Coordinate System

Referring to Figure 3-2, the element's x-axis is defined as the line extending from end A (the end at grid point GA) to end B (the end at grid point GB). You define grid points GA and GB in fields 4 and 5 on the CBAR entry. You can also offset the ends of the CBAR element from the grid points using WA and WB as defined on the CBAR entry. Therefore, the element's x-axis doesn't necessarily extend from grid point GA to grid point GB. It extends from end A to end B.

The element's y-axis is defined to be the axis in Plane 1. Plane 1 extends from end A and is perpendicular to the element's x-axis. You must define Plane 1. Plane 1 is the plane that contains the element's x-axis and the orientation vector $\vec{v}$. After defining the element x- and y-axes, the element z-axis is obtained using the right-hand rule.

The plane formed by the element x-axis and orientation vector $\vec{v}$ is called plane 1. The element y-axis lies in plane 1 and is perpendicular to the element x-axis

Finally, plane 2 is perpendicular to plane 1, and the element z-axis is formed by the cross product of the x and y element axes. Plane 2 contains the element x and z axes.

Plane 2 is the plane containing element's x- and z-axes. Note that once you defined grid point GA and GB and the orientation vector $\vec{v}$, the element coordinate system is computed automatically by NX Nastran.

You can define the vector $\vec{v}$ shown in Figure 3-2 by one of two methods on the CBAR entry.

- You can define vector by entering the components of the vector, (X1, X2, X3), which is defined in a coordinate system located at the end of the CBAR. You enter X1, X2, and X3 in fields 6-8 of the CBAR entry. This coordinate system is parallel to the displacement coordinate system of the grid point GA, you define in field 7 of the GRID entry.

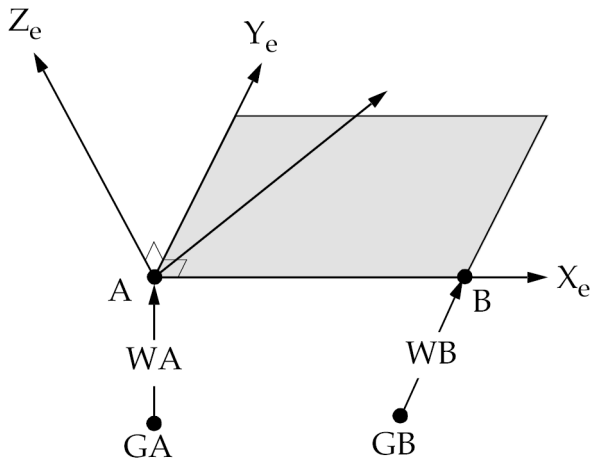
  The direction of $\vec{v}$ with respect to the cross section is arbitrary, but $\vec{v}$ is normally aligned with one of the beam's principal planes of inertia.

- You can define the vector $\vec{v}$ using another grid point, G0, you specify in field 6 of the CBAR entry.

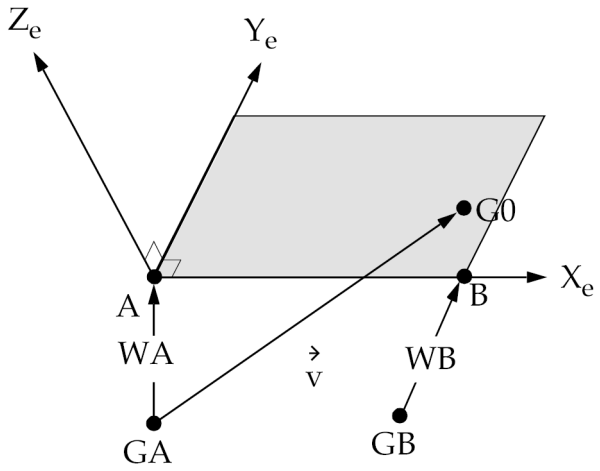
Defining CBAR End Offsets

You can offset the ends of the CBAR from the grid points using the vectors WA and WB. When you specify an offset, you are effectively defining a rigid connection from the grid point to the end of the element.

- If the CBAR is offset from the grid points and you entered the components of vector $\vec{v}$ in fields 6-8, then the tail of vector $\vec{v}$ is at end A, not grid point GA.



- If the CBAR is offset from the grid points and you defined the vector $\vec{v}$ using another grid point G0, then vector $\vec{v}$ is defined as the line originating at grid point GA, not end A, and passing through G0. Note that Plane 1 is parallel to the vector GA-G0 and passes through the location of end A.



You enter the offsets values WA and WB by specifying the components of an offset vector in the displacement coordinate systems for GA and GB, respectively. You enter the three components of the offset vectors using fields 4 through 9 of the CBAR continuation entry.

CBAR Force and Moment Conventions

The CBAR element force and moment conventions are shown in Figure 3-3 and Figure 3-4. If shearing deformations are included in the CBAR element, the reference axes (Planes 1 and 2) and the principal axes must coincide.

The element forces and stresses are computed and output in the element coordinate system. The following figures show the forces acting on the CBAR element. V_1 and M_1 are the shear force and bending moment acting in Plane 1, and V_2 and M_2 are the shear force and bending moment acting in Plane 2.

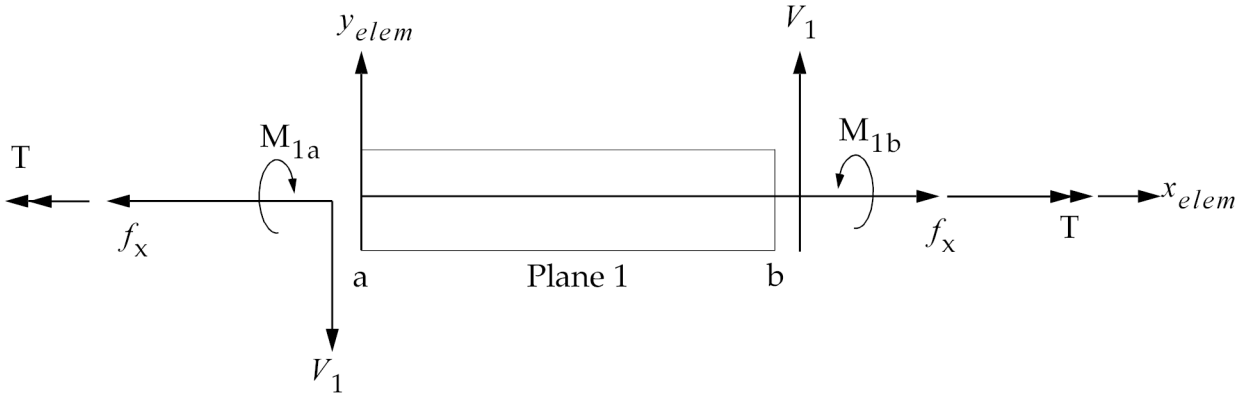


Figure 3-3. CBAR Element Internal Forces and Moments (x-y Plane)

where M_{1a}, M_{1b}, M_{2a}, and M_{2b} are the bending moments at both ends in the two reference planes, V_1 and V_2 are the shear forces in the two reference planes, F_x is the average axial force, and T is the torque about the x-axis

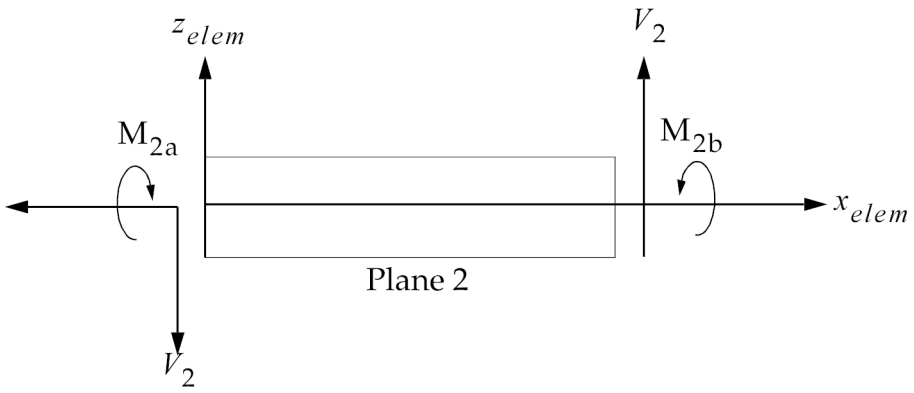


Figure 3-4. CBAR Element Internal Forces and Moments (x-z Plane)

NX Nastran outputs the following element forces, either real or complex (depending on the rigid format), on request:

- Average axial stress.
- Extensional stress due to bending at four points on the cross section at both ends. (Optional, calculated only if you enter stress recovery points on the PBAR entry.)
- Maximum and minimum extensional stresses at both ends.
- Margins of safety in tension and compression for the whole element. (Optional, calculated only if the you enter stress limits on the MAT1 entry.)

Tensile stresses are given a positive sign and compressive stresses a negative sign. Only the average axial stress and the extensional stresses due to bending are available as complex stresses. The stress recovery coefficients on the PBAR entry are used to locate points on the cross section for stress recovery. The subscript 1 is associated with the distance of a stress recovery point from Plane 2. The subscript 2 is associated with the distance from Plane 1.

You can obtain CBAR element force and stress data recovery with distributed loads (PLOAD1) and distributed mass (coupled mass) effects included at intermediate as well as end points from the dynamic solution sequences. You must include the following items in the input file:

- A LOADSET in Case Control which selects an LSEQ entry referencing PLOAD1 entries in the Bulk Data Section.
- Use PARAM,COUPMASS to select the coupled mass option for all elements.

You enter the area moments of inertia I1 and I2 in fields 5 and 6, respectively, of the PBAR entry. I1 is the area moment of inertia to resist a moment in Plane 1. I1 is not the moment of inertia about Plane 1. Consider the cross section shown in Figure 3-12; in this case, I1 is what most textbooks call $I_{zz,}$ and I2 is I_{yy} .

You can enter the area product of inertia I12, if needed, using field 4 of the second continuation entry. For most common engineering cross sections, it isn't usually necessary to define an I12. By aligning the element y- and the z-axes with the principal axes of the cross section, I12, is equal to zero and is therefore not needed.

Using Pin Flags to Remove Selected Connections

The CBAR element also lets you remove some of the connections of individual degrees of freedom from the grid points. This is accomplished using the pin flags feature located in the CBAR entry (fields 2 and 3 of the continuation entry). For example, suppose you want to connect two bar elements together with a hinge (or pin joint) as shown in Figure 3-5. This connection can be made by placing an integer 456 in the PB field of CBAR 1 or a 456 in the PA field of CBAR 2.

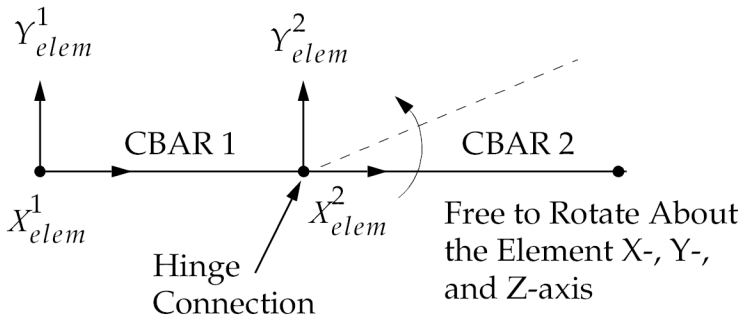


Figure 3-5. A Hinge Connection

The sample bridge model shown in Figure 3-6 is used to illustrate the application of pin flags. In this case, the rotational degree of freedom, θ_z at end B of the braces (grid points 7 and 16) connected to the horizontal span, is released. Note that these degrees of freedoms are referenced in terms of the element coordinate systems. A copy of this input file is shown in Listing 3-1. The deflected shapes for the cases with and without release are shown in Figure 3-7, and Figure 3-8, respectively. The corresponding abridged stress outputs for the cases with and without releases are shown in Figure 3-9 and Figure 3-10, respectively. Elements 30 and 40 are the two brace elements that are connected to the horizontal span. Note that in the case with releases at ends B, there's no moment transfer to the brace (grid points 7 and 16) at these locations. The moments, however, are transferred across the horizontal span (elements 6, 7, 14, and 15). For clarity, only the pertinent element and grid numbers.

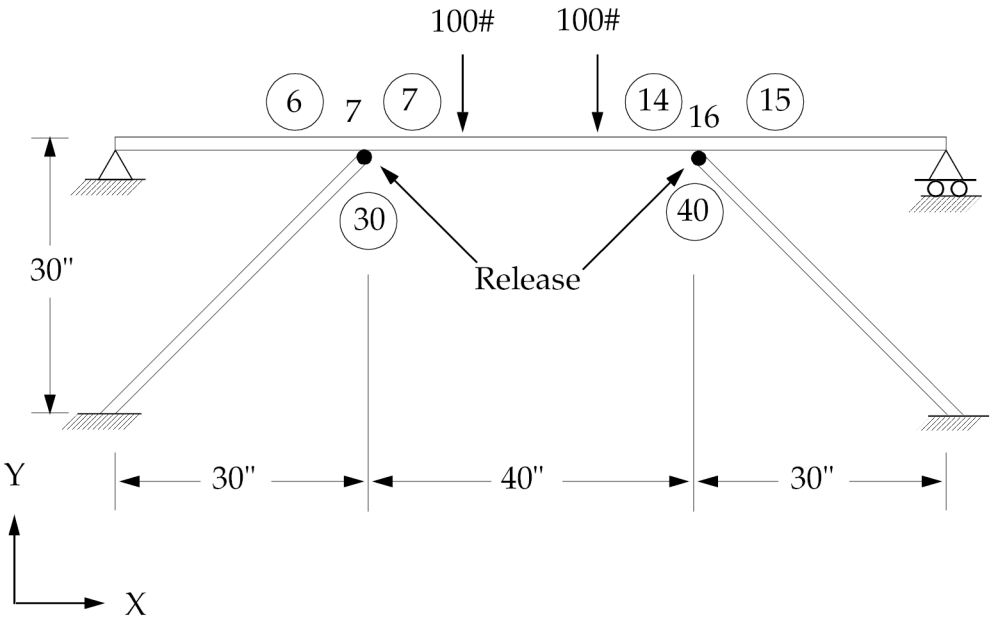


Figure 3-6. Bridge Model Demonstrating the Use of Release

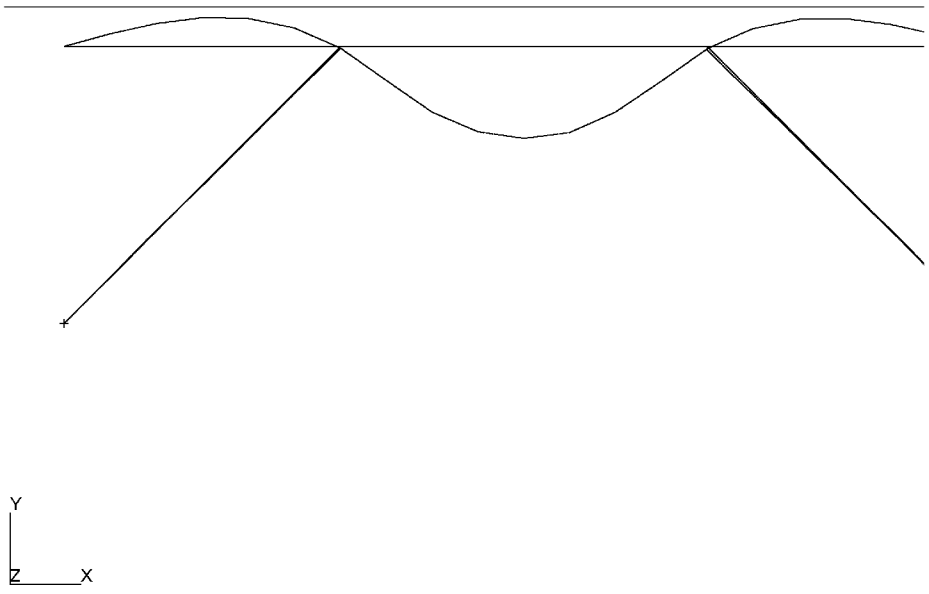


Figure 3-7. Deflected Shape of Bridge with Release at Brace

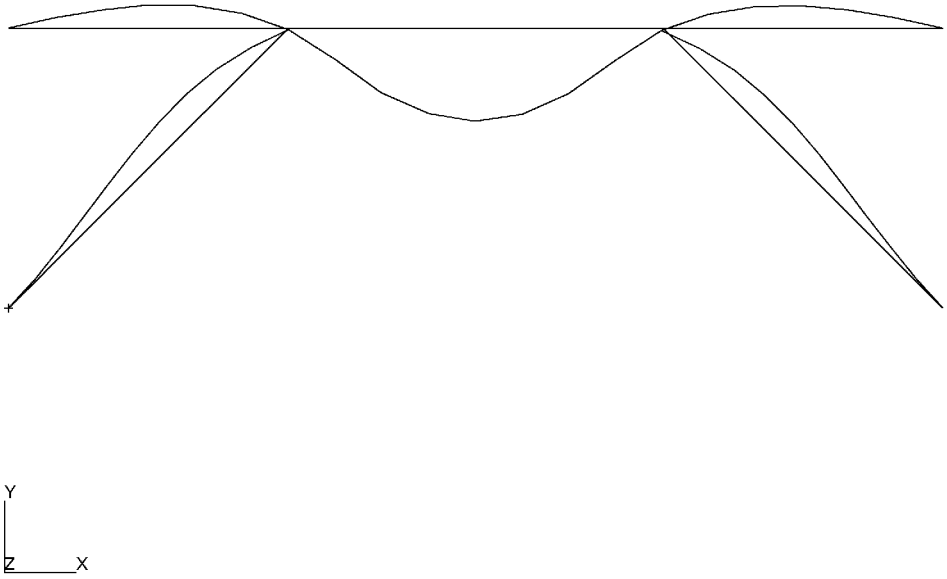


Figure 3-8. Deflected Shape of Bridge without Release at Brace

```
$    bridge1.dat
SOL 101
TIME 60
CEND
TITLE = BRIDGE MODEL - RELEASE FOR BRACE AT SPAN INTERSECTION
SUBCASE 1
   SUBTITLE=Default
   SPC = 2
   LOAD = 1
   DISPLACEMENT=ALL
   SPCFORCES=ALL
   STRESS=ALL
BEGIN BULK
PARAM    POST    -1
PBARL    1       1               I
+      A
+      A 1.      1.      1.      .1      .1      .1
MAT1  1    3.E7 .32
CBAR     1       1       1       2       0.      1.      0.
CBAR     2       1       2       3       0.      1.      0.
CBAR     3       1       3       4       0.      1.      0.
CBAR     4       1       4       5       0.      1.      0.
CBAR     5       1       5       6       0.      1.      0.
CBAR     6       1       6       7       0.      1.      0.
CBAR     7       1       7       9       0.      1.      0.
CBAR     8       1       9       10      0.      1.      0.
CBAR     9       1       10      11      0.      1.      0.
CBAR     10      1       11      12      0.      1.      0.
CBAR     11      1       12      13      0.      1.      0.
CBAR     12      1       13      14      0.      1.      0.
CBAR     13      1       14      15      0.      1.      0.
CBAR     14      1       15      16      0.      1.      0.
CBAR     15      1       16      18      0.      1.      0.
CBAR     16      1       18      19      0.      1.      0.
CBAR     17      1       19      20      0.      1.      0.
CBAR     18      1       20      21      0.      1.      0.
CBAR     19      1       21      22      0.      1.      0.
CBAR     20      1       22      23      0.      1.      0.
CBAR     21      1       24      25      0.      1.      0.
CBAR     22      1       25      26      0.      1.      0.
CBAR     23      1       26      27      0.      1.      0.
CBAR     24      1       27      28      0.      1.      0.
CBAR     25      1       28      29      0.      1.      0.
CBAR     26      1       29      30      0.      1.      0.
CBAR     27      1       30      31      0.      1.      0.
CBAR     28      1       31      32      0.      1.      0.
CBAR     29      1       32      33      0.      1.      0.
CBAR     30      1       33      7       0.      1.      0.
             6
CBAR     31      1       35      36      0.      1.      0.
CBAR     32      1       36      37      0.      1.      0.
CBAR     33      1       37      38      0.      1.      0.
CBAR     34      1       38      39      0.      1.      0.
CBAR     35      1       39      40      0.      1.      0.
CBAR     36      1       40      41      0.      1.      0.
CBAR     37      1       41      42      0.      1.      0.
CBAR     38      1       42      43      0.      1.      0.
CBAR     39      1       43      44      0.      1.      0.
CBAR     40      1       44      16      0.      1.      0.
             6
$ Nodes of the Entire Model
GRID     1               0.      30.     0.
GRID     2               5.      30.     0.
GRID     3               10.     30.     0.
GRID     4               15.     30.     0.
GRID     5               20.     30.     0.
```

```
GRID     6               25.0000 30.     0.
GRID     7               30.     30.     0.
GRID     9               35.     30.     0.
GRID     10              40.     30.     0.
GRID     11              45.     30.     0.
GRID     12              50.     30.     0.
GRID     13              55.     30.     0.
GRID     14              60.     30.     0.
GRID     15              65.     30.     0.
GRID     16              70.     30.     0.
GRID     18              75.     30.     0.
GRID     19              80.     30.     0.
GRID     20              85.     30.     0.
GRID     21              90.     30.     0.
GRID     22              95.     30.     0.
GRID     23              100.    30.     0.
GRID     24              0.      0.      0.
GRID     25              3.      3.      0.
GRID     26              6.      6.      0.
GRID     27              9.      9.      0.
GRID     28              12.0000 12.0000 0.
GRID     29              15.     15.     0.
GRID     30              18.     18.     0.
GRID     31              21.     21.     0.
GRID     32              24.0000 24.0000 0.
GRID     33              27.     27.     0.
GRID     35              100.    0.      0.
GRID     36              97.     3.      0.
GRID     37              94.     6.      0.
GRID     38              91.     9.      0.
GRID     39              88.     12.0000 0.
GRID     40              85.     15.     0.
GRID     41              82.     18.     0.
GRID     42              79.     21.     0.
GRID     43              76.     24.0000 0.
GRID     44              73.     27.     0.
$ Loads for Load Case : Default
SPCADD   2       1       3       4
$ Displacement Constraints of Load Set : disp1
SPC1     1       12345   1
$ Displacement Constraints of Load Set : disp2
SPC1     3       123456  24      35
$ Displacement Constraints of Load Set : disp3
SPC1     4       2345    23
$ Nodal Forces of Load Set : load1
FORCE    1       10      0       100.    0.      -1.     0.
FORCE    1       14      0       100.    0.      -1.     0.
ENDDATA
```

Listing 3-1. Bridge with Release at Brace

ELEMENT ID.		SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
0	1	.0	.0	.0	.0	1.298114E+00	1.298114E+00	1.298114E+00	
		9.251410E+02	-9.251410E+02	-9.251410E+02	9.251410E+02		9.264391E+02	-9.238429E+02	
0	2	9.251410E+02	-9.251410E+02	-9.251410E+02	9.251410E+02	1.298114E+00	9.264391E+02	-9.238429E+02	
		1.850282E+03	-1.850282E+03	-1.850282E+03	1.850282E+03		1.851580E+03	-1.848984E+03	
				.					
				.					
0	6	4.625705E+03	-4.625705E+03	-4.625705E+03	4.625705E+03	1.298114E+00	4.627003E+03	-4.624407E+03	
		5.550846E+03	-5.550846E+03	-5.550846E+03	5.550846E+03		5.552144E+03	-5.549547E+03	
0	7	5.550846E+03	-5.550846E+03	-5.550846E+03	5.550846E+03	-4.153344E+02	5.135511E+03	-5.966180E+03	
		-1.805041E+01	1.805041E+01	1.805041E+01	-1.805041E+01		-3.972839E+02	-4.333848E+02	
				.					
				.					
0	14	-4.863751E+01	4.863751E+01	4.863751E+01	-4.863751E+01	-4.153344E+02	-3.666968E+02	-4.639719E+02	
		5.510063E+03	-5.510063E+03	-5.510063E+03	5.510063E+03		5.094729E+03	-5.925397E+03	
0	15	5.510063E+03	-5.510063E+03	-5.510063E+03	5.510063E+03	.0	5.510063E+03	-5.510063E+03	
		4.591719E+03	-4.591719E+03	-4.591719E+03	4.591719E+03		4.591719E+03	-4.591719E+03	
				.					
				.					
0	30	2.083161E+00	-2.083161E+00	-2.083161E+00	2.083161E+00	-5.893649E+02	-5.872817E+02	-5.914481E+02	
		-7.759377E-08	7.759377E-08	7.759377E-08	-7.759377E-08		-5.893649E+02	-5.893649E+02	
				.					
				.					
0	40	4.021125E+00	-4.021125E+00	-4.021125E+00	4.021125E+00	-5.876757E+02	-5.836545E+02	-5.916968E+02	
		-3.032399E-08	3.032399E-08	3.032399E-08	-3.032399E-08		-5.876757E+02	-5.876757E+02	

Figure 3-9. Abridged Stress Output of Bridge with Release at Brace

			S T R E S S E S I N B A R E L E M E N T S (C B A R)						
ELEMENT ID.		SA1 SB1	SA2 SB2	SA3 SB3	SA4 SB4	AXIAL STRESS	SA-MAX SB-MAX	SA-MIN SB-MIN	M.S.-T M.S.-C
0	1	1.265061E-12	-1.265061E-12	-1.265061E-12	1.265061E-12	2.220387E+00	2.220387E+00	2.220387E+00	
		5.686420E+02	-5.686420E+02	-5.686420E+02	5.686420E+02		5.708624E+02	-5.664216E+02	
0	2	5.686420E+02	-5.686420E+02	-5.686420E+02	5.686420E+02	2.220387E+00	5.708624E+02	-5.664216E+02	
		1.137284E+03	-1.137284E+03	-1.137284E+03	1.137284E+03		1.139504E+03	-1.135064E+03	
				.					
				.					
0	6	2.843210E+03	-2.843210E+03	-2.843210E+03	2.843210E+03	2.220387E+00	2.845430E+03	-2.840990E+03	
		3.411852E+03	-3.411852E+03	-3.411852E+03	3.411852E+03		3.414073E+03	-3.409632E+03	
0	7	6.634688E+03	-6.634688E+03	-6.634688E+03	6.634688E+03	-4.433274E+02	6.191360E+03	-7.078016E+03	
		1.062523E+03	-1.062523E+03	-1.062523E+03	1.062523E+03		6.191954E+02	-1.505850E+03	
				.					
				.					
0	14	1.012320E+03	-1.012320E+03	-1.012320E+03	1.012320E+03	-4.433274E+02	5.689928E+02	-1.455648E+03	
		6.567751E+03	-6.567751E+03	-6.567751E+03	6.567751E+03		6.124424E+03	-7.011079E+03	
0	15	3.384410E+03	-3.384410E+03	-3.384410E+03	3.384410E+03	.0	3.384410E+03	-3.384410E+03	
		2.820342E+03	-2.820342E+03	-2.820342E+03	2.820342E+03		2.820342E+03	-2.820342E+03	
				.					
				.					
0	30	2.742704E+03	-2.742704E+03	-2.742704E+03	2.742704E+03	-5.937782E+02	2.148926E+03	-3.336482E+03	
		3.222836E+03	-3.222836E+03	-3.222836E+03	3.222836E+03		2.629058E+03	-3.816614E+03	
				.					
				.					
0	40	2.711180E+03	-2.711180E+03	-2.711180E+03	2.711180E+03	-5.912410E+02	2.119939E+03	-3.302421E+03	
		3.183342E+03	-3.183342E+03	-3.183342E+03	3.183342E+03		2.592101E+03	-3.774583E+03	

Figure 3-10. Abridged Stress Output of Bridge without Release at Brace

Stress Recovery Points

The first continuation entry defines stress recovery coefficient points (Ci, Di, Ei, Fi) on the beam's cross section. These points are in the y-z plane of the element coordinate system as shown in Figure 3-11.

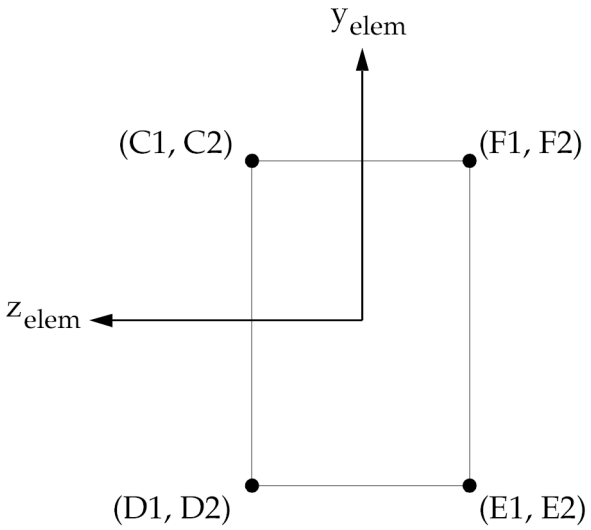


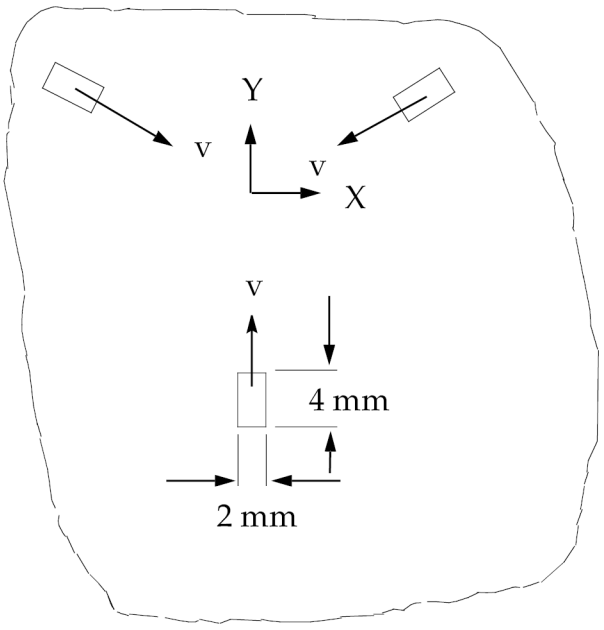
Figure 3-11. Stress Recovery Points on Beam Cross Section

By defining stress recovery points, you are providing c in the equation σ = Mc/I, thereby allowing NX Nastran to calculate stresses in the beam or on its surface.

CBAR Example

As an example, consider a three member truss structure. Now suppose the joints are rigidly connected so that the members are to carry a bending load. Since CROD elements cannot transmit a bending load, they cannot be used for this problem. The CBAR element is a good choice because it contains bending stiffness if you input area moments of inertia within the PBAR entry.

The dimensions and orientations of the member cross sections are shown in Figure 3-12.



All three cross sections are the same.

$A = 8\ mm^2$

$I1 = 10.67\ mm^4$

$I2 = 2.67\ mm^4$

$J = 7.324\ mm^4$

Figure 3-12. CBAR Cross Sections

Here, the area moments of inertia for each of the members are I1 = 10.67 and I2 = 2.67 for Plane 1 and Plane 2, respectively. The Bulk Data input for this problem is shown in Listing 3-2. The vector $\vec{v}$ for each of the CBAR elements is defined using the component method. This is indicated by the fact that field 6 of the CBAR entries contains a real number. Note that the length of vector $\vec{v}$ is not critical—only the orientation is required to define Plane 1.

```
$ FILENAME - BAR1.DAT
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID    1               -433.   250.    0.              123456
GRID    2               433.    250.    0.              123456
GRID    3               0.      -500.   0.              123456
GRID    4               0.      0.      1000.
$
$ MEMBERS ARE MODELED USING BAR ELEMENTS
$ VECTOR V DEFINED USING THE COMPONENT METHOD
$
CBAR    1       1       1       4       43.3    -25.    0.
CBAR    2       1       2       4       -43.3   -25.    0.
CBAR    3       1       3       4       0.      1.      0.
$
$ PROPERTIES  OF BAR ELEMENTS
$
PBAR    1       1       8.      10.67   2.67    7.324
        2.      1.      -2.     1.      2.      -1.     -2.     -1.
$
$ MATERIAL PROPERTIES
$
MAT1    1       19.9E4          .3
$
$ POINT LOAD
$
```

Listing 3-2. $\vec{v}$ Vector Entered Using the Components of the Vector.

```
FORCE   1       4               5000.   0.      -1.     0.
$
ENDDATA
```

All the orientation vectors point toward the geometric center of the bars.

Listing 3-3 shows how you can alternatively define the orientation vectors for this structure using the G0 method. Although the G0 grid point may be any grid point in the model, you should use a grid point that isn't attached to the structure. If the grid point is part of the structure, and the structure is modified, the vector orientation may be inadvertently changed, resulting in a modeling error. A modeling error of this nature is usually very difficult to identify. In this example, a new grid point with ID 99 was created and fixed at location (0,0,0). Both methods shown produce the same results.

```
$ FILENAME - BAR1A.DAT
$
GRID    1               -433.   250.    0.              123456
GRID    2               433.    250.    0.              123456
GRID    3               0.      -500.   0.              123456
GRID    4               0.      0.      1000.
GRID    99              0.      0.      0.              123456
$
$ MEMBERS ARE MODELED USING BAR ELEMENTS
$ VECTOR V DEFINED USING THE COMPONENT METHOD
$
CBAR    1       1       1       4       99
CBAR    2       1       2       4       99
CBAR    3       1       3       4       99
```

Listing 3-3. Vector Entered Using the G0 Method

The displacement and stress results are shown in Figure 3-13. The displacement of grid point 4 is in the YZ plane due to symmetry. However, the displacement of grid point 4 is slightly less because of the addition of the bending stiffness. Also, the rotations at grid point 4 are now nonzero values because they're connected to the structure and are free to move. Interestingly, the bending stiffness makes little difference for this structure. This difference results because the axial forces are the primary loads for this structure. For models such as these, you can save CPU time by using the CROD elements instead of the CBAR elements.

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	0.0	0.0	0.0	0.0	0.0	0.0
3	G	0.0	0.0	0.0	0.0	0.0	0.0
4	G	0.0	-1.170445E+01	-2.574947E-05	1.447189E-02	0.0	0.0

S T R E S S E S I N B A R E L E M E N T S (C B A R)

ELEMENT ID.	SA1	SA2	SA3	SA4	AXIAL	SA-MAX	SA-MIN	M.S.-T
	SB1	SB2	SB3	SB4	STRESS	SB-MAX	SB-MIN	M.S.-C
1	8.434532E-01	1.054002E+01	-1.054002E+01	-8.434532E-01	4.658372E+02	4.763772E+02	4.552972E+02	
	-2.004801E+00	-1.397634E+00	1.397634E+00	2.004801E+00		4.678420E+02	4.638324E+02	
2	-1.054002E+01	-8.434532E-01	8.434532E-01	1.054002E+01	4.658372E+02	4.763772E+02	4.552972E+02	
	1.397634E+00	2.004801E+00	-2.004801E+00	-1.397634E+00		4.678420E+02	4.638324E+02	
3	9.696071E+00	-9.696071E+00	9.696071E+00	-9.696071E+00	-9.316785E+02	-9.219824E+02	-9.413745E+02	
	6.073934E-01	-6.073934E-01	6.073934E-01	-6.073934E-01		-9.310710E+02	-9.322859E+02	

Figure 3-13. Displacement and Stress Results for the Three Member Bar Structure

The SA_i and SB_i are the bending stresses at ends A and B, respectively. The i = 1, 2, 3, and 4 stress recovery locations correspond to the locations C, D, E, and F on the cross section, respectively. The location of these stress recovery coefficients are defined in CBAR's element coordinate system. Consider the cross section for our example as shown in Figure 3-14. By request, the stresses are

computed at four locations. These stress locations represent the farthest points from the neutral axis of the cross section. These points are the locations of the maximum bending stress.

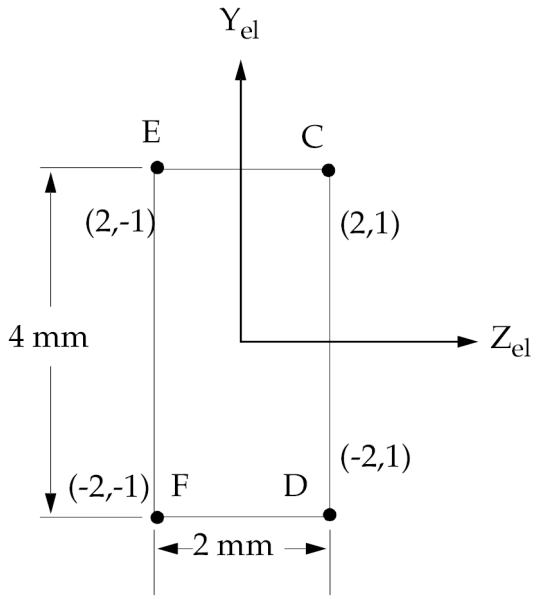


Figure 3-14. Stress Recovery Locations

In addition to the normal stress included in the stress output, there's axial stress in the CBAR elements, which is constant along the length of the bar. The SA-MAX, SB-MAX, SA-MIN, and SB-MIN stresses are the maximum and minimum combined bending and axial stresses for each end. There's no torsional stress recovery for the CBAR element. The last column of the stress output is the margin-of-safety calculation based on the tension and compression stress limits entered on the material entry. Since the stress limit fields are left blank, the software doesn't compute the margins-of-safety for this example.

Remember that the margin-of-safety computation doesn't include the torsional stress. If the torsional stress is important in your stress analysis, use the torsional force output to compute the stress outside of NX Nastran. The torsional stress is highly dependent on the geometry of the CBAR's cross section, which NX Nastran doesn't know. For this example, the cross-sectional properties (A, I1, I2, J) of each member are input, but NX Nastran doesn't know that the cross-section is rectangular. To compute the torsional stress, a formula for a rectangular cross section should be used.

Using PBAR to Define Bar Element Properties

The PBAR entry defines the properties of a CBAR element. The format of the PBAR entry is as follows:

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						

Field	Contents
PID	Property identification number from field 3 of the CBAR entry. (Integer > 0)

Field	Contents
MID	Material identification number. (Integer > 0)
A	Area of bar cross section. (Real)
I1, IA I12	Area moments of inertia. (Real; Il ≥ 0.0, I2 ≥ 0.0, I1 · I2 > $I12^2$)
J	Torsional constant. (Real)
NSM	Nonstructural mass per unit length. (Real)
K1, K2	Area factor for shear. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)

You can omit any of the stiffnesses by leaving the appropriate fields on the PBAR entry blank. For example, if you leave fields 5 and 6, the element lacks bending stiffness.

I1 and I2 (fields 5 and 6) are area moments of inertia:

I1 = area moment of inertia for bending in plane 1 (same as Izz, bending about the z element axis)
I2 = area moment of inertia for bending in plane 2 (same as Iyy, bending about the y element axis)

K1 and K2 (fields 2 and 3 on the continuation entry) depend on the shape of the cross section. K1 contributes to the shear resisting transverse force in plane 1 and K2 contributes to the shear resisting transverse force in plane 2.

Table 3-1. Area Factors for Shear

Shape of Cross Section	Value of K
Rectangular	K1 = K2 = 5/6
Solid Circular	K1 = K2 = 9/10
Thin-wall Hollow Circular	K1 = K2 = 1/2
Wide Flange Beams:	
Minor Axis	≈ A_f / 1.2*A*
Major Axis	≈ A_w / 1.2*A*

where:

A = Beam cross-sectional area
A_f = Area of flange
A_f = Area of web

Note: Using the BAROR Bulk Data entry avoids unnecessary repetition of input when a large number of bar elements either have the same property identification number or have their reference axes oriented in the same manner. BAROR defines default values on the CBAR entry for the property identification number and the orientation vector for the reference axes. The software only uses default values when you leave the corresponding fields on the CBAR entry blank.

See Also

- PBEAM in the *NX Nastran Quick Reference Guide*
- BAROR in the *NX Nastran Quick Reference Guide*

Using PBARL to Define Beam Cross Section Properties

The PBAR entry requires you to calculate the cross-sectional properties of the beam (such as area, moments of inertia, shear center, etc.). Although this is not a particularly difficult task for standard cross sections, it is tedious and can cause unnecessary input errors.

The PBARL entry, in contrast, lets you input a number of common cross-section types such as bar, box, I-beam, channel, and angle sections by their dimensions instead of by their section properties. For example, you can define a rectangular cross section by its height and depth rather than the area or moments of inertia.

Two options are available for calculating the cross-sectional properties. By default, NX Nastran uses hard-coded formulas to calculate the cross-sectional properties. For some cross section types, the formulas are based on thin wall assumptions and may produce inaccurate results if the cross section contains thick walls.

To avoid this problem, you can optionally specify that the cross-sectional properties be calculated using the mesh-based Pilkey method. When you specify this approach, the software uses the dimensions specified on the PBARL entry to create a 2D mesh of the cross section. From the 2D mesh, the Pilkey method computes the cross-sectional properties.

To specify the Pilkey method, include PARAM,PBRPROP,YES in the input file. The Pilkey method is applicable to all PBARL cross section types except rod and tube. For rod and tube, the software always uses the well-known exact formulas to calculate the cross-sectional properties.

One additional difference between the PBARL and the PBAR entries is that you don't need to specify stress recovery points to obtain stress output for the PBARL entry. The stress recovery points are automatically calculated at specific locations to give the maximum stress for the cross section.

The PBARL entry lets you input the following cross section types along with their characteristic dimensions:

- ROD, TUBE, I, CHAN (channel)
- T, BOX, BAR (rectangle)
- CROSS, H, T1, I1, CHAN1, Z, CHAN2, T2, BOX1, HEXA (hexagon)
- HAT (hat section)
- HAT1

For some shapes (I, CHAN, T, and BOX), you can also select different orientations.

Note: You can also add your own library of beam cross sections to NX Nastran.

See Also

- Adding Your Own Beam Cross Section Library

The example shown in Listing 3-4 is the same one used in Listing 3-2, except the PBAR entry is replaced by the PBARL entry. A condensed version of the corresponding output is shown in Figure 3-15. A slight difference in the output can be attributed to the fact that only four significant digits are provided in the PBAR example shown in Listing 3-2. The order of the stress data recovery points is also different in the two examples.

```
$ FILENAME - BAR1N.DAT
ID LINEAR,BAR1N
SOL 101
TIME 2
CEND
TITLE = THREE-BAR FRAME MODEL
SUBTITLE = USING BAR DIMENSION FOR PROPERTY DEFINITION
LABEL = POINT LOAD AT GRID POINT 4
LOAD = 1
DISPLACEMENT = ALL
STRESS = ALL
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID    1               -433.   250.    0.              123456
GRID    2               433.    250.    0.              123456
GRID    3               0.      -500.   0.              123456
GRID    4               0.      0.      1000.
$
CBAR    1       1       1       4       43.3    -25.    0.
CBAR    2       1       2       4       -43.3   -25.    0.
CBAR    3       1       3       4       0.      1.      0.
$
$  DIMENSIONS FOR RECTANGULAR SECTION
$
PBARL   1       1               BAR
        2.      4.
$
$ MATERIAL PROPERTIES
$
MAT1    1       19.9E4          .3
$
$ POINT LOAD
$
FORCE   1       4               5000.   0.      -1.     0.
$
ENDDATA
```

Listing 3-4. CBAR Element Defined by Cross-Sectional Dimension

```
                                    D I S P L A C E M E N T   V E C T O R

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G       .0             .0             .0             .0             .0             .0
            2      G       .0             .0             .0             .0             .0             .0
            3      G       .0             .0             .0             .0             .0             .0
            4      G       .0          -1.170445E+01  -2.574947E-05   1.447124E-02    .0             .0

                                S T R E S S E S   I N   B A R   E L E M E N T S          ( C B A R )
  ELEMENT        SA1            SA2            SA3            SA4           AXIAL          SA-MAX         SA-MIN     M.S.-T
    ID.          SB1            SB2            SB3            SB4           STRESS         SB-MAX         SB-MIN     M.S.-C
0       1    8.434553E-01   1.054030E+01  -8.434553E-01  -1.054030E+01   4.658372E+02   4.763775E+02   4.552969E+02
            -2.004751E+00  -1.398322E+00   2.004751E+00   1.398322E+00                  4.678419E+02   4.638324E+02
0       2   -1.054030E+01  -8.434553E-01   1.054030E+01   8.434553E-01   4.658372E+02   4.763775E+02   4.552969E+02
             1.398322E+00   2.004751E+00  -1.398322E+00  -2.004751E+00                  4.678419E+02   4.638324E+02
0       3    9.696349E+00  -9.696349E+00  -9.696349E+00   9.696349E+00  -9.316785E+02  -9.219821E+02  -9.413748E+02
             6.066558E-01  -6.066558E-01  -6.066558E-01   6.066558E-01                 -9.310718E+02  -9.322851E+02
```

Figure 3-15. Displacement and Stress Results for Bar Structure Using PBARL

The CBAR element assumes that the neutral axis and shear center coincide. For a non-symmetric section, the actual shear center doesn't coincide with the neutral axis. If this difference is significant, you should use the CBEAM element instead, otherwise, your results may be incorrect.

3.3 CBEAM Element

In NX Nastran, you define a beam element with a CBEAM entry and its properties with a PBEAM, PBCOMP, or PBEAML entry. The beam element includes extension, torsion, bending in two perpendicular planes, and the associated shear. The CBEAM element provides all of the capabilities of the CBAR element, plus the following additional capabilities:

- You can define different cross-sectional properties at both ends and at up to nine intermediate locations along the length of the beam.
- The neutral axis and shear center don't need to coincide, which is important for unsymmetrical sections.
- The effect of cross-sectional warping on torsional stiffness is included (PBEAM only).
- The effect of taper on transverse shear stiffness (shear relief) is included (PBEAM only).
- The CBEAM lets you apply either concentrated or distributed loads along the beam, using the PLOAD1 entry.
- You may include a separate axis for the center of nonstructural mass.
- Distributed torsional mass moment of inertia is included for dynamic analysis.
- The CBEAM lets you model a beam made up of offset rods, using the PBCOMP entry.
- CBEAMs support nonlinear material properties: elastic perfectly plastic only (see TYPE = PLASTIC on MATS1 entry).
- You can have separate shear center, neutral axis, and nonstructural mass center of gravity.
- Arbitrary variation of the section properties (A, I1, 12, I12, J) and of the nonstructural mass (NSM) along the beam (PBEAM only).

CBEAM Element Format

The format, as shown below for the CBEAM entry, is similar to that of the CBAR entry. The only difference is the addition of the SA and SB fields located in fields 2 and 3 of the second continuation entry. The SA and SB fields are scalar point entries ID used for warping terms.

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							

Field	Contents
EID	Unique element identification number.
PID	Property identification number of PBEAM or PBCOMP entry.
GA, GB	Grid point identification numbers of connection points.
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA.
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. The vector $\vec{v}$ is then transferred to end A.
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 the following figure). 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, W3AW2A, 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.
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 $d\theta/dx$.

CBEAM Coordinate System

The coordinate system for the CBEAM element, shown in Figure 3-16, is similar to that of the CBAR element. The only difference is that the element x-axis for the CBEAM element is along the shear center of the CBEAM. The neutral axis and the nonstructural mass axis may be offset from the elemental x-axis. (For the CBAR element, all three are coincident with the x-axis.) The vector $\vec{v}$ is defined in the same manner as it is for the CBAR element.

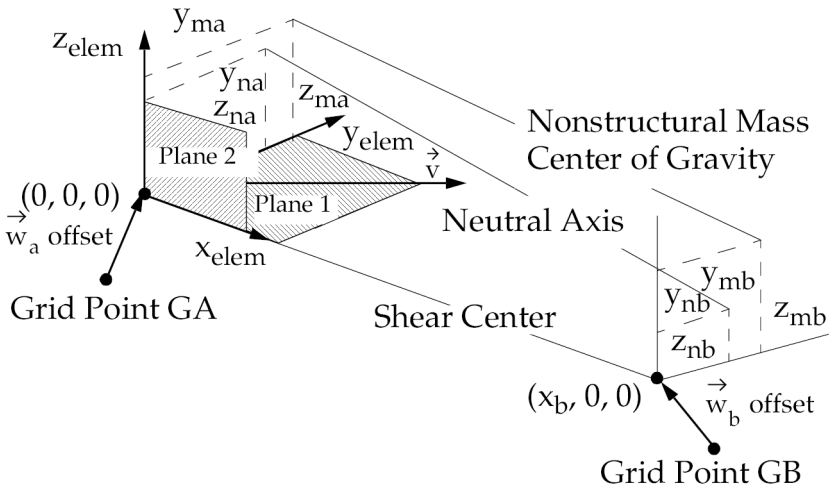


Figure 3-16. CBEAM Element Geometry System

The CBEAM element has a number of different optional fields that you can use. However, unlike the CBAR element, you must always enter positive values for fields A, I1, and I2 with the CBEAM element. The software uses that input data to generate an element flexibility matrix. It then inverts that matrix to produce the element stiffness matrix. If you leave A, I1 or I2 blank, the software issues a fatal message.

CBEAM Orientation

The orientation of the beam element is described in terms of two reference planes. The reference planes are defined with the aid of vector $\vec{v}$. This vector may be defined directly with three components in the global system at end A of the beam, or by a line drawn from end A parallel to the line from GA to a third referenced grid point. The first reference plane (Plane 1) is defined by the *x* -axis and the vector $\vec{v}$. The second reference plane (Plane 2) is defined by the vector cross product $\vec{x} \times \vec{v}$ and the *x* -axis. The subscripts 1 and 2 refer to forces and geometric properties associated with bending in Planes 1 and 2, respectively. The reference planes are not necessarily principal planes. The coincidence of the reference planes and the principal planes is indicated by a zero product of inertia (I_{12}) on the PBEAM entry. When pin flags and offsets are used, the effect of the pin is to free the force at the end of the element *x* -axis of the beam, not at the grid point.

Defining CBEAM End Offsets

End A is offset from grid point GA an amount measured by vector $\vec{w}_a$, and end B is offset from grid point GB an amount measured by vector $\vec{w}_b$. The vectors $\vec{w}_a$ and $\vec{w}_b$ are measured in the global coordinates of the connected grid point. The *x* -axis of the element coordinate system is defined by a line connecting the shear center of end A to that at end B of the beam element.

Using Pin Flags with CBEAMs

Any five of the six forces at either end of the element may be set equal to zero by using the pin flags on the CBEAM entry. The integers 1 to 6 represent the axial force, shearing force in Plane 1, shearing force in Plane 2, axial torque, moment in Plane 2 and moment in Plane 1, respectively. The structural

and nonstructural mass of the beam are lumped at the ends of the elements, unless you request coupled mas with the PARAM,COUPMASS option.

See Also

- COUPMASS in the *NX Nastran Quick Reference Guide*

CBEAM Force and Moment Conventions

The positive directions for element forces are shown in Figure 3-17. The following element forces, either real or complex (depending on the solution sequence), are output on request at both ends and at intermediate locations defined on the PBEAM entry:

- Beam element internal forces and moments
- Bending moments in the two reference planes at the neutral axis.
- Shear forces in the two reference planes at the shear center.
- Axial force at the neutral axis.
- Total torque about the beam shear center axis.
- Component of torque due to warping.

The following real element stress data are output on request:

- Real longitudinal stress at the four points prescribed for each cross section defined along the length of the beam on the PBEAM entry.
- Maximum and minimum longitudinal stresses.
- Margins of safety in tension and compression for the element if you enter stress limits on the MAT1 entry.

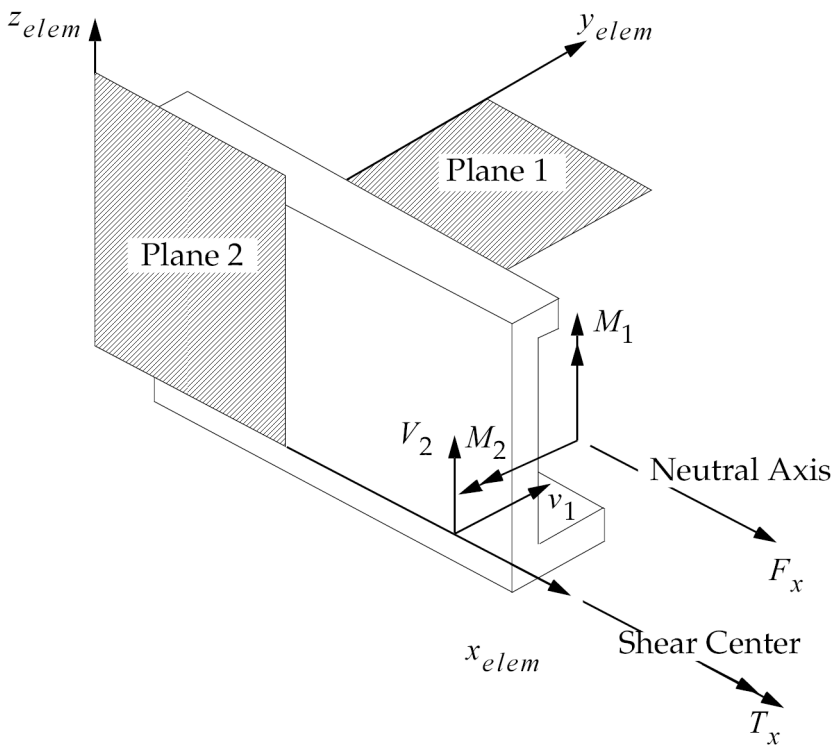


Figure 3-17. CBEAM Element Internal Forces and Moments

Tensile stresses are given a positive sign and compressive stresses a negative sign. Only the longitudinal stresses are available as complex stresses. The stress recovery coefficients on the PBEAM entry are used to locate points on the cross section for stress recovery. The subscript 1 is associated with the distance of a stress recovery point from Plane 2. The subscript 2 is associated with the distance from Plane 1. Note that if zero value stress recovery coefficients are used, the axial stress is output.

Since there's no torsional stress recovery for the CBEAM element, the margin-of-safety computation does not include the torsional stress. If the torsional stress is important in your stress analysis, use the torsional force output to compute the stress outside of NX Nastran. The torsional stress is highly dependent on the geometry of the CBAR's cross section, which NX Nastran doesn't know.

CBEAM Example 1 (Simple Truss)

For example, consider a simple truss model. To convert a CBAR model to a CBEAM model, only three changes are needed. Change the CBAR name to CBEAM, change the PBAR name to a PBEAM, and change the location of J from field 7 of the PBAR entry to field 8 of the PBEAM entry.

One difference between the CBAR element and the CBEAM element that isn't obvious is the default values used for the transverse shear flexibility. For the CBAR element, the default values for K1 and K2 are infinite, which is equivalent to zero transverse shear flexibility. For the CBEAM element, the default values for K1 and K2 are both 1.0, which includes the effect of transverse shear in the elements. If you want to set the transverse shear flexibility to zero, which is the same as the CBAR element, use a value of 0.0 for K1 and K2.

The resulting stress output is shown in Figure 3-18.

S T R E S S E S I N B E A M E L E M E N T S (C B E A M)

ELEMENT-ID	GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
1										
	1	0.000	4.666807E+02	4.763772E+02	4.552972E+02	4.649937E+02	4.763772E+02	4.552972E+02		
	4	1.000	4.638323E+02	4.644395E+02	4.672348E+02	4.678420E+02	4.678420E+02	4.638323E+02		
2										
	2	0.000	4.552972E+02	4.649937E+02	4.666807E+02	4.763772E+02	4.763772E+02	4.552972E+02		
	4	1.000	4.672348E+02	4.678420E+02	4.638323E+02	4.644395E+02	4.678420E+02	4.638323E+02		
3										
	3	0.000	-9.219825E+02	-9.413745E+02	-9.219825E+02	-9.413745E+02	-9.219825E+02	-9.413745E+02		
	4	1.000	-9.310710E+02	-9.322858E+02	-9.310710E+02	-9.322858E+02	-9.310710E+02	-9.322858E+02		

Figure 3-18. CBEAM Stress Output

Stress recovery is performed at the end points, and at any intermediate location you define on the PBEAM entry. In Figure 3-18, results are presented for the end points of elements 1, 2, and 3. The results are presented in tabular fashion with the column headings defined as follows:

- The SXC, SXD, SXE, and SXF columns list the superposed stress resulting from both bending and axial loading at locations C, D, E, and F of the cross section, respectively.
- The S-MAX and S-MIN columns list the maximum and minimum stresses, respectively.
- The M.S.-T and M.S.-C columns list the margins of safety in tension and compression, respectively. (Margins of safety are only calculated if the you enter stress limits on the MAT1 entry.)

The stress output for CBEAM elements differs from that of CBAR elements. For CBAR elements, the SAi and SBi columns list the stresses due to bending only, and the axial stresses are listed in a separate column.

CBEAM Example 2 (Tapered Beam)

As another example for the beam element that uses more of the CBEAM features, consider the tapered beam shown in Figure 3-19.

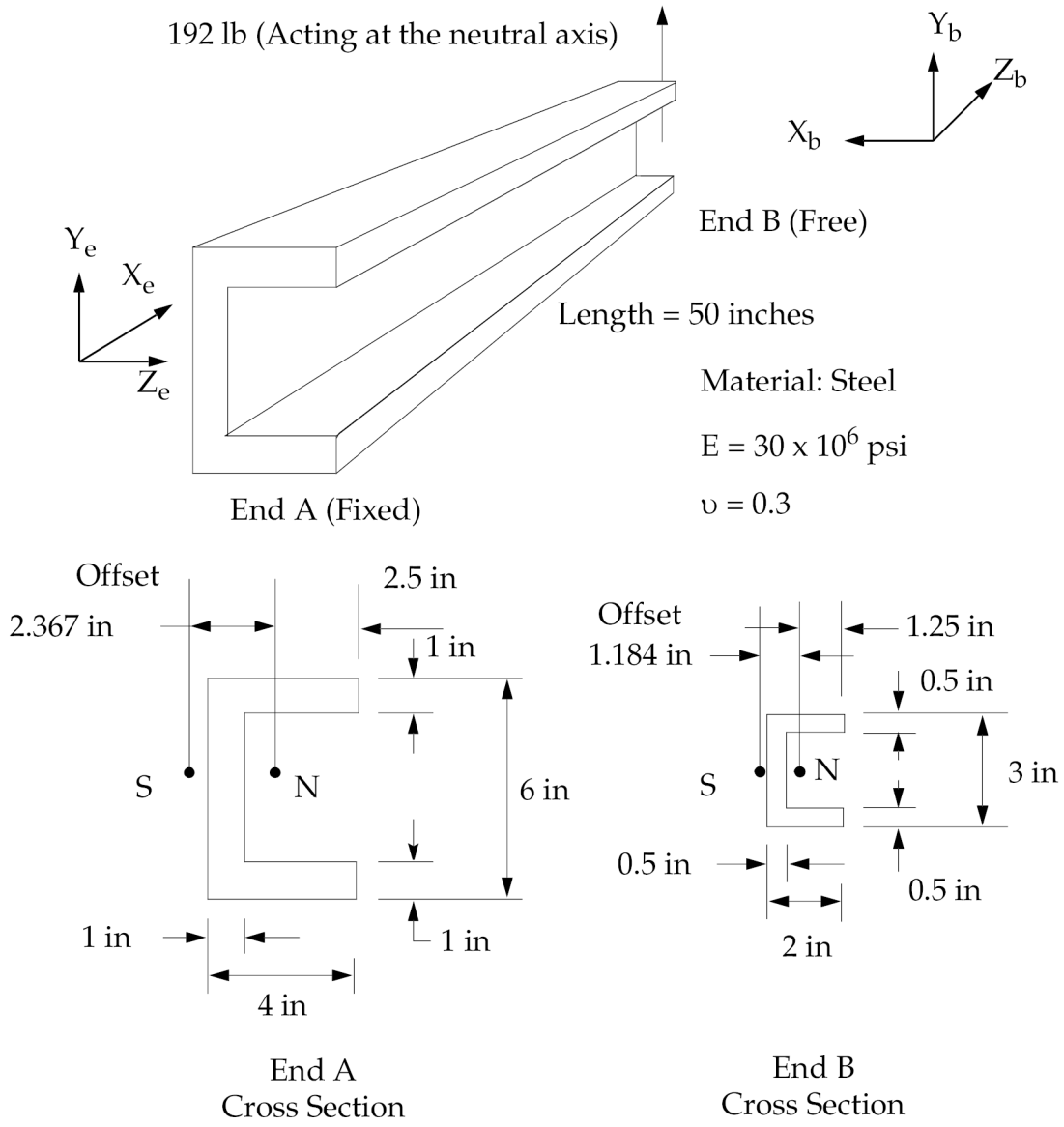


Figure 3-19. Tapered Beam Example

Because the cross section is an open channel section, the shear center of the beam doesn't coincide with the neutral axis. In general, you must decide in your model planning whether having noncoincident shear and neutral axes is significant for your analysis. In this example, the entire structure is a single tapered beam, so modeling the offset is important. There are two methods that can be used to model the offset.

- The first method is to place the shear axis on the line between the end grid points 1 and 2. In this case the neutral axis is offset from the shear axis using the y_{na} , z_{na} , y_{nb} , and z_{nb} offsets that you enter on the PBEAM entry.

- The second method to model the noncoincident axes in this example is to place the neutral axis on the line extending from grid point 1 to grid point 2. In this case, the offsets WA and WB, entered on the CBEAM entry, are used to position the shear axis at the appropriate location.

The y_{na} , z_{na} , y_{nb} , and z_{nb} offsets entered on the PBEAM entry are then used to position the neutral axis to the appropriate position.

Both methods represent the same CBEAM element but are positioned differently with respect to the grid points. The first method requires one set of offset values to be entered, and the second method requires two sets of offset values. The forces are applied to the grid points. By using the second method for this problem, the loads are then applied at the neutral axis instead of the shear center axis. By doing so, you can observe the twisting of the beam due to a pure vertical load.

The Bulk Data is shown in Listing 3-5. You enter the offsets WA = (2.367,0.0,0.0) and WB = (1.184,0.0,0.0) on the CBEAM entry to define the locations of the shear axis. The neutral axis is offset from the shear axis using the offsets y_{na} = 0.0, z_{na} = 2.367, y_{nb} = 0.0, and z_{nb} = 1.184 entered on the PBEAM entry. It may appear that the offsets do not accomplish the desired goal of placing the neutral axis in the right location because all of the offset are positive. However, keep in mind that the shear center offsets (WA and WB) are in the displacements coordinate system, measured from GA and GB, respectively. The neutral axis offsets are in the CBEAM's element coordinate system.

Nine intermediate stations are used to model the taper. Since the properties aren't a linear function of the distance along the beam (A is, but I1 and I2 are not), it is necessary to compute the cross-sectional properties for each of the stations. The properties for the nine stations are entered on the PBEAM entry.

To demonstrate all of the capabilities of the CBEAM element, beam warping is included; however, beam warping isn't significant for this problem.

You should note the locations of the stress recovery locations on the PBEAM entry. The stress recovery locations are entered with respect to the shear axis, not the neutral axis (i.e., they are input with respect to the element coordinate system). NX Nastran computes the distance from the neutral axis internally for the stress recovery.

```
$ FILENAME - BEAM2.DAT
ID     LINEAR,BEAM2
SOL    101
TIME   5
CEND
TITLE = TAPERED BEAM MODEL
DISP = ALL
STRESS = ALL
FORCE = ALL
LOAD = 1
SPC = 1
BEGIN BULK
PARAM   POST    0
PARAM   AUTOSPC YES
$
GRID    1               0.0     0.0     0.0
GRID    2               0.0     0.0     50.0
SPOINT  101     102
SPC     1       1       123456  0.0
CBEAM   1       11      1       2       0.      1.      0.
                        2.367   0.      0.      1.184   0.      0.
        101     102
$
$       2       3       4       5       6       7       8       9
$
PBEAM         11      21  12.000  56.000  17.000          3.930
          -3.000    .867  -3.000   4.867   3.000   4.867   3.000    .867
        YES         .100  10.830  45.612  13.847          3.201
          -2.850    .824  -2.850   4.624   2.850   4.624   2.850    .824
```

```
        YES         .200   9.720  36.742  11.154           2.579
          -2.700    .780  -2.700   4.380   2.700   4.380   2.700    .780
        YES         .300   8.670  29.232   8.874           2.052
          -2.550    .737  -2.550   4.137   2.550   4.137   2.550    .737
        YES         .400   7.680  22.938   6.963           1.610
          -2.400    .694  -2.400   3.894   2.400   3.894   2.400    .694
        YES         .500   6.750  17.719   5.379           1.244
          -2.250    .650  -2.250   3.650   2.250   3.650   2.250    .650
        YES         .600   5.880  13.446   4.082            .944
          -2.100    .607  -2.100   3.407   2.100   3.407   2.100    .607
        YES         .700   5.070   9.996   3.035            .702
          -1.950    .564  -1.950   3.164   1.950   3.164   1.950    .564
        YES         .800   4.320   7.258   2.203            .509
          -1.800    .520  -1.800   2.920   1.800   2.920   1.800    .520
        YES         .900   3.630   5.124   1.556            .360
          -1.650    .477  -1.650   2.677   1.650   2.677   1.650    .477
        YES        1.000   3.000   3.500   1.062            .246
          -1.500    .434  -1.500   2.434   1.500   2.434   1.500    .434
            .241           -.666     0.            70.43   1.10
                                                0.   2.367     0.   1.184
$
MAT1    21      3.+7            .3
$
FORCE   1       2               192.    0.      1.      0.
ENDDATA
```

Listing 3-5. Tapered Beam Input File

The displacement results, as shown in Figure 3-20, include the displacements of the grid points at the end of the CBEAM and scalar points 101 and 102. As mentioned previously, the force is applied directly to the grid point; therefore, the force acts at the neutral axis of the beam. Since the shear center is offset from the neutral axis, a loading of this type should cause the element to twist. This result can be observed in the R3 displacement, which represents the twist of the beam. If the shear center is not offset from the neutral axis, R3 will be zero. The displacements of scalar points 101 and 102 represent the twist due to the warping at ends A and B, respectively.

The forces in the beam are shown in Figure 3-21, along with the total torque and the warping torque acting along the beam. The warping for this case is negligible. Note that the inclusion of warping does not affect any of the other forces in the CBEAM element. The stress recovery output is shown in Figure 3-22. The stress output shows only the longitudinal stress; hence, any stress due to torsional or warping is not included.

```
D I S P L A C E M E N T   V E C T O R

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
             1      G       .0             .0             .0             .0             .0             .0
             2      G       .0            1.181932E-02    .0            -3.621016E-04   .0            -1.170325E-03
           101      S      2.322214E-05   2.322214E-05
```

Figure 3-20. Displacement Output for the Tapered Beam

F O R C E S I N B E A M E L E M E N T S (C B E A M)

ELEMENT-ID	GRID	STAT DIST/ LENGTH	- BENDING MOMENTS - PLANE 1	PLANE 2	- WEB SHEARS - PLANE 1	PLANE 2	AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
0 1									
	1	.000	9.608063E+03	.0	3.199436E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.100	8.647795E+03	.0	3.071564E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.200	7.687526E+03	.0	2.943692E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.300	6.727258E+03	.0	2.815820E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.400	5.766989E+03	.0	2.687948E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.500	4.806720E+03	.0	2.560076E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.600	3.846451E+03	.0	2.432204E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.700	2.886183E+03	.0	2.304332E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.800	1.925914E+03	.0	2.176460E+02	.0	.0	-2.272644E+02	-3.206116E-16
	0	.900	9.656460E+02	.0	2.048588E+02	.0	.0	-2.272644E+02	-3.206116E-16
	2	1.000	5.377076E+00	.0	1.920716E+02	.0	.0	-2.272644E+02	-3.206116E-16

Figure 3-21. Force Output for the Tapered Beam

S T R E S S E S I N B E A M E L E M E N T S (C B E A M)

ELEMENT-ID	GRID	STAT DIST/ LENGTH	SXC	SXD	SXE	SXF	S-MAX	S-MIN	M.S.-T	M.S.-C
1										
	1	.000	5.147177E+02	5.147177E+02	-5.147177E+02	-5.147177E+02	5.147177E+02	-5.147177E+02		
	0	.100	5.403450E+02	5.403450E+02	-5.403450E+02	-5.403450E+02	5.403450E+02	-5.403450E+02		
	0	.200	5.649208E+02	5.649208E+02	-5.649208E+02	-5.649208E+02	5.649208E+02	-5.649208E+02		
	0	.300	5.868400E+02	5.868400E+02	-5.868400E+02	-5.868400E+02	5.868400E+02	-5.868400E+02		
	0	.400	6.033994E+02	6.033994E+02	-6.033994E+02	-6.033994E+02	6.033994E+02	-6.033994E+02		
	0	.500	6.103686E+02	6.103686E+02	-6.103686E+02	-6.103686E+02	6.103686E+02	-6.103686E+02		
	0	.600	6.007398E+02	6.007398E+02	-6.007398E+02	-6.007398E+02	6.007398E+02	-6.007398E+02		
	0	.700	5.630309E+02	5.630309E+02	-5.630309E+02	-5.630309E+02	5.630309E+02	-5.630309E+02		
	0	.800	4.776310E+02	4.776310E+02	-4.776310E+02	-4.776310E+02	4.776310E+02	-4.776310E+02		
	0	.900	3.109516E+02	3.109516E+02	-3.109516E+02	-3.109516E+02	3.109516E+02	-3.109516E+02		
	2	1.000	2.304461E+00	2.304461E+00	-2.304461E+00	-2.304461E+00	2.304461E+00	-2.304461E+00		

Figure 3-22. Stress Output for the Tapered Beam

PBEAM Format

The PBEAM entry may be substantially different than the PBAR entry, depending on which features you use. The format of the PBEAM entry is as follows:

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, and SO and X/XB must be specified.

1	2	3	4	5	6	7	8	9	10
	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	DI	D2	E1	E2	F1	F2	

The last two continuations are:

1	2	3	4	5	6	7	8	9	10
	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)	

Field	Contents
PID	Property identification number.
MID	Material identification number.
A(A)	Area of the beam cross section at end A.
I1(A)	Area moment of inertia at end A for bending in Plane 1 about the neutral axis.
I2(A)	Area moment of inertia at end A for bending in Plane 2 about the neutral axis.
I12(A)	Area product of inertia at end A.
J(A)	Torsional stiffness parameter at end A.
NSM(A)	Nonstructural mass per unit length at end A.
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.
SO	Stress output request option. “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.
X/XB	Distance from end A in the element coordinate system divided by the length of the element.

Field	Contents
A, I1, I2, I12,J, NSM	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x.
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 for the cross section located at X/XB. The values are fiber locations for stress data recovery.
K1, K2	Shear stiffness factor K in K • A • G for Plane 1 and Plane 2.
S1, S2	Shear relief coefficient due to taper for Plane 1 and Plane 2.
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B.
CW(A), CW(B)	Warping coefficient for end A and end B.
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B.
N1(A), N2(A),N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B.

Element Properties on the PBEAM Entry

When you use the PBEAM entry to define a beam element's properties, you can define a number of different cross-sectional properties. The following table lists the properties you can define and the manner in which these properties are interpolated along the x-axis of the element

Quantity	Definition	Locations at WhichYou Specify Properties	Method of Interpolation
A	Cross-sectional Area	Ends, Interior points	Linear between points
I1,I2,I12	Moments and product of inertia about neutral axis for planes 1 and 2	Ends, Interior points	Linear between points
J	Torsional stiffness parameter	Ends, Interior points	Linear between points
NSM	Nonstructural mass per unit length	Ends, Interior points	Linear between points

Quantity	Definition	Locations at WhichYou Specify Properties	Method of Interpolation
K1,K2	Shear factors in (K)AG for planes 1 and 2	One value	Constant
S1,S2	Shear relief factors for planes 1 and 2	One Value	Constant
CW	Warping stiffness parameter	Ends	Linear between ends
NSI	Nonstructural mass moment of inertia per unit length	Ends	Linear between ends

Using PBEAML to Define Beam Cross Section Properties

The PBEAM entry requires you to calculate the cross-sectional properties of the beam (such as area, moments of inertia, shear center, etc.). Although this is not a particularly difficult task for standard cross sections, it is tedious and can cause unnecessary input errors.

The PBEAML entry, in contrast, lets you input a number of common cross-section types such as bar, box, I-beam, channel, and angle sections by their dimensions instead of by their section properties. For example, you can define a rectangular cross section by its height and depth rather than the area or moments of inertia.

Two options are available for calculating the cross-sectional properties. By default, NX Nastran uses hard-coded formulas to calculate the cross-sectional properties. For some cross section types, the formulas are based on thin wall assumptions and may produce inaccurate results if the cross section contains thick walls.

To avoid this problem, you can optionally specify that the cross-sectional properties be calculated using the mesh-based Pilkey method. When you specify this approach, the software uses the dimensions specified on the PBEAML entry to create a 2D mesh of the cross section. From the 2D mesh, the Pilkey method computes the cross-sectional properties.

To specify the Pilkey method, include PARAM,PBRPROP,YES in the input file. The Pilkey method is applicable to all PBEAML cross section types except rod and tube. For rod and tube, the software always uses the well-known exact formulas to calculate the cross-sectional properties.

One additional difference between the PBEAML and the PBEAM entries is that you don’t need to specify stress recovery points to obtain stress output for the PBEAML entry. The stress recovery points are automatically calculated at specific locations to give the maximum stress for the cross section.

The PBEAML entry lets you input the following cross section types along with their characteristic dimensions:

- ROD, TUBE, I, CHAN (channel)
- T, BOX, BAR (rectangle)
- CROSS, H, T1, I1, CHAN1, Z, CHAN2, T2, BOX1, HEXA (hexagon)
- HAT (hat section)
- HAT1

For some shapes (I, CHAN, T, and BOX), you can also select different orientations.

Note: You can also add your own library of beam cross sections to NX Nastran.

See Also

- Adding Your Own Beam Cross Section Library

The problem shown in Figure 3-19 is rerun using the PBEAML entry. Because the cross section geometry is a channel section, the TYPE field (field five) on the PBEAML entry is assigned the value “CHAN”. Four dimensional values are required at each station that output is desired, or cross sectional properties that cannot be interpolated linearly between the values at the two ends of the CBEAM element. For this example, since only the dimensional values for the two end points and at the middle of the CBEAM element are provided, only output at these locations is available. The complete input and partial output files are shown in Listing 3-6 and Figure 3-23, respectively.

In this case, the values 4.0, 6.0, 1.0, and 1.0 on the first continuation entry represent DIM1, DIM2, DIM3, and DIM4, respectively at end A. The values YES, 0.5, 3.0, 4.5, 0.75, and 0.75 on the first and second continuation entries represent stress output request, value of X(1)/XB, DIM1 at X(1)/XB, DIM2 at X(1)/X(B), DIM3 at X(1)/XB, and DIM4 at X(1)/X(B), respectively at X(1)/X(B) = 0.5. The values YES, 1.0, 2.0, 3.0 0.5, and 0.5 on the second and third continuation entries represent stress output request, end B, DIM1 at end B, DIM2 at end B, DIM3 at end B, and DIM4 at end B, respectively.

```
$ FILENAME - BEAM2N.DAT
ID     LINEAR,BEAM2N
SOL    101
TIME   5
CEND
TITLE = TAPERED BEAM MODEL
SUBTITLE = CROSS-SECTION DEFINED BY CHARACTERISTIC DIMENSIONS
DISP = ALL
STRESS = ALL
FORCE = ALL
LOAD = 1
SPC = 1
BEGIN BULK
PARAM   AUTOSPC YES
$
GRID    1               0.0     0.0     0.0
GRID    2               0.0     0.0     50.0
SPOINT  101     102
SPC     1       1       123456  0.0
CBEAM   1       11      1       2       0.      1.      0.
                        2.367   0.      0.      1.184   0.      0.
        101     102
$
$ 2 3 4 5 6 7 8 9
$
$
PBEAML 11 21  CHAN
 4.0 6.0 1.0 1.0  YES 0.5 3.0
 4.5 0.75 0.75  YES 1.0 2.0 3.0
 0.5 0.5
$
MAT1    21      3.+7            .3
$
FORCE   1       2               192.    0.      1.      0.
$
ENDDATA
```

Listing 3-6. CBEAM Element Defined by Cross-Sectional Dimension

```
0

                                    S T R E S S E S   I N   B E A M   E L E M E N T S        ( C B E A M )
                        STAT DIST/
   ELEMENT-ID  GRID      LENGTH     SXC           SXD           SXE           SXF           S-MAX         S-MIN         M.S.-T   M.S.-C
0         1
                   1      .000   -5.147177E+02  5.147177E+02  5.147177E+02 -5.147177E+02  5.147177E+02 -5.147177E+02
                   0      .500   -6.103772E+02  6.103772E+02  6.103772E+02 -6.103772E+02  6.103772E+02 -6.103772E+02
                   2     1.000   -2.304461E+00  2.304461E+00  2.304461E+00 -2.304461E+00  2.304461E+00 -2.304461E+00
```

Figure 3-23. Stress Output for the Tapered Beam using Cross-Sectional Dimension

As a side topic to help understand the implementation of warping, it is useful to see the actual equations being used. The basic equation for twist about the shear center of a beam is given by:

$$m_0 = \frac{d^2}{dx^2}\left(EC_w\frac{d^2\theta_x}{dx^2}\right) - \frac{d}{dx}\left(GJ\frac{d\theta_x}{dx}\right)$$

Equation 3-1.

where C_w is the warping coefficient.

The twist of the beam is defined as:

$$\phi = \frac{d\theta_x}{dx}$$

Equation 3-2.

Substituting Equation 3-1 into Equation 3-2 and transferring the applied internal torsional moments to the end of the beam, the equation for the warping stiffness is reduced to Equation 3-3.

$$T_x = \frac{d}{dx}\left(EC_w\frac{d\phi}{dx}\right) - JG\phi$$

Equation 3-3.

The scalar points defined on the CBEAM entry are used to represent the φ. T_x is the warping torque.

Returning to our taper beam model, it is interesting to see how the single beam element compares to the same member modeled as plate elements and solid elements. Table 3-2 shows the results of modeling the tapered member using a single CBEAM element, plate elements (CQUAD4), and solid elements (CHEXA).

Table 3-2. Comparison of the Beam, Plate, and Solid Element Model for the Tapered Beam

Element Type	Number of Elements	Number of DOFs	Y-Disp. at Free End x 10^{-2} in	θ_z at the Free Endx 10^{-3}	Maximum NormalStress psi
CBEAM	1	14	1.02	1.36	610
CQUAD4	960	6,174	0.99	0.67	710
CHEXA	3,840	15,435	1.08	0.97	622

The single beam element model with only 14 degrees-of-freedom compares well with the 15,435-degree-of-freedom solid model. The CQUAD4 plate model is included for completeness. Typically, you do not use plate elements for this type of structure. The flanges and web are very thick and do not behave like plates. The stress results shown in Table 3-2 reflect this situation.

The solid model, on the other hand, represents a good use of the CHEXA element. This is discussed later in the solid element section. The entire input file for this example is available in the Test Problem Library.

Using PBCOMP

You can use the PBCOMP entry to input offset rods to define the beam's section properties. A program automatically converts the data to an equivalent PBEAM entry. The input options that allow efficient descriptions of various symmetric cross sections are shown in the *NX Nastran Quick Reference Guide* .

See Also

- PBCOMP in the *NX Nastran Quick Reference Guide*

Mass Matrix

The inertia properties of the CBEAM element include the following terms:

- Structural mass per unit length, RHO · A, on the neutral axis.
- Nonstructural mass per unit length, NSM.
- Moment of inertia of structural mass per unit length RHO · (I1 + I2), about neutral axis.
- Moment of inertia of nonstructural mass, NSI.

where RHO is the density defined on the MAT1 entry.

Cross Sectional Warping

Open section members such as channels will undergo torsion as well as bending when transverse loads act anywhere except at the shear center of a cross-section. This torsion produces warping of the cross-section so that plane sections do not remain plane and, as a result, axial stresses are produced. This situation can be represented in the following differential equation for the torsion of a beam about the axis of shear centers:

$$G\frac{d}{dx}\left(J\frac{d\theta}{dx}\right) - E\frac{d^2}{dx^2}\left(C_w\frac{d^2\theta}{dx^2}\right) = m$$

Equation 3-4.

E = Young's modulus of elasticity
C_w = Warping constant
G = Shear modulus
J = Torsion constant

θ = Angle of rotation at any cross-section
m = Applied torsional moment per unit length

Note that C_w, the warping constant, has units of (length)^6. The development of the above differential equation and methods for the numerical evaluations of the warping constant are available in the literature. (See, for example, Timoshenko and Gere, *Theory of Elastic Stability*, McGraw Hill Book Company, 1961.) An example that demonstrates the use of the warping constant is the section titled “Example Problem of Channel Section.”

Shear Relief

The shear relief factor accounts for the fact that in a tapered flanged beam the flanges sustain a portion of the transverse shear load. This situation is illustrated in Figure 3-24:

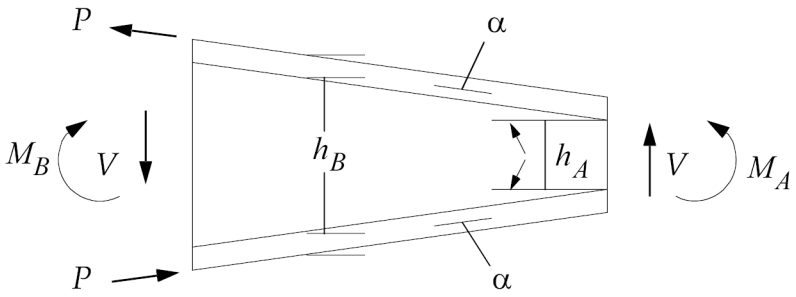


Figure 3-24. The Shear Relief Factor

Here, the net transverse shear, Q, at the cross-section of depth h_B is

$$Q = V - 2P\sin\alpha$$

Equation 3-5.

and, if the entire bending moment is carried by the flanges,

$$Q = V\frac{2\tan\alpha}{h_r}M_B$$

Equation 3-6.

Eq. 3-6 is represented in NX Nastran as:

$$Q_1 = V_1 - \frac{S_1}{l}M_1$$

$$Q_2 = V_2 - \frac{S_2}{l}M_2$$

Equation 3-7.

where *l* is the length of the CBEAM element in question and the subscripts refer to plane 1 and plane 2, respectively. The terms S_1 and S_2 are denoted as the shear relief coefficients. The value of the shear coefficient for a tapered beam with heavy flanges that sustain the entire moment load may then be written as:

$$S_1 = \frac{2(h_A - h_B)}{(h_A + h_B)}$$

where:

h_A = depth at end A of CBEAM element (GA on CBEAM entry)
h_B = depth at end B of CBEAM element (GB on CBEAM entry)

Example Problem of Channel Section

The simply supported beam illustrated in Figure 3-25 is modeled with five CBEAM elements. Since the beam is an open section channel with a single plane of symmetry, buckling failure can occur either through a combination of torsion and bending about the element x-axis or the lateral bending about the y-axis. The effect of cross-sectional warping coefficients, CW(A) and CW(B), on the PBEAM entry are necessary to capture these effects. Since the column is of uniform cross-section only the PBEAM entry illustrated below is required.

1	2	3	4	5	6	7	8	9	10
PBEAM	1	1	.986	1.578465	.1720965		.0094985		
	NO	1.							
							.3010320	.3010320	
						.7659450		.769450	

Warping requires 7 DOF for the beam element. Thus, for warping, each grid must have an associated scalar point.

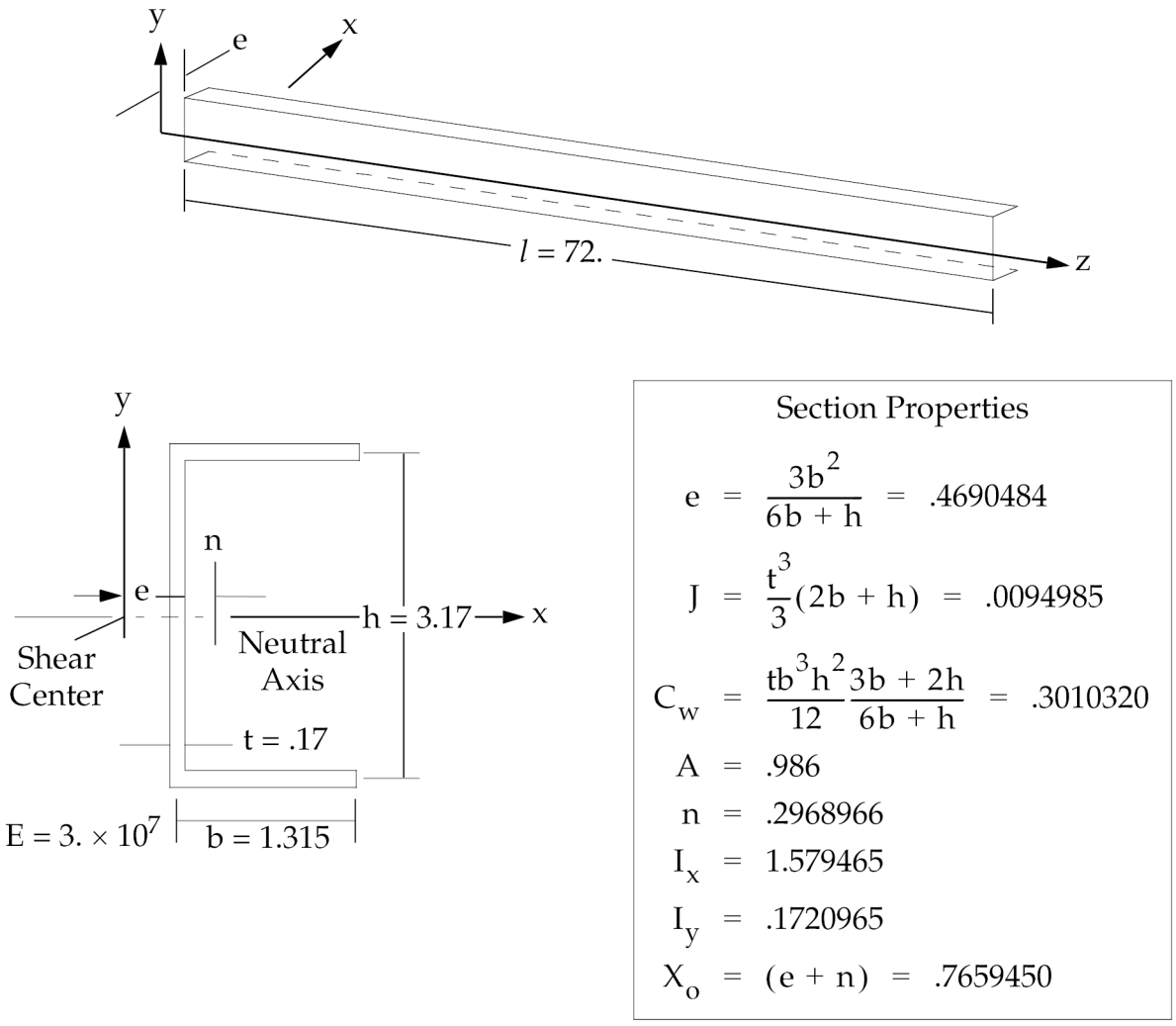


Figure 3-25. Simply-Supported Channel

Example of Tapered Beam

The cantilevered tapered flanged beam illustrated in Figure 3-26 is utilized to demonstrate the use of shear relief factors, S1 and S2, with the CBEAM element. As noted earlier in the section, shear relief provides for the fact that in a tapered flanged beam some of the transverse shear load is carried by the flanges. All of the bending load is carried by the flanges. The shear relief coefficient for the tip cell is computed by the following formula:

$$S_1 = \frac{2(h_A - h_B)}{(h_A + h_B)} = \frac{2(10. - 20.)}{(10. + 20.)} = -.66667$$

The PBEAM entry for this tip cell is illustrated below:

1	2	3	4	5	6	7	8	9	10
PBEAM	1	1	1.	60.	1.				
	5.		-5.						

1	2	3	4	5	6	7	8	9	10
	YES	1.	2.	240.					
	10.		-10.						
			-.666667						

Note

A unique PBEAM entry is required for each CBEAM element as the shear relief factor among other properties vary from element to element. Also note that the default values of 1. are accepted for the shear stiffness factors.

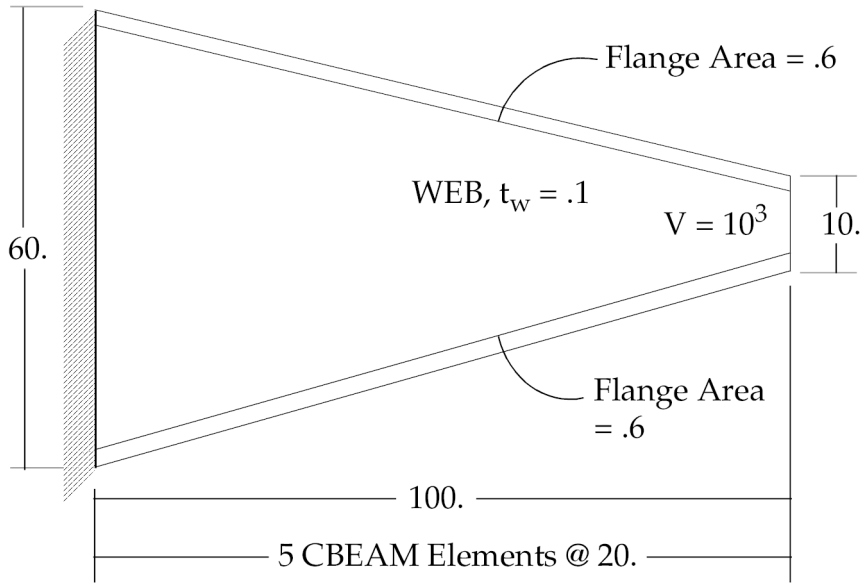


Figure 3-26. Tapered Cantilevered Beam With Shear Relief

3.4 CBEND Element

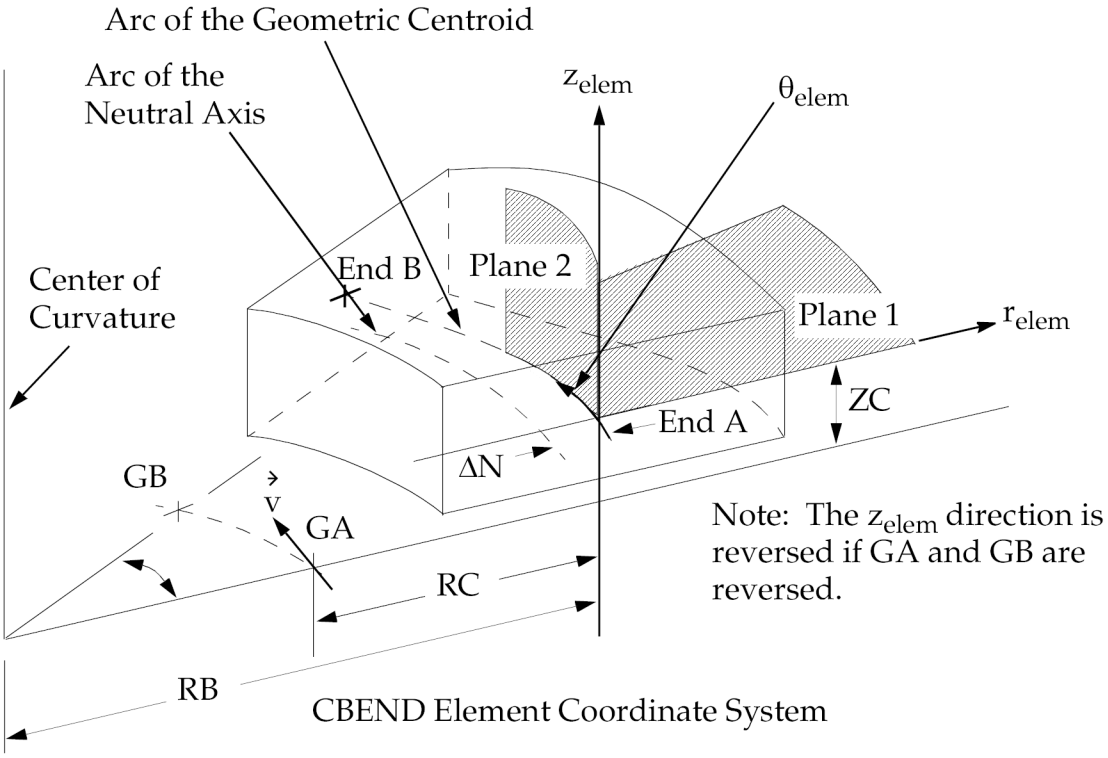
In NX Nastran, you can define a bend element with a CBEND entry and its properties are defined with a PBEND entry. The CBEND element is a one-dimensional bending element with a constant radius of curvature (circular arc) that connects two grid points. This element has extensional and torsional stiffness, bending stiffness, and transverse shear flexibility in two perpendicular directions. You can omit the transverse shear flexibility by leaving the appropriate fields blank on the PBEND entry.

You can use the CBEND element to analyze either curved beams, pipe elbows or miter bends. For example, you can use CBEND to model pressurized pipe systems and curved components that behave as one-dimensional members.

NX Nastran assumes that the CBEND element's cross section properties are constant along the length of the element and the principal axes coincide with the element coordinates shown in Figure 3-28. Structural and nonstructural mass for the bend element is obtained only as consistent mass. The lumped mass option isn't available.

Specific features of the CBEND element are as follows:

- Principal bending axes must be parallel and perpendicular to the plane of the element (see Figure 3-27).
- The geometric center of the element may be offset in two directions (see Figure 3-27).
- The offset of the neutral axis from the centroidal center due to curvature is calculated automatically with a user-override (DN) available for the curved beam form of the element.
- Four methods are available to define the plane of the element and its curvature.
- Six methods are available in the curved pipe form to account for the effect of curvature on bending stiffness and stress.
- The effect of internal pressure on stiffness and stress can be accounted for using four of the six methods mentioned in the previous item.
- Axial stresses can be output at four cross-sectional points at each end of the element. Forces and moments are output at both ends.
- Distributed loads may be placed along the length of the element by means of the PLOAD1 entry.



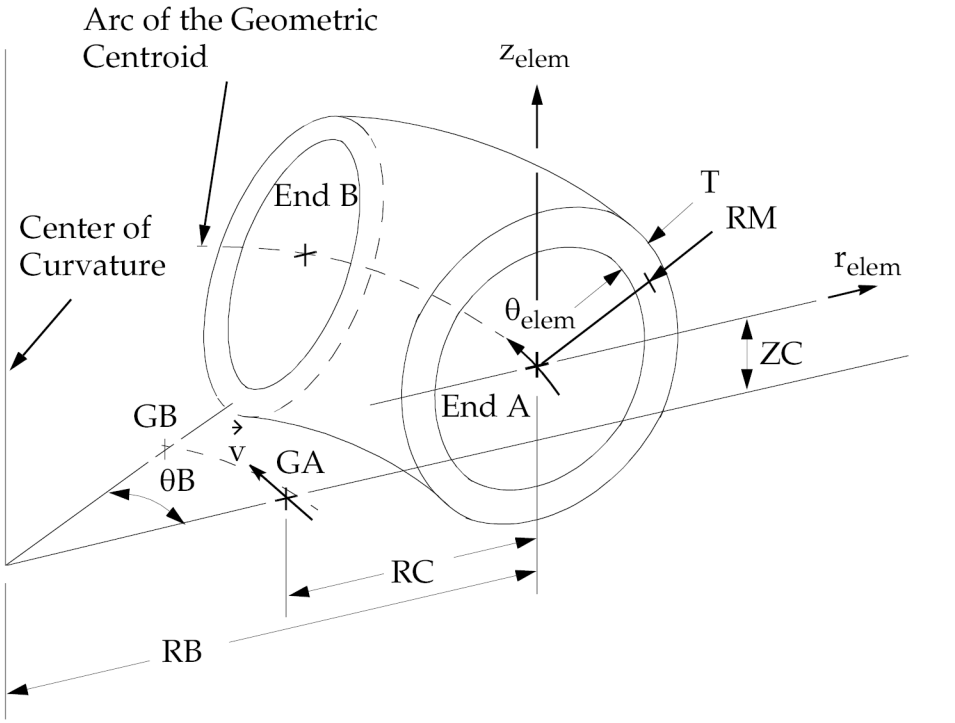


Figure 3-27. The CBEND Element

CBEND Format

The format of the CBEND entry is as follows:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

Field	Contents
EID	Unique element identification number.
PID	Property identification number of a PBEND entry.
GA, GB	Grid point identification numbers of connection points.
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA.
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. The vector $\vec{v}$ is then translated to End A.
GEOM	Flag to select specification of the bend element.

PBEND Format

The PBEND entry has two alternate forms.

- The first form corresponds to a curved beam of an arbitrary cross section.
- The second form is used to model pipe elbows and miter bends.

Like the CBEAM element, with the CBEND element, you must enter positive values for A, I1, and I2. You can omit the transverse shear flexibility by leaving the appropriate fields blank on the PBEND entry. The format of the PBEND entry is as follows:

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			

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			

Field	Contents
PID	Property identification number. (Integer > 0)

Field	Contents
MID	Material identification number. (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. (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. (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. (Real)
ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. (Real)
DELTAN	Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature.
SACL	Miter spacing at center line. See Figure 3-29 (Real > 0.0)
ALPHA	One-half angle between the adjacent miter axes (Degrees). Required for FSI=5 with miter bend. See Figure 3-29.
FLANGE	For FSI=5, defines the number of flanges attached. (Integer; Default=0)
KX	For FSI=6, the user defined flexibility factor for the torsional moment. (Real ≥ 1.0) Value less than 1.0 will be reset to 1.0.
KY	For FSI=6, the user defined flexibility factor for the out-of-plane bending moment. (Real ≥ 1.0) Value less than 1.0 will be reset to 1.0.
KZ	For FSI=6, the user defined flexbility factor for the in-plane bending moment. (Real ≥ 1.0) Value less than 1.0 will be reset to 1.0.

Field	Contents
SY	For FSI=6, the user defined stress intensifictation factor for the out-of-plane bending. (Real ≥ 1.0) Value less than 1.0 will be reset to 1.0.
SZ	For FSI=6, the user defined stress intensification factor for the in-plane bending. (Real ≥ 1.0) Value less than 1.0 will be reset to 1.0.

CBEND Element Coordinate System

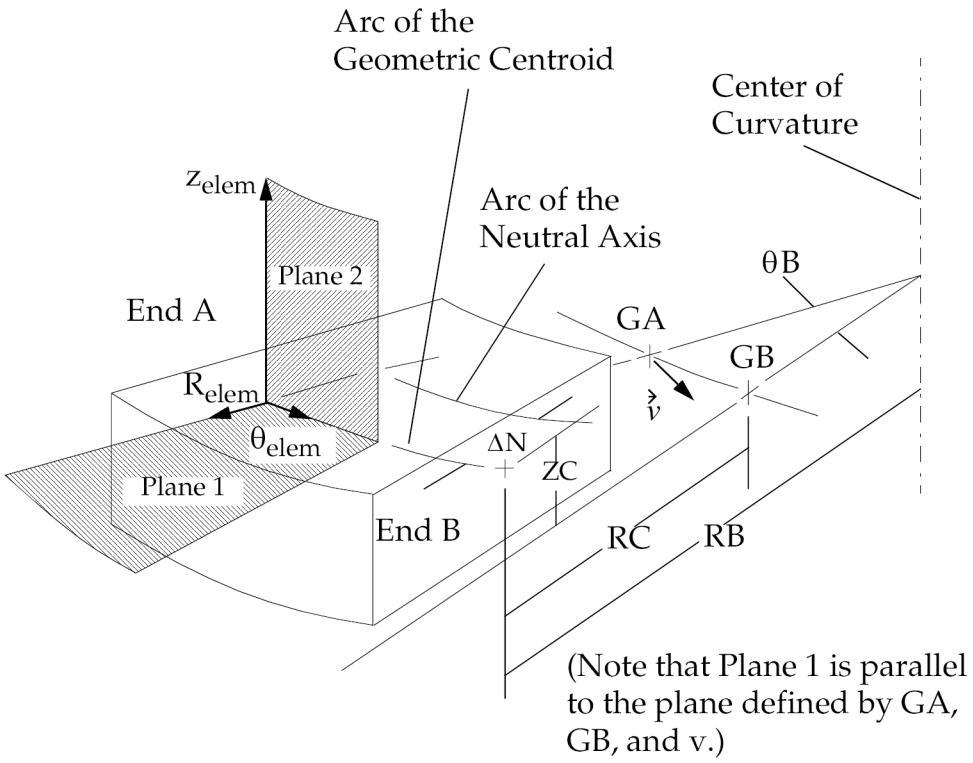


Figure 3-28. CBEND Element Coordinate System

Offsets of the ends of the element from the grid points are the same at both ends. The offsets are measured in the element coordinate system as shown in Figure 3-28. The element coordinate system is defined by one of four methods that you specify in the GEOM field on the CBEND entry.

- The *Z* -direction of the element coordinate system is defined by the cross product $\vec{v}_{ab} \times \vec{v}$ of the vector $\vec{v}_{ab}$ connecting grid point GA to grid point GB and the vector $\vec{v}$ for GEOM = 1. For GEOM = 2, 3, or 4, the *Z* -direction is defined by the cross product $\vec{v} \times \vec{v}_{ab}$. The center of curvature and intersection of the tangent lines from end A and end B are located using the data required for each of the four options.

- The *R* -direction is obtained by the vector extending from the center of curvature to end *A* . The θ -direction is the cross product of $\vec{Z} \times \vec{R}$. When θ = 0, end *A* of the element is indicated and θ = θ_B represents end B. Plane 1 of the element lies in the *R* θ plane of the element coordinates.

- Plane 1 is parallel to the plane defined by GA, GB and the vector $\vec{v}$, but it is offset by ZC in the *Z* -direction. Plane 2 lies in the θ*Z* plane and is offset from GA and GB by RC in the *R* -direction. The subscripts 1 and 2 refer to forces and geometric properties associated with bending in Planes 1 and 2, respectively. These reference planes are the principal planes of the element cross section.

The neutral axis radial offset shown in Figure 3-28 from the geometric centroid due to bending of a curved beam with a constant radius of curvature is defined as follows:

$$\Delta N = \frac{R_B}{1 + \frac{A R_R^2}{Z}}$$

Equation 3-8.

where:

R_B = the bend radius
A = the cross section area
Z = $\int \frac{r^2 dA}{1 + \frac{R}{R_B}}$
r = a local variable aligned with R_{elem} direction

You can use the default provided with the general format or you can calculate and input a value using the above formula. For the circular section format, the neutral axis offset is automatically calculated with analytical expressions for hollow and solid circular cross-sectional elements.

Flexibility and stress intensification factors

The flexibility factors which multiply the bending terms of the flexibility matrix and the stress intensification factors are selected by the FSI field on the hollow circular section format of the property entry. The options available are as follows:

FSI = 1:		
In and out-of-plane flexibility factors	=	1.0
Out-of-plane stress intensification factor	=	1.0

In-plane stress intensification factor	=	$\frac{I1}{(A)\cdot(R_B)}\left[\frac{1}{r_{elem}}+\frac{R_B-\Delta N}{\Delta N(R_B+r_{elem})}\right]$
FSI = 2:		ASME code Section III, NB-3687.2, NB-3685.2., 1977
In and out-of-plane flexibility factors	=	$\frac{1.65(R_M)^2}{(R_B)(T)}\left[\frac{1}{1+6\frac{(P)(R_M)}{(E)(T)}\left(\frac{R_M}{T}\right)^{4/3}\left(\frac{R_B}{R_M}\right)^{1/3}}\right]$
Out-of-plane stress intensification factor	=	$\nu\left(\sin\phi+\frac{[(1.5X_2-18.75)\sin 3\phi+11.25\sin 5\phi]}{X_4}\right)+\frac{\lambda(9X_2\cos 2\phi+225\cos 4\phi)}{X_4}$
In-plane stress intensification factor	=	$\nu\left(\cos\phi+\frac{[(1.5X_2-18.75)\cos 3\phi+11.25\cos 5\phi]}{X_4}\right)-\frac{\lambda(9X_2\sin 2\phi+225\sin 4\phi)}{X_4}$
where:		
λ:	=	$\frac{(R_B)(T)}{(R_M)^2\sqrt{1-\nu^2}}$
Ψ	=	$\frac{(P)(R_B)^2}{(E)(R_M)(T)}$
X_1	=	$5 + 6\lambda^2 + 24\ \Psi$
X_2	=	$17 + 600\lambda^2 + 480\ \Psi$
X_3	=	$X_1\ X_2 - 6.25$
X_4	=	$(1 - \nu^2)\ (X_3 - 4.5\ X_2)$
φ	=	Locations of stress recovery on the cross section at the locations 0°, 90°, 180°, 270°
FSI = 3:		Empirical factors from the Welding Research Council Bulletin 179, by Dodge and Moore

In and out-of-plane flexibility factors	=	$\frac{1.73}{\lambda}\left[\frac{1}{1 + 1.75\lambda^{-4/3}\exp(-1.15\Psi^{-1/4})}\right]$
In and out-of-plane stress intensification factors	=	$\frac{2\lambda^{-2/3}\left(1 + 0.25\left(\frac{R_B}{R_M}\right)^{-1}\right)}{1 + \lambda^{-4/3}\exp(-\Psi^{-1/4})}$
FSI = 4:		ASME code N-319-3 (approval date of January 17,2000).
Out-of-plane flexibility factor	=	$\frac{1.25(R_M)^2}{(T)(R_B)}\left[\frac{1}{1 + (((P)(R_M))/((T)(E)))X_k}\right]$ cannot be less than 1.0.
In-plane flexibility factors	=	$\frac{1.65(R_M)^2}{(T)(R_B)}\left[\frac{1}{1 + (((P)(R_M))/((T)(E)))X_k}\right]$ for THETAB ≥ 180° $\frac{1.30(R_M)^2}{(T)(R_B)}\left[\frac{1}{1 + (((P)(R_M))/((T)(E)))X_k}\right]$ for THETAB = 90° $\frac{1.10(R_M)^2}{(T)(R_B)}\left[\frac{1}{1 + (((P)(R_M))/((T)(E)))X_k}\right]$ for THETAB = 45° $\frac{1.0(R_M)^2}{(T)(R_B)}\left[\frac{1}{1 + (((P)(R_M))/((T)(E)))X_k}\right]$ for THETAB = 0° Linear interpolation of THETAB will be done for values between 180° and 0°; KZ (in-plane flexibility factor) shall not be less than 1.0
Out-of-plane stress intensification factor	=	$\frac{1.71}{((T)(R_B)/(R_M)^2)^{0.53}}$, but not less than 1.0.

In-plane stress intensification factors	=	$\frac{1.95}{((T)(R_B)/(R_M)^2)^{2/3}}$ for THETAB ≥ 90° $\frac{1.75}{((T)(R_B)/(R_M)^2)^{0.56}}$ for THETAB = 45° 1.0 for THETAB = 0° Linear interpolation with THETAB will be done, but the in-plane stress intensification factor shall not be less than interpolated for THETAB = 30° and not less than 1.0 for any THETAB.
where:		$X_k = 6(R_M/T)^{4/3}(R_B/R_M)^{1/3}$
FSI = 5:		ASME code B31.1 - 2001 which defines flexibility and stress intensification factors for an elbow, pipes and miter bends. These flexibility factors also apply to the class 2 (2001 edition of ASME Boiler & Pressure Vessel Code NC-3600) & class 3 (2001 edition of ASME Boiler & Pressure Vessel Code ND-3600) with the only difference being that the flexibility correction for pressure is not specified in the Figure NC/ND-3673.2(b)-1 equations but defaults to the same equation when the pressure is input as zero. All must be greater or equal to 1.0.
In and Out-of-plane flexibility factors	=	Welding elbow or pipe bend: $\frac{1.65}{h_e}$ Closely spaced miter bend: $\frac{1.52}{(h_c)^{5/6}}$ Widely spaced miter bend: $\frac{1.52}{(h_w)^{5/6}}$

In and Out-of-plane stress intensification factors	=	Welding elbow or pipe bend: $\frac{.9}{(h_e)^{2/3}}$ Closely spaced miter bend: Widely spaced miter bend: $\frac{.9}{(h_c)^{2/3}}$ $\frac{.9}{(h_w)^{2/3}}$
where:		
h_e	=	(T) (R_B / (R_M)^2) (for welding elbow or pipe bend)
h_c	=	(SACL) (T) cot (ALPHA) / 2 (R_M)^2) (for closely spaced miter bend)
h_w	=	T (1 + cot (ALPHA)) / (2 R_M) (for widely spaced miter bend)
FSI=6		User definable flexibility and stress intensification factors: KY = out-of-plane flexibility factor, KZ = in-plane flexibility factor, SY = out-of-plane stress factor, SZ = in-plane stress factor. All must be greater or equal to 1.0.

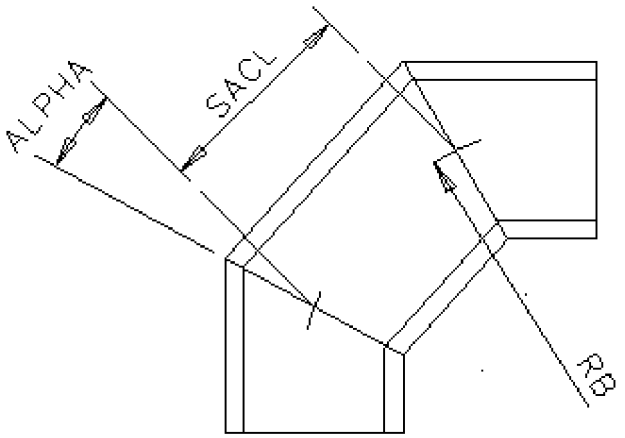


Figure 3-29. Definition of SACL and ALPHA

where

$$R_B = \frac{(SACL)\cot(ALPHA)}{2}$$

for a closely spaced miter

$$R_B = \frac{R_M(1 + \cot(ALPHA))}{2}$$

for a widely spaced miter

The positive sign conventions for internal element forces are shown in Figure 3-30. The following element forces, either real or complex (depending on the rigid format), are output on request at both ends:

- Bending moments in the two reference planes, M_1 and M_2.
- Shears in the two reference planes, V_1 and V_2.
- Average axial force, F_θ.
- Torque about the bend axis, M_θ.

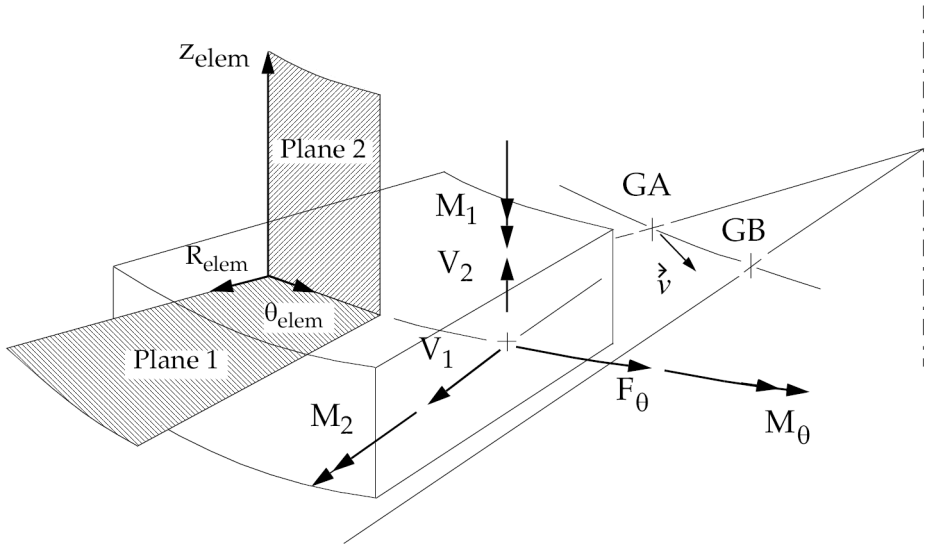
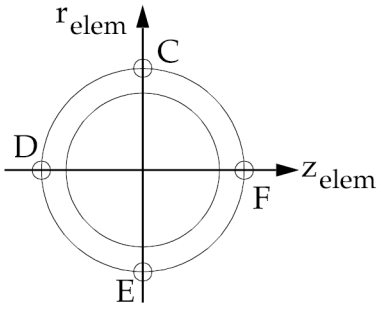


Figure 3-30. CBEND Element Internal Forces and Moments

The following real element stress data are output on request:

- Real longitudinal stress at the four points which are the same at both ends for the general cross-sectional property entry format. If the circular cross-sectional property and format is used, the stress points are automatically located at the points indicated on the following figure.



- Maximum and minimum longitudinal stresses.
- Margins of safety in tension and compression for the element if you enter stress limits on the MAT1 entry.
- When you use the pipe format, NX Nastran modifies the stress data to account for stress intensification resulting from internal pressurization and curvature of the element. The internal pressure is prescribed on the property entry. The methods used to calculate the stress intensification factor are selected through the FSI parameters.

  See Also

  o PBEND in the *NX Nastran Quick Reference Guide*

Tensile stresses are given a positive sign and compressive stresses a negative sign. Only the longitudinal stresses are available as complex stresses. The stress recovery coefficients on the general form of the PBEND entry are used to locate points on the cross section for stress recovery. The subscript 1 is associated with the distance of a stress recovery point from Plane 2. The subscript 2 is associated with the distance from Plane 1. If zero value stress recovery coefficients are used, the axial stress is output.

3.5 CONROD Rod Element

The CONROD entry is an alternate form of the CROD element that includes both the connection and property information on a single entry. It has two grid points, one at each end, and supports axial force and axial torsion. Thus, stiffness terms exist for only two DOFs per grid point. All element connectivity and property information is contained directly on the CONROD entry—no separate property entry is required. This element is convenient when you're defining several rod elements that have different properties.

Figure 3-31. CONROD Element Convention

The CONROD element x-axis (x_{elem}) is defined along the line connecting G1 to G2. Torque T is applied about the x_{elem} axis in the right-hand rule sense. Axial force P is shown in the positive (tensile) direction.

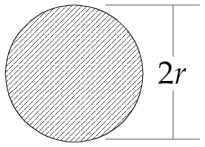
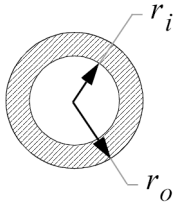
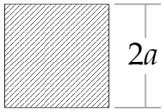
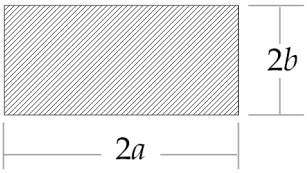
CONROD Format

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

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)

MID in field 5 points to a MAT1 material property entry. Equations used to calculate the torsional constant J (field 7) are shown below for a variety of cross sections.

Table 3-3. Torsional Constant J for Line Elements

Type of Section	Formula for J	Cross Section
Solid Circular	$J = \frac{1}{2}\pi r^4$	
Hollow Circular	$J = \frac{1}{2}\pi(r_o^4 - r_i^4)$	
Solid Square	$J = 2.25\ a^4$	
Solid Rectangular	$J = ab^3\left[\frac{16}{3} - 3.36\frac{b}{a}\left(1 - \frac{b^4}{12a^4}\right)\right]$	

The torsional stress coefficient C (field 8) is used by NX Nastran to calculate torsional stress according to the following relation:

$$\tau = C\frac{M_\theta}{J}$$

Equation 3-9.

3.6 CROD Element

The rod element is defined with a CROD entry and its properties with a PROD entry. The CROD element is a straight prismatic element (the properties are constant along the length) that has only axial and torsional stiffness. The CROD element is the same as the CONROD element, except that its element properties are listed on a separate Bulk Data entry (the PROD rod element property). This element is convenient when defining you're defining rod elements that have the same properties.

The CROD element is the simplest element of all the elements that have geometry associated with them. While you can use a CBAR or CBEAM element to represent a rod member, these elements are somewhat more difficult to define because you must explicitly specify an element coordinate system. The CROD element is ideal when you need an element with only tension-compression and torsion.

The structural and nonstructural mass of the rod are lumped at the adjacent grid points unless you request coupled mass with PARAM,COUPMASS.

See Also

- COUPMASS in the *NX Nastran Quick Reference Guide*
- Section 5.2 of *The NASTRAN Theoretical Manual* (for theoretical aspects of the rod element)

The *x* -axis of the element coordinate system is defined by a line connecting end a to end b, as shown in Figure 3-32. The axial force and torque are output on request in either real or complex form. The positive directions for these forces are indicated in Figure 3-32. NX Nastran outputs the following real element stresses on request:

- axial stress
- torsional stress
- margin of safety for axial stress
- margin of safety for torsional stress

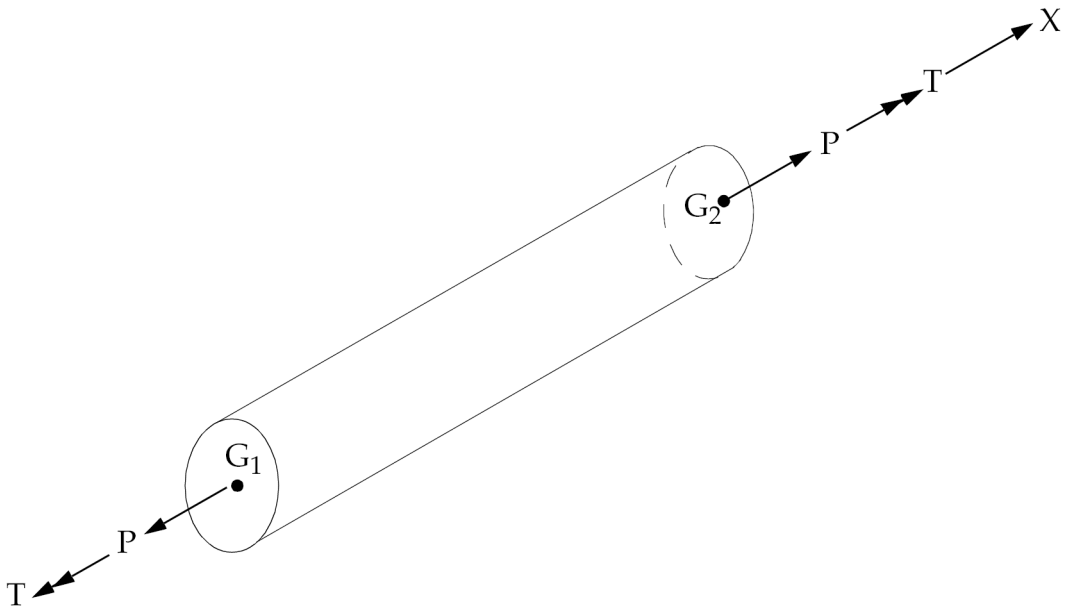


Figure 3-32. Rod Element Coordinate System and Element Forces

Positive directions are the same as those indicated in Figure 3-32 for element forces. Only the axial stress and the torsional stress are available as complex stresses.

CROD Format

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PROD entry. (Integer > 0; Default is EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)

You define the CROD element by specifying the two grid points G1 and G2 that denote the end points of the element. The PID identifies the PROD entry that defines the cross-sectional area A, and the torsional constant J associated with the CROD element. If you don't define these values on the PROD entry, the CROD elements will lack axial and torsional stiffness.

See Also

- CROD in the *NX Nastran Quick Reference Guide*

PROD Format

You use the PROD entry to define properties for the CROD element, such as cross-sectional area and torsional stiffness.

See Also

- PROD in the *NX Nastran Quick Reference Guide*

CROD Element Coordinate System

The conventions for the element coordinate system and the internal forces of the CROD element are shown below.

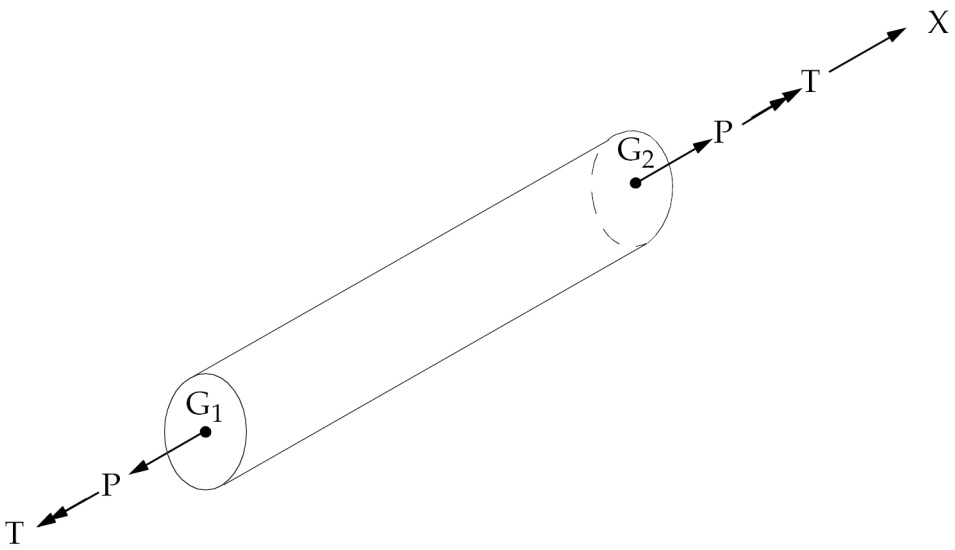


Figure 3-33. CROD Element Internal Forces and Moments

CROD Example

To illustrate the use of the CROD element, consider a three member truss structure attached to a rigid wall as shown in Figure 3-34. The input file is shown in Listing 3-7.

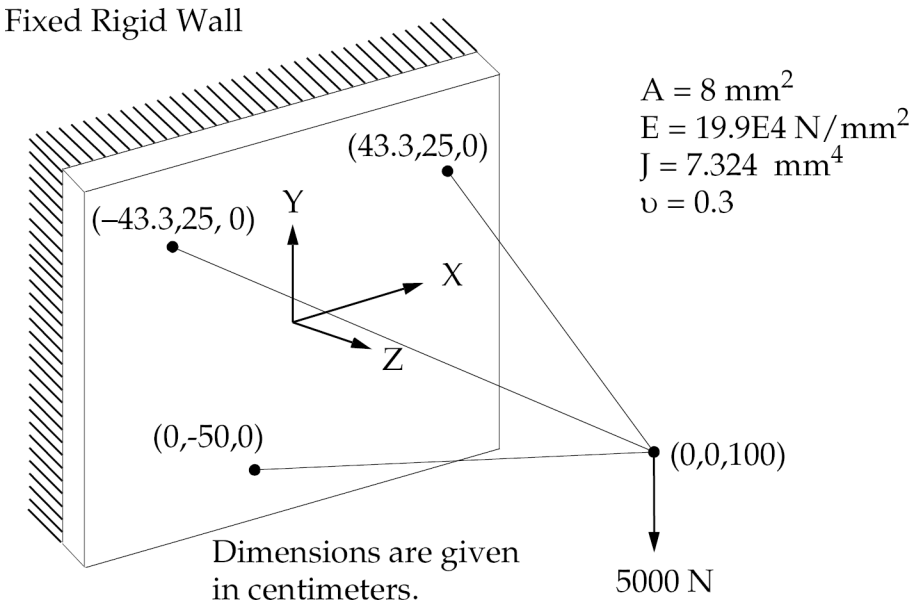


Figure 3-34. Three Member Structure

```
$ FILENAME - ROD2.DAT
ID LINEAR,ROD2
SOL 101
TIME 2
CEND
TITLE = NX NASTRAN USER'S GUIDE
SUBTITLE = THREE ROD TRUSS MODEL
LABEL = POINT LOAD AT GRID POINT 4
LOAD = 1
DISPLACEMENT = ALL
STRESS = ALL
BEGIN BULK
$
$ THE GRID POINTS LOCATIONS DESCRIBE THE GEOMETRY
$ DIMENSIONS HAVE BEEN CONVERTED TO MM FOR CONSISTENCY
$
GRID    1               -433.   250.    0.              123456
GRID    2               433.    250.    0.              123456
GRID    3               0.      -500.   0.              123456
GRID    4               0.      0.      1000.
$
$ MEMBERS ARE MODELED USING ROD ELEMENTS
$
CROD    1       1       1       4
CROD    2       1       2       4
CROD    3       1       3       4
$
$ PROPERTIES  OF ROD ELEMENTS
$
PROD    1       1       8.      7.324
$
$ MATERIAL PROPERTIES
$
MAT1    1       19.9E4          .3
$
$ POINT LOAD
$
FORCE   1       4               5000.   0.      -1.     0.
$
ENDDATA
```

Listing 3-7. Three Member Truss Example

A selected portion of the output illustrating the displacements and the stresses is shown below.

```
D I S P L A C E M E N T   V E C T O R

  POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
        1      G      0.0            0.0            0.0            0.0            0.0            0.0
        2      G      0.0            0.0            0.0            0.0            0.0            0.0
        3      G      0.0            0.0            0.0            0.0            0.0            0.0
        4      G      0.0          -1.170466E+01  -2.575031E-05   0.0            0.0            0.0

                                S T R E S S E S   I N   R O D   E L E M E N T S     ( C R O D )
   ELEMENT       AXIAL       SAFETY      TORSIONAL     SAFETY       ELEMENT       AXIAL       SAFETY      TORSIONAL     SAFETY
     ID.         STRESS      MARGIN        STRESS      MARGIN         ID.         STRESS      MARGIN        STRESS      MARGIN
         1    4.658455E+02              0.0                                2    4.658455E+02              0.0
         3   -9.316950E+02              0.0
```

Figure 3-35. Three Member Truss Selected Output

The boundary conditions of the truss are applied as permanent constraints using the value 123456 in field 8 of the GRID entries with IDs 1, 2, and 3. There are no constraints applied to grid point 4, so it is free to move in any direction. To verify that boundary conditions are applied correctly, review

the displacement vector in Figure 3-35. The displacements of grid points 1, 2, and 3 are exactly equal to zero in each direction.

Note that grid point 4 displacement is acting in the YZ plane. This makes sense because the structure is symmetric about the YZ plane, and the applied load is in the Y-direction. Also note that the rotation of grid point 4 is equal to zero in each of the three directions. The CROD element can't transmit a bending load, and as such, the rotation at the end of the CROD element is not coupled to the translational degrees of freedom. In fact, if the rotation degrees of freedom for grid point 4 is constrained, the answers are the same.

The stress output shows an axial and torsional stress only. There are no bending stresses in a CROD element. In this case, the torsional stress is zero because there is no torsional load on the members. The safety of margin is blank in our example because we left the stress limits fields in the MAT1 entry blank

3.7 CTUBE Element

The CTUBE element is the same as the CROD element except that its section properties are expressed as the outer diameter and the thickness of a circular tube. The extensional and torsional stiffness are computed from these tube dimensions.

You define the tube element with a CTUBE entry, and its properties with a PTUBE entry.

See Also

- CTUBE in the *NX Nastran Quick Reference Guide*
- PTUBE in the *NX Nastran Quick Reference Guide*

3.8 CVISC Element

Another kind of rod element is the viscous damper. It has extensional and torsional viscous damping properties rather than stiffness properties. The viscous damper element is defined with a CVISC entry and its properties with a PVISC entry. This element is used in the formulation of dynamic matrices. You must also select the mode displacement method (PARAM,DDRMM,-1) for element force output.

See Also

- CVISC in the *NX Nastran Quick Reference Guide*
- PVISC in the *NX Nastran Quick Reference Guide*
- DDRMM in the *NX Nastran Quick Reference Guide*

Chapter 4: 2D Elements

- *Introduction to Two-Dimensional Elements*
- *The Shear Panel Element (CSHEAR)*
- *Two-Dimensional Crack Tip Element (CRAC2D)*
- *Conical Shell Element (RINGAX)*
- *Plate and Shell Elements*
- *Plane Stress and Plane Strain Elements*

4.1 Introduction to Two-Dimensional Elements

Surface elements, also called two-dimensional elements, are used to represent a structure whose thickness is small compared to its other dimensions. You can use surface elements to model plates, which are flat, or shells, which have single curvature (e.g. cylinder) or double curvature (e.g. sphere). For the grid points used to represent plate elements, stiffness terms exist for five of the possible six degrees of freedom per grid point. There is no stiffness associated to the rotation about the normal to the plate. This rotational DOF must be constrained to prevent stiffness singularities.

In NX Nastran, you can use the following types of surface elements

- Shear panel (CSHEAR)
- 2D crack tip element (CRAC2D)
- Conical shell (RINGAX)
- Shell (CQUAD4, CTRIA3, CQUAD8, CTRIA6, CQUADR, CTRIAR)

For linear analysis, NX Nastran plate elements assume classical assumptions of thin plate behavior:

- A thin plate is one in which the thickness is much less than the next larger dimension.
- The deflection of the plate's midsurface is small compared with its thickness.
- The midsurface remains unstrained (neutral) during bending—this applies to lateral loads, not in-plane loads.
- The normal to the midsurface remains normal during bending.

4.2 The Shear Panel Element (CSHEAR)

You define a shear panel element with a CSHEAR entry and its properties with a PSHEAR entry. A shear panel is a two-dimensional structural element that resists the action of tangential forces applied to its edges, and the action of normal forces if effectiveness factors are used on the PSHEAR Bulk Data entry. The structural and nonstructural mass of the shear panel is lumped at the connected grid points. Details of the shear panel element are described in Section 5.3 of *The NASTRAN Theoretical Manual.*

The most important application of the CSHEAR element is in the analysis of thin reinforced plates and shells, such as thin aircraft skin panels. In such applications, reinforcing rods (or beams) carry the extensional load, and the CSHEAR element carries the in-plane shear. This is particularly true if the real panel is buckled or if it is curved.

The CSHEAR element is a quadrilateral element with four grid points. The element models a thin buckled plate. It supports shear stress in its interior and also extensional force between adjacent grid points. Typically you use the CSHEAR element in situations where the bending stiffness and axial membrane stiffness of the plate is negligible. Using a CQUAD4 element in such situations results in an overly stiff model.

See Also

- "CSHEAR" in the *NX Nastran Quick Reference Guide*

- “PSHEAR” in the *NX Nastran Quick Reference Guide*

CSHEAR Element Coordinate System

The element coordinate system for a shear panel is shown in Figure 4-1. The labels G1, G2, G3, and G4 refer to the order of the connected grid points on the CSHEAR entry.

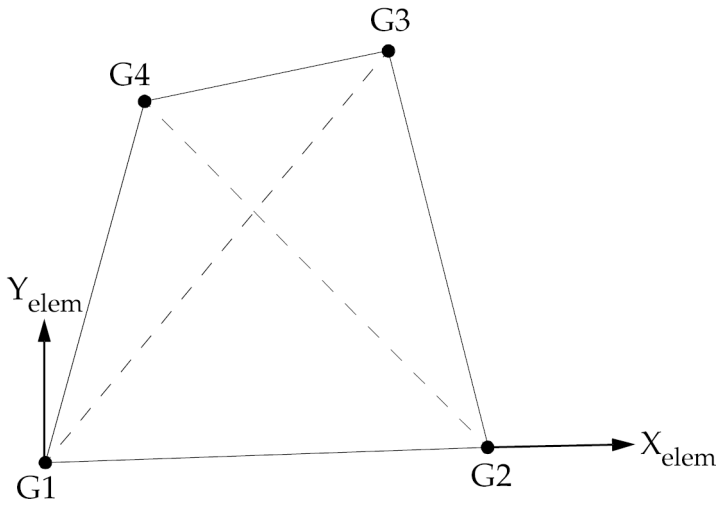


Figure 4-1. Shear Panel Connection and Coordinate System

CSHEAR Output

You can have NX Nastran output CSHEAR element forces in either real or complex form. The output for the CSHEAR element is the components of force at the corners of the element, the shear flows (force per unit length) along each element edge, the average shear stress, and the maximum shear stress. Positive directions for these quantities are identified in the figure below.

NX Nastran calculates the shear stresses at the corners in skewed coordinates parallel to the exterior edges. You can also have NX Nastran output the average of the four corner stresses and the maximum stress are output on request in either the real or complex form. The software also calculates a margin of safety when you request stresses in real form.

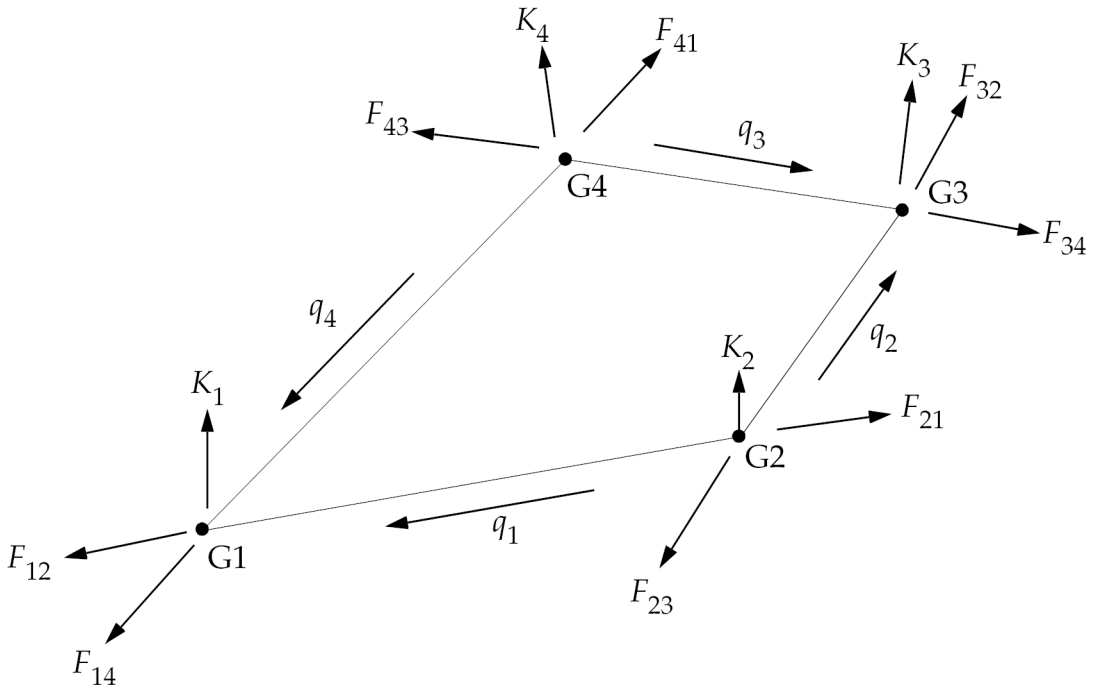


Figure 4-2. CSHEAR Element Corner Forces and Shear Flows

4.3 Two-Dimensional Crack Tip Element (CRAC2D)

NX Nastran crack tip elements include both two-dimensional (CRAC2D) and three-dimensional (CRAC3D) types. You can use the two-dimensional crack tip elements to model surfaces with a discontinuity due to a crack. You use the CRAC2D entry to define the element geometry and the PRAC2D entry to define its properties. You can model a CRAC2D element with temperature-independent anisotropic materials. The 2-D element may be either plane stress or plane strain. The element generates either coupled or lumped mass matrices.

See Also

- “Three-Dimensional Crack Tip Element (CRAC3D)”

The CRAC2D element is based upon a 2-D formulation, but you can use it in three-dimensional structures. However, the element should be planar. NX Nastran checks for any deviation from a planar element, and if it detects significant deviations, it issues error messages.

The figures below show quadrilateral and symmetric half options for the CRAC2D element. Grid points 1 through 10 are required; grid points 11 through 18 are optional. The element may be plane stress or plane strain. You may specify a quadrilateral or a symmetric half option.

- For the quadrilateral option, NX Nastran automatically divides the element into eight basic triangular elements (1-8). For the symmetric half-crack option, NX Nastran subdivides the element into four basic triangular elements. For the quadrilateral option, the stresses are computed by averaging the stresses from triangles 4 and 5, and the stress intensity factors K_i and K_i are computed from triangles 1 and 8.

Stresses and the local coordinates of these stresses for the quadrilateral option are computed at the origin of the natural coordinates of triangles 4 and 5. The stresses and coordinates are then averaged and reported. Stress intensity factors, K_I and K_{II} , are computed for triangles 1 and 8, averaged and reported.

- For the symmetric half option, NX Nastran automatically divides the element into four basic triangular elements (1-4). The stress is determined from triangle 4, and the stress intensity factor K_i is computed from triangle 1 only. Grid points 1 through 7 are required for the symmetric half-crack option, while grid points 11 through 14 are optional.

  For the symmetric half-crack option, coordinates and stresses are reported at the origin of the natural coordinates of triangle 4 while the stress intensity factor K_I only is reported for triangle 1.

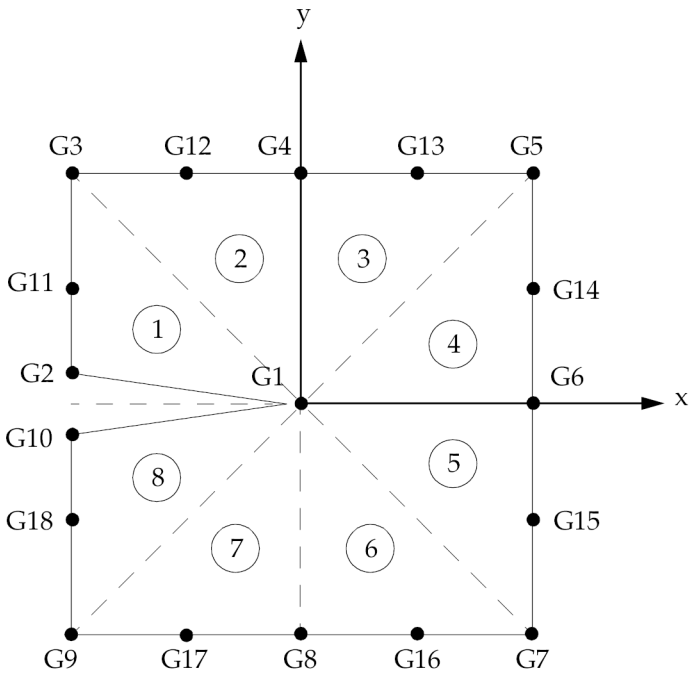


Figure 4-3. Quadrilateral Crack Element

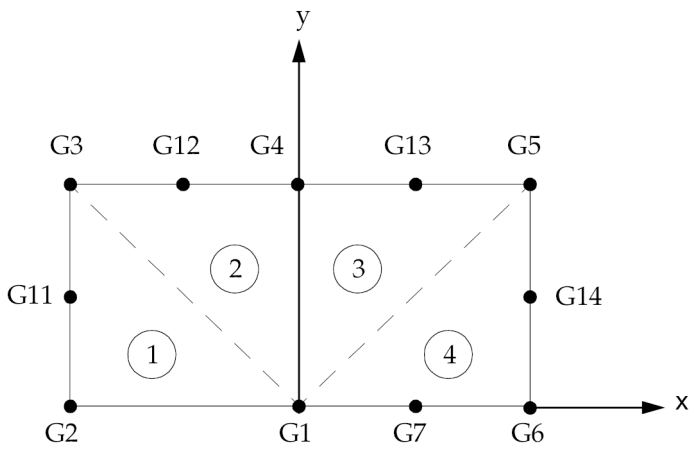


Figure 4-4. Symmetric Half-Crack Option

S1	S2	S3	S4	S5	S6	S7	S8	S9
x	y	σ_x	σ_y	τ_{xy}	K_I	K_{II}	0	0

where *x* and *y* are the element coordinates where stresses are reported. K_I and K_{II} are stress intensity factors.

CRAC2D and ADUM8 Format

Because the CRAC2D is not fully implemented, you must supply additional input on the ADUM8 entry. The element is what is known as a “dummy” element because it was added to NX Nastran using a prototype element routine. NX Nastran includes prototype element routines to allow advanced users to add their own elements to the element library. The formats of the CRAC2D and ADUM8 entries are as follows:

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					

ADUM8	18	0	5	0	CRAC2D				

Field	Contents
EID	Element identification number.
PID	Property identification number of a PRAC2D entry.
Gi	Grid point identification numbers of connection points.

See Also

- “CRAC2D” inthe *NX Nastran Quick Reference Guide*
- “ADUMi” in the *NX Nastran Quick Reference Guide*

PRAC2D

When you create CRAC2D elements, you enter its properties and the stress evaluation techniques on the PRAC2D entry.

See Also

- "PRAC2D" in the *NX Nastran Quick Reference Guide*

4.4 Conical Shell Element (RINGAX)

In NX Nastran, you can create a conical shell element with the RINGAX bulk data entry. The properties of the conical shell element are assumed to be symmetrical with respect to the axis of the shell. However, the loads and deflections need not be axisymmetric because they are expanded in Fourier series with respect to the azimuthal coordinate. Due to symmetry, the resulting load and deformation systems for different harmonic orders are independent, a fact that results in a large time saving when the use of the conical shell element is compared with an equivalent model constructed from plate elements. See *The NASTRAN Theoretical Manual* for the theoretical aspects of this element

You can't combine the conical shell element with other types of elements. This is primarily because the Fourier coefficients are stored on internally generated pseudo grid points. The unconventional nature of this element results in its capabilities being limited. For example, you can't use RINGAX in the superelement solution sequences.

See Also

- "Element Summary – Small Strain Elements"

Defining the Problem with AXIC

You use the AXIC entry to indicate that you're analyzing a conical shell problem. This entry also indicates the number of harmonics desired in the problem formulation. You can only use a a limited number of Bulk Data entries when your model contains conical shell elements.

See Also

- "AXIC" in the *NX Nastran Quick Reference Guide*

Defining the Geometry with RINGAX

With a conical shell problem, you define the model's geometry with RINGAX entries instead of GRID entries. The RINGAX entries describe concentric circles about the basic *z* -axis, with their locations given by radii and *z* -coordinates as shown in Figure 4-5. The degrees-of-freedom defined by each RINGAX entry are the Fourier coefficients of the motion with respect to angular position around the circle. For example, the radial motion u_r at any angle φ is described by the equation

$$u_r(\phi) = \sum_{n=0}^{N} u_r^n \cos n\phi + \sum_{n=0}^{N} u_r^{n*} \sin n\phi$$

Equation 4-1.

where $u^n{}_r$ and $u^{n*}{}_r$ are the Fourier coefficients of radial motion for the n -th harmonic. For calculation purposes, the series is limited to N harmonics as defined by the AXIC entry. The first sum in the above equation describes symmetric motion with respect to the φ plane. The second sum with the "starred" (*) superscripts describes the antisymmetric motion. Thus each RINGAX entry will produce six times N degrees-of-freedom for each series.

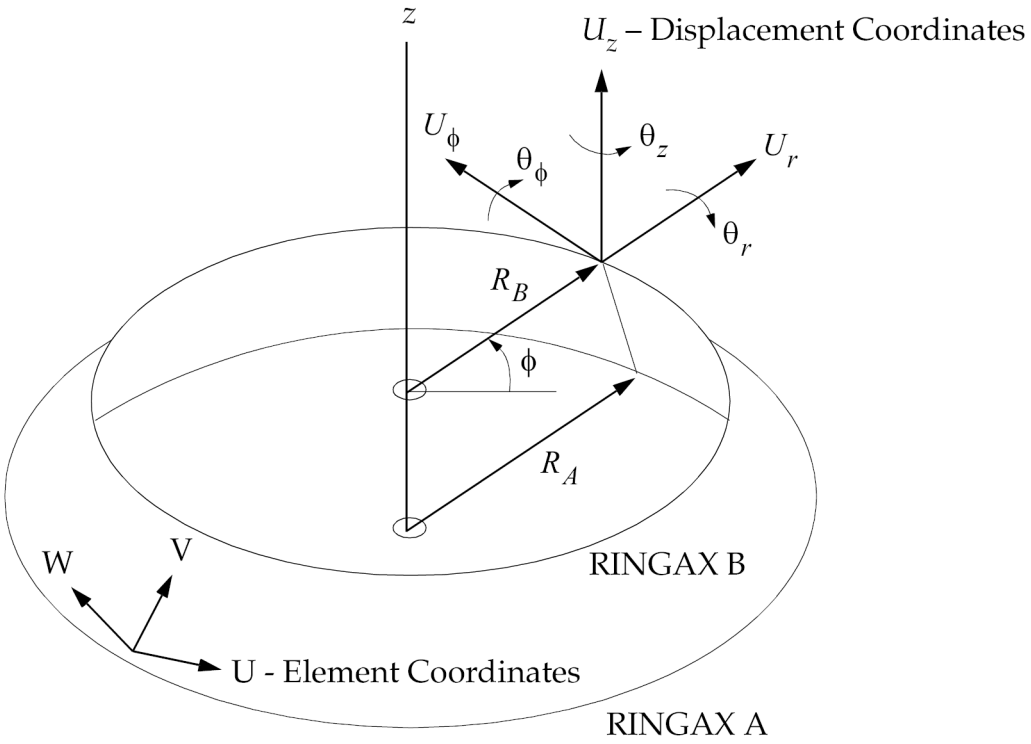


Figure 4-5. Geometry for Conical Shell Element

The selection of symmetric or antisymmetric solutions is controlled by the AXISYMMETRIC case control command. For general loading conditions, a combination of the symmetric and antisymmetric solutions must be made using the SYMCOM case control command. If the AXISYMMETRIC command isn't present, the software ignores stress and element force requests.

See Also

- "AXISYMMETRIC" in the *NX Nastran Quick Reference Guide*
- "SYMCOM" in the *NX Nastran Quick Reference Guide*

Since it is rare to be interested in applying loads in terms of Fourier harmonics and interpreting data by manually performing the above summations, NX Nastran provides special data entries which automatically perform these operations:

- You use the POINTAX entry like a GRID entry to define physical points on the structure for loading and output.
- You define sections of the circle with a SECTAX entry which defines a sector with two angles and a referenced RINGAX entry.

The POINTAX and SECTAX entries define six degrees-of-freedom each. The basic coordinate system for these points is a cylindrical system (r , φ , z), and their applied loads must be described in this coordinate system. Since the displacements of these points are dependent on the harmonic motions, they may not be constrained in any manner.

The conical shell element is connected to two RINGAX rings with a CCONEAX entry. The properties of the conical shell element are described on the PCONEAX entry. The RINGAX points must be placed on the neutral surface of the element, and the points for stress calculation must be given on the PCONEAX entry relative to the neutral surface. Up to fourteen angular positions around the element may be specified for stress and force output. These values will be calculated midway between the two connected rings.

Constraints for Conical Shells

The structure defined with RINGAX and CCONEAX entries must be constrained in a special manner. All harmonics may be constrained for particular degree-of-freedom on a ring by using permanent single-point constraints on the RINGAX entries. Specified harmonics of each degree-of-freedom on a ring may be constrained with an SPCAX entry. The entry is the same as the SPC entry except that a harmonic must be specified. The MPCAX, OMITAX, and SUPAX data entries correspond to the MPC, OMIT, and SUPORT data entries except that harmonics must be specified. SPCADD and MPCADD entries may be used to combine constraint sets in the usual manner.

The stiffness matrix includes five degrees-of-freedom per grid circle per harmonic when transverse shear flexibility is included. Since the rotation about the normal to the surface is not included, the sixth degree-of-freedom must be constrained to zero when the angle between the meridional generators of two adjacent elements is zero. Since only four independent degrees-of-freedom are used when the transverse shear flexibility is not included, the fifth and sixth degrees-of-freedom must be constrained to zero for all rings. You can specify these constraints on the RINGAX entry.

Loading for Conical Shells

You can load a conical shell structure in various ways. For example:

- You can create concentrated forces by applying FORCE and MOMENT entries to POINTAX points.
- You can input pressure loads with the PRESAX data entry which defines an area bounded by two rings and two angles.
- You can use the TEMPAX entry to define temperature sets. Temperature fields are described by a paired list of angles and temperatures around a ring.
- You can create direct loads on the harmonics of a RINGAX point with the FORCEAX and MOMAX entries.

  Since the basic coordinate system is cylindrical, the loads are given in the r , φ , and z directions. The value of a harmonic load F_n is the total load on the whole ring of radius r . If a sinusoidal load-per-unit length of maximum value is given, the value on the FORCEAX entry must be

$$F_n = 2\pi r a_n \qquad n = 0$$

Equation 4-2.

$$F_n = \pi r a_n \qquad n = 0$$

Equation 4-3.

Output for Conical Shells

You can request displacements of rings and forces in conical shell elements in two ways:

- The harmonic coefficients of displacements on a ring or forces in a conical element.
- The displacements at specified points or the average value over a specified sector of a ring. The forces in the element at specified azimuths or average values over specified sectors or a conical element.

Harmonic output is requested by ring number for displacements and conical shell element number for element forces. The number of harmonics that will be output for any request is a constant for any single execution. This number is controlled by the Case Control command, HARMONICS.

See Also

- “HARMONICS” in the *NX Nastran Quick Reference Guide* .

You can request that NX Nastran output the following element forces per unit of width either as harmonic coefficients or at specified locations:

- Bending moments on the *u* and *v* faces
- Twisting moments
- Shearing forces on the *u* and *v* faces.

You can also request that NX Nastran calculate the following element stresses at two specified points on the cross section of the element and output as harmonic coefficients or at specified locations:

- Normal stresses in *u* and *v* directions
- Shearing stress on the *u* face in the *v* direction
- Angle between the *u* -axis and the major principal axis
- Major and minor principal stresses
- Maximum shear stress

4.5 Plate and Shell Elements

NX Nastran supports triangular and quadrilateral isoparametric shell elements that can be used to account for bending and/or membrane forces. These elements include:

- CTRIA3 – Isoparametric triangular element with optional coupling of bending and membrane stiffness.
- CTRIA6 – Isoparametric triangular element with optional coupling of bending and membrane stiffness and optional midside nodes.
- CTRIAR– Isoparametric triangular element with no coupling of bending and membrane stiffness; the membrane stiffness formulation includes rotation about the normal to the plane of the element.
- CQUAD4– Isoparametric quadrilateral element with optional coupling of bending and membrane stiffnesses.
- CQUAD8 – Isoparametric quadrilateral element with optional coupling of bending and membrane stiffness and optional midside nodes.
- CQUADR – Isoparametric quadrilateral element with no coupling of bending and membrane stiffnesses; the membrane stiffness formulation includes rotation about the normal to the plane of the element.

These elements differ principally in their shape, number of connected grid points, and number of internal stress recovery points. You can use each element type to model thick or thin plates and shells. The important distinction among the elements is the accuracy that is achieved when they are used in various applications.

The CQUAD8 and CTRIA6 elements have the same features as the CQUAD4 and CTRIA3 elements, but are not used as frequently. The CQUAD8 and CTRIA6 are higher-order elements that let you use mid-side nodes in addition to corner nodes. Mid-side nodes increase the accuracy of the element but can make meshing more difficult.

For accuracy reasons, the quadrilateral elements (CQUAD4 and CQUAD8) are generally preferred over the triangular elements (CTRIA3 and CTRIA6). Triangular elements are mainly used for mesh transitions or for modeling portions of a structure when quadrilateral elements are impractical.

Theoretical aspects of the plate elements are described in the *NX Nastran Theoretical Manual*.

Depending on the element, you define the properties for plate and shell elements with either PSHELL, PCOMP, PCOMPG, or PLPLANE bulk entries. Anisotropic material may be specified for all shell elements. Transverse shear flexibility may be included for all bending elements on an optional basis. Structural mass is calculated from the membrane density and thickness. Non-structural mass can be specified for all shell elements. Lumped mass procedures are used unless coupled mass is requested with the parameter COUPMASS. Differential stiffness matrices are generated for all shell elements except CQUADR and CTRIAR. A plane strain formulation can be requested for all shell elements.

PSHELL Format

The PSHELL entry defines the membrane, bending, transverse shear, and coupling properties of thin plate and shell elements. The format of the PSHELL entry is as follows:

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						

Field	Contents
PID	Property identification number.
MID1	Material identification number for the membrane.
T	Default membrane thickness for Ti.
MID2	Material identification number for bending.
$12I/T^3$	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.
MID3	Material identification number for transverse shear.
TS/T	Transverse shear thickness ratio TS/T. Ratio of the shear thickness, (TS), to the membrane thickness of the shell T. The default value is for a homogeneous shell.
NSM	Nonstructural mass per unit area.
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.
MID4	Material identification number for membrane-bending coupling.

You use the PSHELL entry to define the material ID for the membrane properties, the bending properties, the transverse shear properties, the bending-membrane coupling properties, and the bending and transverse shear parameters. By choosing the appropriate materials and parameters, virtually any plate configuration may be obtained.

The most common use of the PSHELL entry is to model an isotropic thin plate. The preferred method to define an isotropic plate is to enter the same MAT1 ID for the membrane properties (MID1) and bending properties (MID2) only and leave the other fields blank. For a thick plate, you may also wish to enter an MAT1 ID for the transverse shear (MID3). You can also use PSHELL to model anisotropic plates.

See Also

- "Using the PSHELL Method" in the *NX Nastran User's Guide*.

There are two ways you can input the thickness of the plate elements. The simplest and way is to enter a constant element thickness in field 4 of the PSHELL entry. If the element has nonuniform thickness, the thickness at each of the corner points is entered on the continuation line of the CQUAD4/CTRIA3 connectivity entry. If you enter the thickness on both the PSHELL entry and the connectivity entry, the individual corner thicknesses take precedence.

Also located on the PSHELL entry are the stress recovery locations Z1 and Z2. By default, Z1 and Z2 are equal to one-half of the plate thickness (typical for a homogeneous plate). If you are modeling a composite plate, you may want to enter values other than the defaults to identify the outermost fiber locations of the plate for stress analysis.

PID in field 2 is referenced by a surface element (e.g., CQUAD4 or CTRIA3). MID1, MID2, and MID3 are material identification numbers that normally point to the same MAT1 material property entry. T is the uniform thickness of the element. For solid homogenous plates, the default values of $121/T^3$ (field 6) and TS/T (field 8) are correct.

The CQUAD4 element can model in-plane, bending, and transverse shear behavior. The element's behavior is controlled by the presence or absence of a material ID number in the appropriate field(s) on the PSHELL entry.

To model a membrane (i.e., no bending), fill in MID1 only. For example:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1	204	.025						

To model bending only, fill in MID2 only. For example:

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204					

To add transverse shear flexibility to bending, fill in MID2 and MID3. For example:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204		204			

Adding transverse shear flexibility means that using MID3 adds a shear term in the element's stiffness formulation. Therefore, a plate element with an MID3 entry will deflect more (if transverse shear is present) than an element without an MID3 entry. For very thin plates, this shear term adds very little to the deflection result. For thicker plates, the contribution of transverse shear to deflection becomes more pronounced, just as it does with short, deep beams.

For a solid, homogeneous, thin, stiff plate, use MID1, MID2, and MID3 (all three MIDs reference the same material ID). For example:

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T		
PSHELL	1		.025	204		204			

CQUAD4 and CTRIA3 Elements

The formulation of the CQUAD4 and CTRIA3 elements are based on the Mindlin-Reissner shell theory. These elements do not provide direct elastic stiffness for the rotational degrees-of-freedom which are normal to the surface of the element.

Consequently, for example, if a grid point is attached only to CQUAD4 elements only, all the elements are in the same plane, then the rotational degrees of freedom about the surface normal have zero stiffness. This zero stiffness results in a singular stiffness matrix, which prevents NX Nastran from solving the model.

To avoid this problem, you can:

- Constrain the rotational degree-of-freedom either manually with an SPC entry (either in field 8 of the GRID entry or an SPC entry) or automatically with the AUTOSPC parameter. If using the SPC method, ensure that you do not constrain any components that have stiffness attached.

- Apply an artificial stiffness term to the degrees of freedom using PARAM K6ROT. Remember when using this parameter that the stiffness being included for the rotational degree of freedom

is not a true stiffness and should not be used as such. For example, if you want to connect a CBAR element to the CQUAD4 element, you shouldn't rely on the K6ROT stiffness to transfer the bending moment at the end of the CBAR into the plate.

See Also

- "Single-Point Constraints" in the in the *NX Nastran User's Guide*.
- "Automatically Applying Single-Point Constraints" in the *NX Nastran User's Guide*.
- "K6ROT" in the *NX Nastran Quick Reference Guide*.

CQUAD4

CQUAD4 is NX Nastran's most commonly used element for modeling plates, shells, and membranes. The CQUAD4 is a quadrilateral flat plate connecting four grid points. It can represent in-plane, bending, and transverse shear behavior, depending upon data provided on the PSHELL property entry.

You should use the CQUAD4 element when the surfaces you are meshing are reasonably flat and the geometry is nearly rectangular. For these conditions, the quadrilateral elements eliminate the modeling bias associated with the use of triangular elements, and the quadrilaterals give more accurate results for the same mesh size. If the surfaces are highly warped, curved, or swept, you should use triangular elements.

Under extreme conditions, quadrilateral elements will give results that are considerably less accurate than triangular elements for the same mesh size. Quadrilateral elements should be kept as nearly square as possible, because their accuracy tends to deteriorate as their aspect ratio increases.

CQUAD4 Element Coordinate System

The element coordinate systems for the CQUAD4 is shown in Figure 4-6. The orientation of the element coordinate system is determined by the order of the connectivity for the grid points. The element z-axis, often referred to as the positive normal, is determined using the right-hand rule. Therefore, if you change the order of the grid points connectivity, the direction of this positive normal might also reverse.

This rule is important to remember when you're applying pressure loads or viewing the element forces or stresses. Often, element stress contours appear to be strange when they're displayed by a post-processor because the normals of the adjacent elements may be inconsistent. Remember that NX Nastran always outputs components of forces, moments, and element stresses in the element coordinate system.

- The element's x-axis bisects the angle 2α. The positive direction is from G1 to G2.
- The element's y-axis is perpendicular to the element x-axis and lies in the plane defined by G1, G2, G3, and G4. The positive direction is from G1 to G4.
- The element's z-axis is normal to the x-y plane of the element. The positive z direction is defined by applying the right-hand rule to the ordering sequence of G1 through G4.

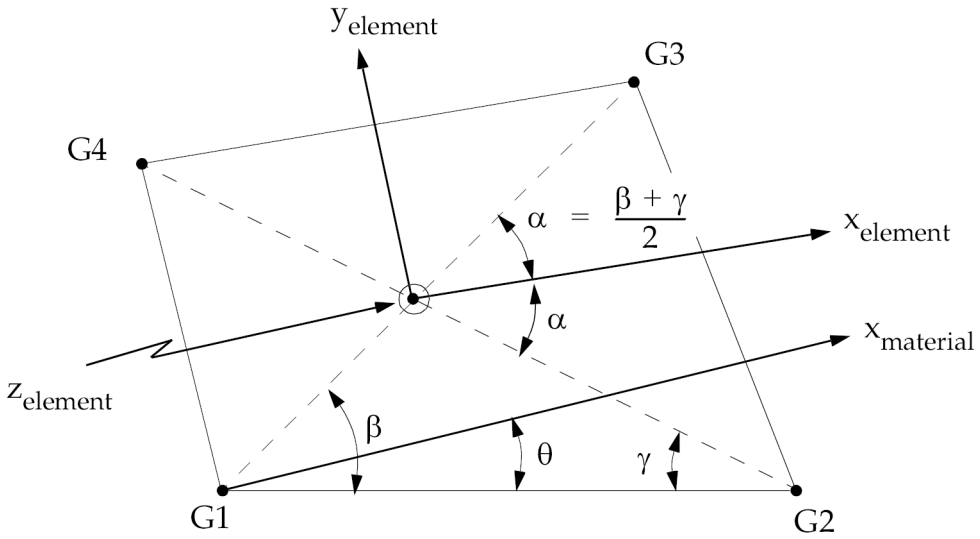


Figure 4-6. CQUAD4 Element Geometry and Coordinate Systems

Figure 4-6 also shows that each element has an element coordinate system and a material coordinate system that may be the same or different. Using a material coordinate system different from the element coordinate system is useful when the material properties are orthotropic or anisotropic.

CQUAD4 Format

The format of the CQUAD4 entry is as follows:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
			T1	T2	T3	T4			

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHELL or PCOMP entry.
Gi	Grid point identification numbers of connection points.
THETA	Material property orientation angle in degrees.
MCID	Material coordinate system identification number.
ZOFFS	Offset from the surface of grid points to the element reference plane.
Ti	Membrane thickness of element at grid points G1 through G4.

Grid points G1 through G4 must be ordered consecutively around the perimeter of the element. THETA and MCID are not required for homogenous, isotropic materials. ZOFFS is used when offsetting the element from its connection point. The continuation entry is optional. If you don't supply values for T1 to T4, the software sets them equal to the value of T (plate thickness) you define on the PSHELL entry. Finally, all interior angles of the CQUAD4 element must be less than 180°.

See Also

- "CQUAD4" in the *NX Nastran Quick Reference Guide*.

CTRIA3

The CTRIA3 element is a triangular plate connecting three grid points. The CTRIA3 is most commonly used for mesh transitions and filling in irregular boundaries. The element may exhibit excessive stiffness, particularly for membrane strain. Thus, as a matter of good modeling practice, you should locate CTRIA3 elements away from areas of interest whenever possible. In other respects, the CTRIA3 is analogous to the CQUAD4. Triangular elements should be kept as nearly equilateral as possible as their accuracy tends to deteriorate when the element's shape becomes obtuse and the ratio of the longest to the shortest side increases.

CTRIA3 Element Coordinate System

CTRIA3 element forces and stresses are output in the element coordinate system. The element coordinate system is established as follows:

- The element x-axis lies in the direction from G1 to G2.
- The element y-axis is perpendicular to the element x-axis, and the positive x-y quadrant contains G3.
- The element z-axis is normal to the plane of the element. The positive z direction is established by applying the right-hand rule to the ordering sequence of G1 through G3.

NX Nastran calculates forces and moments at the element centroid. It calculates stresses at distances Z1 and Z2 from the element reference plane. You specify Z1 and Z2 on the PSHELL entry).

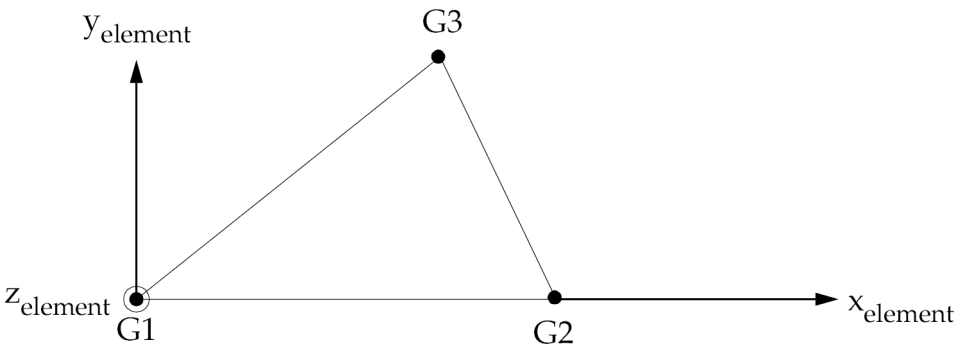


Figure 4-7. CTRIA3 Element Geometry and Element Coordinate System

CTRIA3 Format

The format of the CTRIA3 element entry is as follows:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
			T1	T2	T3				

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL or PCOMP entry.(Integer > 0; Default is EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. (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. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real)
Ti	Membrane thickness of element at grid points G1, G2, and G3. (Real ≥ 0.0 or blank, not all zero)

If you don't supply values for Ti, then the software sets the element corner thicknesses T1 through T3 equal to the value of T on the PSHELL entry.

See Also

- "CTRIA3" in the *NX Nastran Quick Reference Guide*.

The CQUADR and CTRIAR Elements

The CQUADR and CTRIAR elements are improved plate elements.

- CTRIAR is a triangular isoparametric plate element.
- CQUADR is a quadrilateral isoparametric plate element.

They take advantage of the normal rotational degrees of freedom (which have no stiffness associated with them in the standard plate elements) to provide improved membrane accuracy. The software computes a rotational stiffness about the normal to the element at the vertices, which is used in the formulation of the element stiffness. Consequently, this degree-of-freedom must not be constrained unless it occurs at a prescribed boundary.

When compared to the CQUAD4 and CTRIA3 elements, the CQUADR and CTRIAR are much less sensitive to high aspect ratios and values of Poisson's ratio near 0.5. For example, the CQUADR element provides better performance for modeling planar structures with in-plane loads than CQUAD4.

CQUADR and CTRIAR Guidelines and Limitations

In general, you should not mix elements of different formulations in your model. For example, you should not model part of a structure with the CQUAD4 and CTRIA3 elements and another part with the CQUADR and CTRIAR elements.

The CQUAD8 and CTRIA6 Elements

The CQUAD8 and CTRIA6 elements are similar to the CQUAD4 and CTRIA3 elements except they may also have mid-side grid points and curved edges.

- CQUAD8 is an isoparametric element with four corner and four mid-side grid points. Although they are useful for modeling singly-curved shells like a cylinder, CQUAD4 elements perform better on doubly-curved shells like a sphere.

- CTRIA6 is an isoparametric triangular element with three corner and three mid-side grid points that is used for transitioning meshes in regions with curvature.

Only use the CQUAD8 and CTRIA6 elements when at least one of the mid-side nodes is defined because these elements are excessively stiff when none of the mid-side nodes are defined. NX Nastran issues a warning message if no mid-side nodes are defined. If you do not need mid-side nodes, use the CQUAD4 or CTRIA3 elements instead.

CQUAD8 Format

The format of the CQUAD8 entry is as follows:

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	

Field	Contents
EID	Element identification number.
PID	Property identification number of a PSHELL or PCOMP entry.
G1, G2, G3, G4	Identification numbers of connected corner grid points.
G5, G6, G7, G8	Identification numbers of connected edge grid points (optional).
Ti	Membrane thickness of element at corner grid points.
THETA	Material property orientation angle in degrees.
MCID	Material coordinate system identification number.
ZOFFS	Offset from the surface of grid points to the element reference plane.

The first four grid points on the CQUAD8 entry define the corners of the element and are required. The last four grid points define the mid-side grid points, any of which you can omit (principally to accommodate changes in mesh spacing, but omitting mid-side nodes in general is not recommended). An edge grid point does not need to lie on the straight line segment joining adjacent corner points. However, it should be separated from the midpoint of the line by no more than 20% of the length of the line.

You can enter the properties of the CQUAD8 and CTRIA6 elements on the PSHELL entry. All the capabilities described for the CQUAD4 element apply to the CQUAD8 and CTRIA6 elements. The principal advantage of the these elements is that they may be more accurate in curved shell applications for the same number of degrees of freedom. The disadvantage is that with the addition of the mid-side node, they are more difficult to mesh for irregular shape structures.

CQUAD8 and CTRIA6 Element Coordinate System

The element coordinate system for CQUAD8 and CTRIA6 is defined implicitly by the location and connection order of the grid points (ξ, η) as is the case for the CQUAD4 and CTRIA3 elements with the following orientation:

1. The plane containing x_{elem} and y_{elem} is tangent to the surface of the element.
2. For the CQUAD8 element, x_{elem} and y_{elem} are obtained by doubly bisecting the lines of constant ξ and η.
3. For the CTRIA6 element, x_{elem} is tangent to the line of constant η.
4. x_{elem} increases in the general direction of increasing ξ and y_{elem} of η.

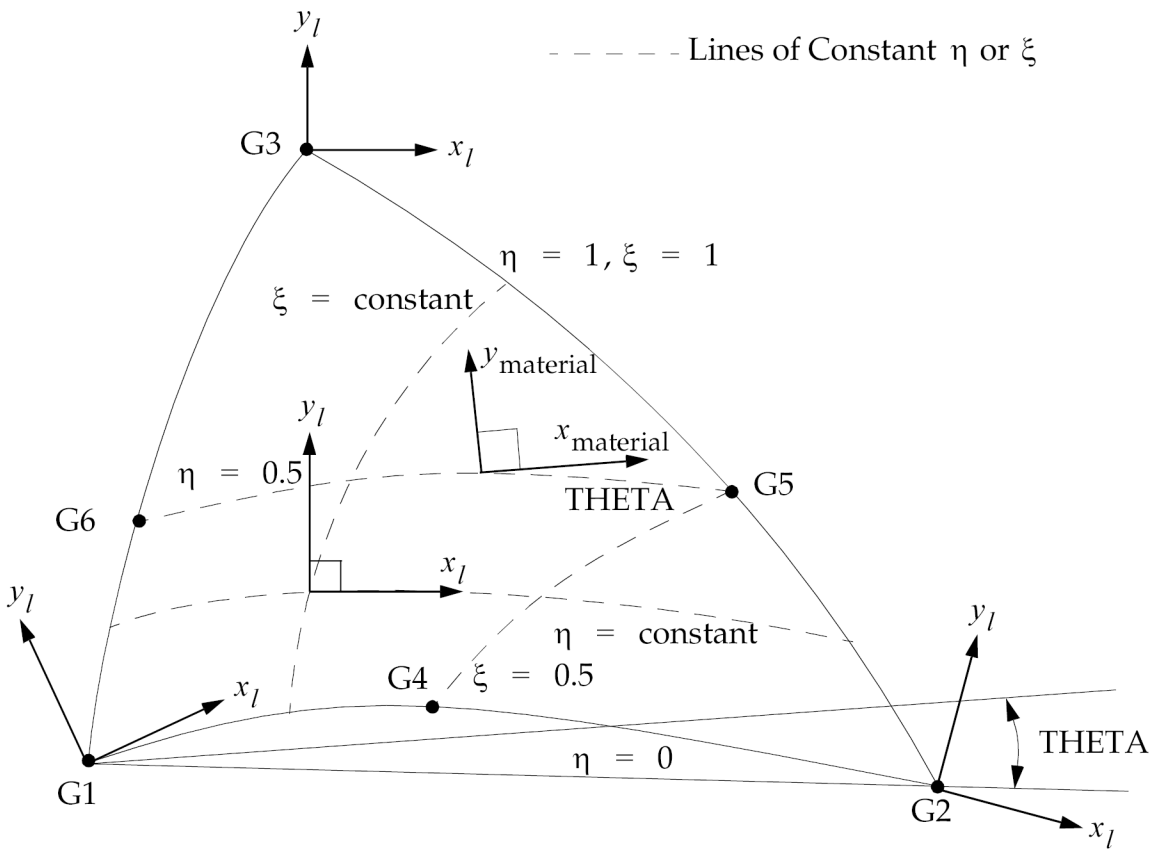


Figure 4-8. CTRIA6 Coordinate System

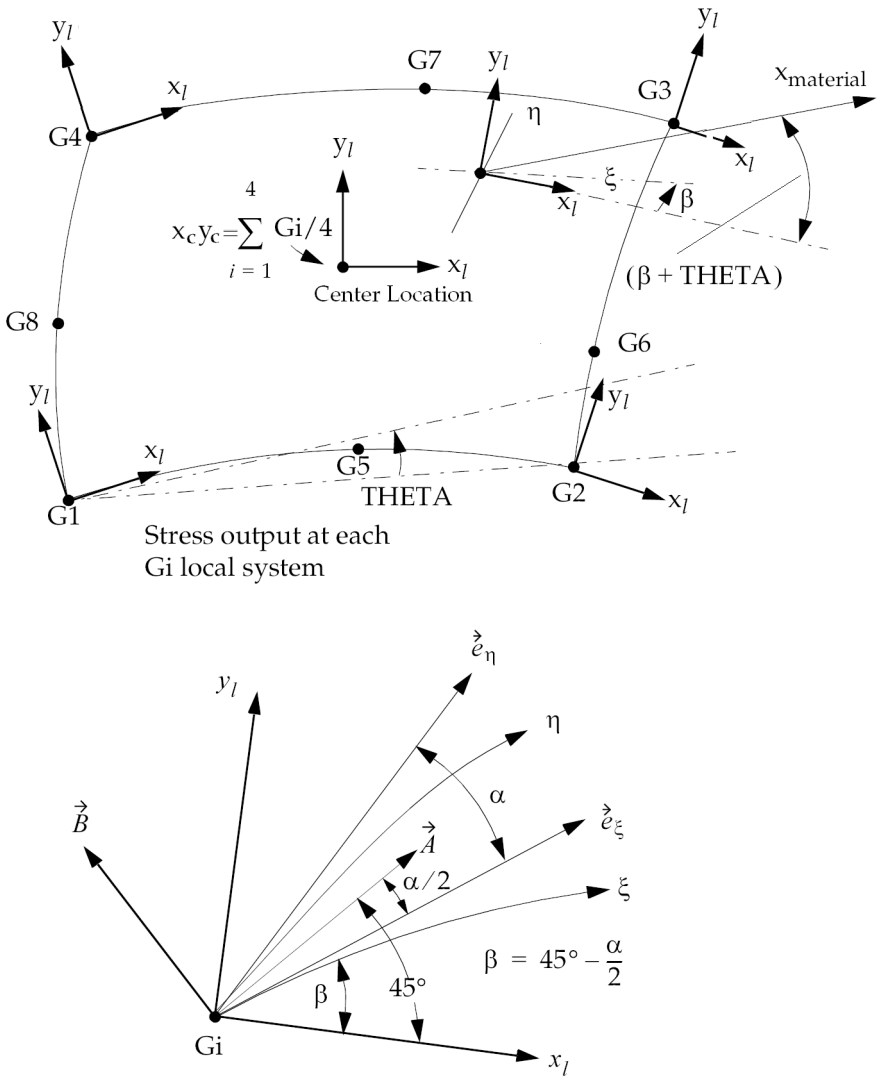


Figure 4-9. CQUAD8 Coordinate System

Understanding Plate and Shell Element Output

CQUAD4, CTRIAR, and CQUADR elements:	By default, element forces, stresses, and strains are only calculated at the centroid of these elements. You have the option to compute and output these quantities at the corner grid points using the corner option. For example, to obtain the corner stresses in addition to the centroidal stress, you should request: STRESS(CORNER) = n where n is a set of elements or STRESS(CORNER) = ALL for all the elements
CTRIA3 element	Element forces, stresses, and strains are only calculated at the centroid of the CTRIA3. The corner option is ignored for this element.

CTRIA6 and CQUAD8 elements	Element forces, stresses, and strains are always calculated at both the centroid of the element and at the corner locations for these elements. The center only option is ignored for these elements.

Figure 4-10. Element Force, Stress and Strain Output in Linear Solutions

Stresses are calculated at distances Z1 and Z2 from the element's reference plane (Z1 and Z2 are specified on the PSHELL property entry, and are normally specified as the surfaces of the plate; i.e., Z1, Z2 = ±thickness/2).

Hierarchy for Output Type

NX Nastran only supports one output type per run. You cannot mix CENTER and CORNER output types even for different output requests. To determine this output type, you should use the following hierarchy:

1. NX Nastran only considers requests made in the first subcase and above the subcase level when setting the output type. Subcases below the first are not considered for determining the output type (and the output type is then set to CENTER, i.e., the default).

2. The output type of the STRESS request in the first subcase determines the output type for STRESS, STRAIN, and FORCE for the entire run.

3. If there is no STRESS request in the first subcase, then the output type of the STRESS request above the subcase level determines the output type for STRESS, STRAIN, and FORCE for the entire run.

4. If there is no STRESS request above or in the first subcase, then the output type of the STRAIN request in the first subcase determines the output type for STRAIN and FORCE for the entire run.

5. If there is no STRAIN request in the first subcase, then the output type of the STRAIN request above the subcase level determines the output type for STRAIN and FORCE for the entire run.

6. If there is no STRAIN request above or in the first subcase, then the output type of the FORCE request in the first subcase determines the output type for FORCE for the entire run.

7. If there is no FORCE request in the first subcase, then the output type of the FORCE request above the subcase level determines the output type for FORCE for the entire run.

Forces and Moments

Figure 4-11 shows the positive directions of forces. Figure 4-12 shows the positive directions of moments. These diagrams can be helpful in understanding the element force output generated when using the FORCE (or ELFORCE) Case Control command in the Case Control section.

The forces shown are defined to be:

F_x , F_y	Normal (membrane) forces acting on the x and y faces per unit length.
F_{xy}	In-plane shear (membrane) force per unit length.
M_x , M_y	Bending moments on the x and y faces per unit length.
M_{xy}	Twisting moment per unit length.
V_x , V_y	Transverse shear forces acting on the x and y faces per unit length.

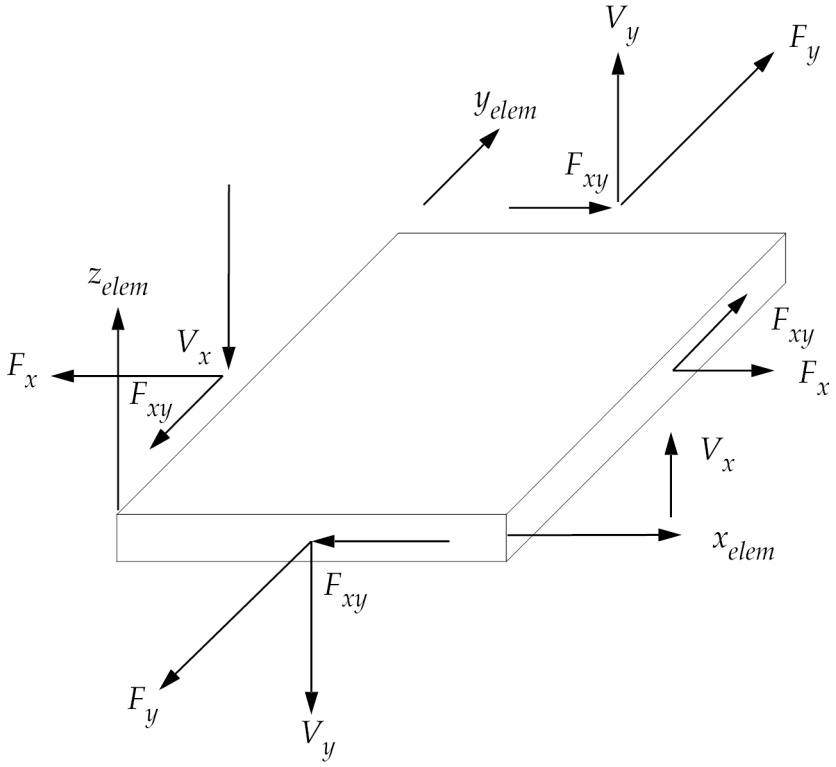


Figure 4-11. Forces in Shell Elements

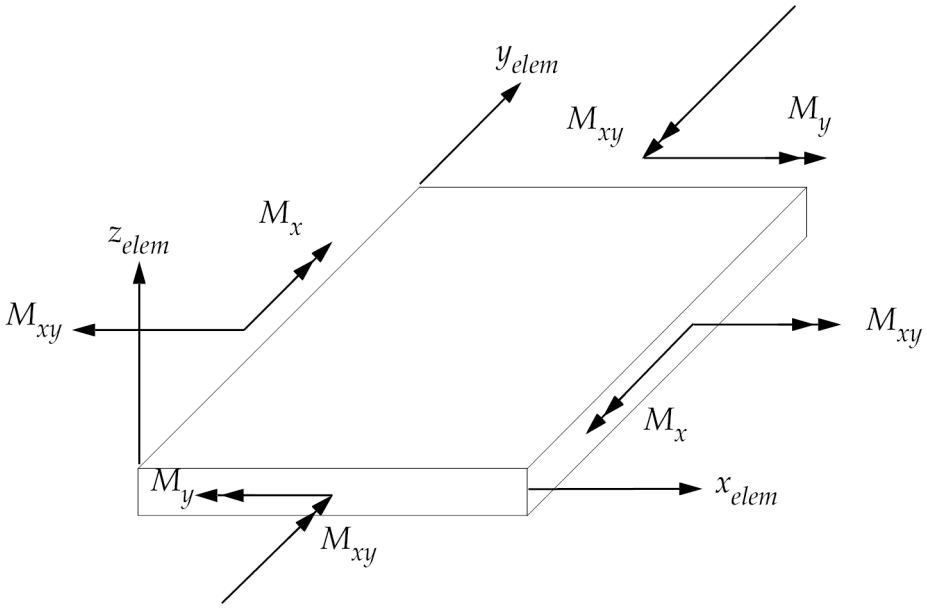


Figure 4-12. Moments in Shell Elements

Stresses

Figure 4-13 shows the stresses generated for a plate element. You can request that NX Nastran output the following stresses generated using the STRESS (or ELSTRESS) Case Control command:

- σ_x , σ_y – Normal stresses in the x- and y-directions.
- τ_{xy} – Shear stress on the x face in the y-direction.
- Major and minor principal stresses.
- Angle between the x-axis and the major principal direction. This angle is derived from σ_x , σ_y , and τ_{xy} .
- von Mises equivalent stress if you request STRESS(VONM) or maximum shear stress if you request STRESS(MAXS). These stresses are derived from σ_x , σ_y , and τ_{xy} .

NX Nastran calculates the stresses in the element coordinate system.

See Also

- "STRESS" in the *NX Nastran Quick Reference Guide*.

Note that :

- For the CQUAD4 and CTRIA3 elements, NX Nastran evaluates stresses at the centroid of the element.
- For the CQUAD4, you can request stress at the corner with the STRESS(CORNER) command.
- For the CQUAD8, CTRIA6, CQUADR, and CTRIAR elements, the stresses are evaluated at the centroid and at the vertices.

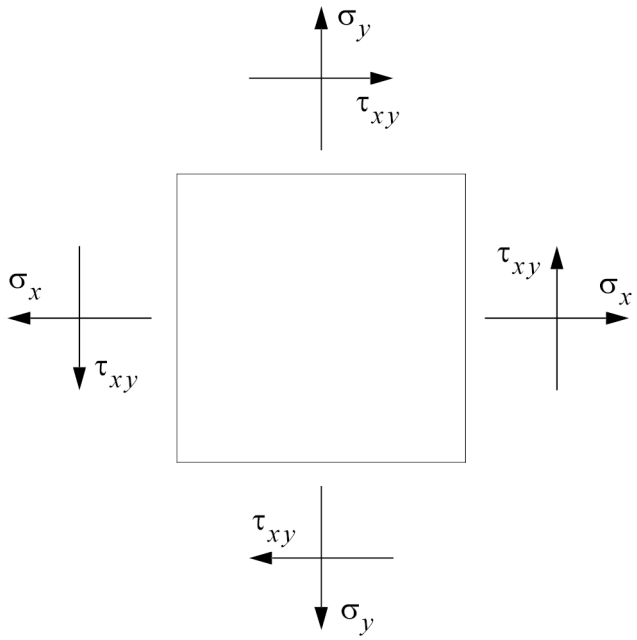


Figure 4-13. Stresses in Shell Elements

The von Mises equivalent stress for plane strain analysis is defined as follows:

$$\bar{\tau}_v = \left[1/2 \left\{ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 \right\} + 3\tau_{xy}^2 \right]^{1/2}$$

Equation 4-4.

For plane stress analysis, $\sigma_z = 0$. Only the normal stresses and shearing stresses are available in the complex form.

The von Mises equivalent strain is defined as:

$$\bar{\varepsilon}_v = \left[\frac{4}{9} (\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x \varepsilon_y) + \frac{1}{3} \gamma_{xy}^2 \right]^{1/2}$$

Equation 4-5.

where the strain components are defined as:

$$\varepsilon_x = \frac{\partial u}{\partial x}; \ \varepsilon_y = \frac{\partial v}{\partial y}; \ \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

Equation 4-6.

and the curvatures are defined as:

$$\chi_x = \frac{\partial^2 w}{\partial x^2}; \ \chi_y = \frac{\partial^2 w}{\partial y^2}; \ \chi_{xy} = 2\frac{\partial^2 w}{\partial x \partial y}$$

Equation 4-7.

The maximum shear stress is:

$$\hat{\tau} = \left[\left(\frac{\sigma_x - \sigma_y}{2} \right)^2 + \tau_{xy}^2 \right]^{1/2}$$

Equation 4-8.

The maximum shear strain is:

$$\hat{\gamma} = \left[(\varepsilon_x - \varepsilon_y)^2 + \gamma_{xy}^2 \right]^{1/2}$$

Equation 4-9.

The stresses are calculated at two specified points on the cross section. The distances to the specified points are given on the property entries. The default distance is one-half the thickness. The

positive directions for these fiber distances are defined according to the right-hand sequence of the grid points specified on the connection entry.

In addition, interpolated grid point stresses and mesh stress discontinuities are calculated in user-specified coordinate systems for grid points which connect the shell elements. Only real stresses are available at the grid points. Mesh stress discontinuities are available in linear static analysis only.

Grid point stresses are computed by:

$$\sigma_g = \sum_{e=1}^{N_e} W_{ge}\sigma_{ge}$$

Equation 4-10.

where σ_{ge}, a grid point stress component, multiplied by W_{ge}, the interpolation factor, and summed for all elements, N_e, connected to the grid point.

The stress discontinuity for one component and one element is:

$$\delta_{ge} = \sigma_{ge} - \sigma_g$$

Equation 4-11.

The discontinuity from all elements for one component is then obtained by:

$$\delta_g = \sqrt{\sum_{e=1}^{N_e} (W_{ge}\delta_{ge})^2}$$

Equation 4-12.

The total discontinuity at a grid point from all elements and all components N_c defines the error and is obtained by:

$$\delta_g = \sqrt{\frac{1}{N_c}\sum_{c=1}^{N_c} \delta_{gc}^2}$$

Equation 4-13.

δ_{gc} and δ_g are requested and printed with the GPSDCON Case Control command.

The total discontinuity at an element for one component over all of its grid points N_g is:

$$\delta_{ec} = \sqrt{\frac{1}{N_g}\sum_{g=1}^{N_g}\delta_g^2}$$

Equation 4-14.

and the error for an element is:

$$\delta_{e} = \sqrt{\frac{1}{N_g}\sum_{g=1}^{N_g}\delta_g^2}$$

Equation 4-15.

δ_{ec} and δ_e are requested and printed with the ELSDCON Case Control command.

See Also

- "ELSDCON" in the *NX Nastran Quick Reference Guide*.

Strains

You can use the STRAIN Case Control command to request strain output for a plate element. Deformation in the X-Y plane of the plate element at any point C at a distance z in the normal direction to plate middle surface is:

$$U = U_o + z\theta_y$$

Equation 4-16.

$$V = V_o - z\theta_x$$

Equation 4-17.

where *U,V* are the displacements in the element coordinate system, and θ_x, θ_y are the curvatures. The U_o and V_o are the plate midsurface displacements.

The strain-displacement-middle surface strain and curvatures relationship is given by:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial U_o}{\partial X} \\ \frac{\partial V_o}{\partial Y} \\ \frac{\partial U_o}{\partial Y} + \frac{\partial V_o}{\partial X} \end{Bmatrix} + z \begin{Bmatrix} \frac{\partial \theta_y}{\partial x} \\ -\frac{\partial \theta_x}{\partial y} \\ \frac{\partial \theta_y}{\partial y} - \frac{\partial \theta_x}{\partial x} \end{Bmatrix} = \begin{Bmatrix} \varepsilon_x^o \\ \varepsilon_y^o \\ \gamma_{xy}^o \end{Bmatrix} - z \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}$$

Equation 4-18.

where the ε^o's and χ's are the middle surface strains and curvatures, respectively.

You can request strain output as strains at the reference plane and curvatures or strains at locations Z1 and Z2. The following strain output Case Control command

STRAIN = n

or

STRAIN(STRCUR) = n

requests strains and curvatures at the reference plane. Similarly, the following strain output Case Control command:

```
STRAIN(FIBER) = n
```

requests strains at Z1 and Z2. The example problem in Listing 4-1 contains two identical subcases except for the strain output request format. The output is shown in Figure 4-14. The first and second lines of the strain output for the first subcase represents the mean strains and curvatures, respectively, at the reference plane. The first and second lines of the strain output for the second subcase represents the strains at the bottom (Z1) and top (Z2) fibers, respectively.

See Also

- "STRAIN" in the *NX Nastran Quick Reference Guide*.

In a linear static analysis, the strain output are total strain – mechanical strain plus thermal strain.

```
$  strain2.dat
SOL    101
TIME   5
CEND
SPC = 100
DISPLACEMENT = ALL
LOAD = 10
$
SUBCASE 1
   LABEL = MEAN STRAIN AND CURVATURE
   STRAIN        = ALL
$
SUBCASE 2
   LABEL = STRAIN AT FIBER LOCATIONS
   STRAIN(FIBER) = ALL
$
BEGIN BULK
$
CQUAD4  1       1       1       2       3       4
FORCE   10      2               10.     1.      0.      -1.
```

```
FORCE   10      3               10.     1.      0.      -1.
GRID    1               0.      0.      0.
GRID    2               10.     0.      0.
GRID    3               10.     10.     0.
GRID    4               0.      10.     0.
MAT1    1       190000.         .3
PSHELL  1       1       1.0     1
SPC1    100     123456  1       4
$
ENDDATA
```

Listing 4-1. Input File Requesting Strain Output

```
0     MEAN STRAIN AND CURVATURE                                                                  SUBCASE 1

                          S T R A I N S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )
  ELEMENT      STRAIN              STRAINS IN ELEMENT COORD SYSTEM            PRINCIPAL  STRAINS (ZERO SHEAR)
    ID.       CURVATURE          NORMAL-X       NORMAL-Y      SHEAR-XY       ANGLE         MAJOR           MINOR         VON MISES
0       1     .0               1.027233E-05  -2.311274E-06  -8.470329E-22     .0000    1.027233E-05   -2.311274E-06   7.734624E-06
           -1.000000E+00     -6.163397E-04   1.386764E-04   1.084202E-19    90.0000    1.386764E-04   -6.163397E-04   4.640774E-04
1                                                                            AUGUST   6, 1996  NX NASTRAN   8/ 2/04   PAGE    11

0     STRAIN AT FIBER LOCATIONS                                                                  SUBCASE 2

                          S T R A I N S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )
  ELEMENT      FIBRE               STRAINS IN ELEMENT COORD SYSTEM            PRINCIPAL  STRAINS (ZERO SHEAR)
    ID.       DISTANCE           NORMAL-X       NORMAL-Y      SHEAR-XY       ANGLE         MAJOR           MINOR         VON MISES
0       1   -5.000000E-01     -2.978976E-04   6.702695E-05   5.336308E-20    90.0000    6.702695E-05   -2.978976E-04   2.243041E-04
             5.000000E-01      3.184422E-04  -7.164950E-05  -5.505714E-20     .0000    3.184422E-04   -7.164950E-05   2.397733E-04
```

Figure 4-14. Strain Output for Plate Elements

SGAGE, CUBIC, CORNER, BILIN Strain/Stress Options

The STRAIN options CENTER, BILINEAR, CUBIC, and SGAGE have to do with the method of strain recovery for CQUAD4/CQUADR elements. The latter three options apply to recovery at corner locations with BILINEAR as the default corner option. Therefore the specification of CORNER option is the same as BILINEAR.

All the methods start by recovering strain at the element center and gauss points using the elemental strain matrix and the computed grid point displacements. From there, the various options control how these strains are extrapolated to corner locations for output. It is noted that stress results can be computed from strain using standard stress-strain relations. Thus the STRESS output control has the same recovery options as STRAIN. The explanation here describes the recovery for strain, but the analogy to stress is clear.

Except for the CENTER option which only returns strain at the center, all the other options return strain at the element center and corners. The strain at the center location is computed the same for all the options. The strain at corner locations is computed differently for the various corner options. BILINEAR uses a linear extrapolation method and is the more stable in most cases and is thus the default. The other corner options use higher order extrapolations in an attempt to be more accurate.

More details on the corner methods are as follows:

BILINEAR – This is the default corner option and its usage is interchangeable with the CORNER option. This option uses the element linear interpolation functions to extrapolate the strain at the gauss points to the strain at the nodes. In the example of a linear varying moment in a cantilever modeled with CQUAD4 shell elements the strain variation across each element is constant. Across

the length of the model, from the fixed point to the load application point, the strain will vary as a step function from element to element. This is because the CQUAD4 element has almost constant strain curvature, giving constant curvature at the gauss points, and when linear interpolated linearly yields constant strain over an element.

The discontinuity of strain from element to element can be minimized by refining the mesh. Nodal averaging of stress results, common in most post-processors, will also smooth out the results.

CUBIC – Is a corner option intended to smooth out the discontinuity of strain results between adjacent elements. Like the BILINEAR method it extrapolates strain to the corners using the element interpolation functions. Then it uses grid displacements and rotations to curve-fit a cubic equation that is used to adjust the linear corner strains. In the example of a linear varying moment in the cantilevered shell model, the grid point rotations will vary across the element so the curve fit gives the correct linear variation of strain curvature across the element, which translates to a linear varying stress. There is still some discontinuity of strain from element to element, but it is less than with the CORNER method.

Again, mesh refinement and nodal averaging can be used to minimize strain discontinuity.

SGAGE – This method is similar to the CUBIC. But in-plane strains and curvatures are calculated independently for the cubic equation. First strains are calculated in the u and v and diagonal u-v directions at each grid point. The state of in-plane strain at the grid point is calculated using rosette strain gauge equations. Grid strain curvatures are done similarly. In the example of a linear varying moment in the cantilevered shell model, a non-constant strain variation is obtained across each element, however the accuracy is not as good as the CUBIC method. The SGAGE method is not recommended for most cases.

CQUAD4 Example 1

An example using CQUAD4 elements to model a cantilevered flat plate is shown in Figure 4-15.

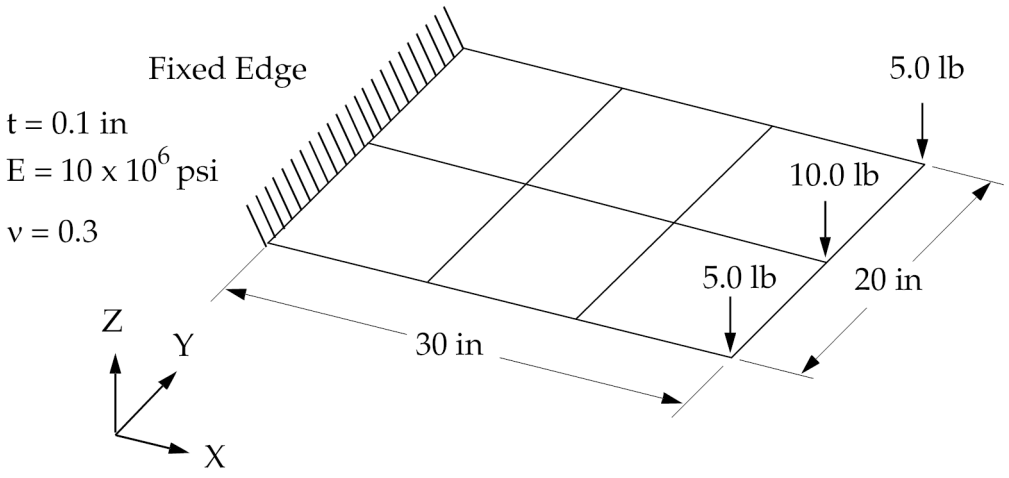


Figure 4-15. Cantilever Plate Model

The input file is shown in Listing 4-2.

```
$ FILENAME - PLATE2.DAT
$
BEGIN BULK
PARAM   AUTOSPC YES
PARAM   POST    0
$
GRID    1               0.0     -10.    0.0
GRID    2               10.     -10.    0.0
GRID    3               20.     -10.    0.0
GRID    4               30.     -10.    0.0
GRID    5               0.0     0.0     0.0
GRID    6               10.     0.0     0.0
GRID    7               20.     0.0     0.0
GRID    8               30.     0.0     0.0
GRID    9               0.0     10.     0.0
GRID    10              10.     10.     0.0
GRID    11              20.     10.     0.0
GRID    12              30.     10.     0.0
$
CQUAD4  1       1       1       2       6       5
CQUAD4  2       1       2       3       7       6
CQUAD4  3       1       3       4       8       7
CQUAD4  4       1       5       6       10      9
CQUAD4  5       1       6       7       11      10
CQUAD4  6       1       7       8       12      11
$
FORCE   1       12      0       -5.0    0.0     0.0     1.
FORCE   1       4       0       -5.0    0.0     0.0     1.
FORCE   1       8       0       -10.    0.0     0.0     1.
$
SPC1    1       123456  1       5       9
$
PSHELL  1       1       .1      1
$
MAT1    1       1.+7            .3
ENDDATA
```

Listing 4-2. Cantilever Plate Input File

The AUTOSPC parameter is used to constrain the rotational degrees-of-freedom (DOF) normal to the plate. The grid point singularity table output, as shown in Figure 4-16, is generated by PARAM,AUTOSPC. The rotational DOF (DOF 6 in this case) is removed from the *f* - set (free set) to the *s* - set (SPC set). The asterisk at the end of the lines indicates that the action was taken.

G R I D P O I N T S I N G U L A R I T Y T A B L E

POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	OLD USET UNION	NEW USET EXCLUSIVE	NEW USET UNION	
2	G	6	0.00E+00	B	F	SB	S	*
3	G	6	0.00E+00	B	F	SB	S	*
4	G	6	0.00E+00	B	F	SB	S	*
6	G	6	0.00E+00	B	F	SB	S	*
7	G	6	0.00E+00	B	F	SB	S	*
8	G	6	0.00E+00	B	F	SB	S	*
10	G	6	0.00E+00	B	F	SB	S	*
11	G	6	0.00E+00	B	F	SB	S	*
12	G	6	0.00E+00	B	F	SB	S	*

Figure 4-16. Parameter AUTOSPC Output

The displacement and stress output for the plate model are shown in Figure 4-17.

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	-1.395891E+00	-3.679435E-02	2.621396E-01	.0
3	G	.0	.0	-5.132846E+00	-4.282482E-02	4.666319E-01	.0
4	G	.0	.0	-1.013968E+01	-2.199761E-02	5.164736E-01	.0
5	G	.0	.0	.0	.0	.0	.0
6	G	.0	.0	-1.571300E+00	9.460242E-16	2.948987E-01	.0
7	G	.0	.0	-5.344157E+00	-1.331533E-15	4.418921E-01	.0
8	G	.0	.0	-1.024780E+01	-9.773031E-16	5.206971E-01	.0
9	G	.0	.0	.0	.0	.0	.0
10	G	.0	.0	-1.395891E+00	3.679435E-02	2.621396E-01	.0
11	G	.0	.0	-5.132846E+00	4.282482E-02	4.666319E-01	.0
12	G	.0	.0	-1.013968E+01	2.199761E-02	5.164736E-01	.0

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	-5.000000E-02	-1.500000E+04	-3.580141E+03	-6.687832E+02	-86.6598	-3.541109E+03	-1.503903E+04	1.361826E+04
	5.000000E-02	1.500000E+04	3.580141E+03	6.687832E+02	3.3402	1.503903E+04	3.541109E+03	1.361826E+04
2	-5.000000E-02	-9.000000E+03	-7.095209E+02	-1.350937E+02	-89.0667	-7.073201E+02	-9.002200E+03	8.670206E+03
	5.000000E-02	9.000000E+03	7.095209E+02	1.350937E+02	.9333	9.002200E+03	7.073201E+02	8.670206E+03
3	-5.000000E-02	-3.000000E+03	7.205607E+02	3.975338E+02	83.9688	7.625621E+02	-3.042001E+03	3.486400E+03
	5.000000E-02	3.000000E+03	-7.205607E+02	-3.975338E+02	-6.0312	3.042001E+03	-7.625621E+02	3.486400E+03
4	-5.000000E-02	-1.500000E+04	-3.580141E+03	6.687832E+02	86.6598	-3.541109E+03	-1.503903E+04	1.361826E+04
	5.000000E-02	1.500000E+04	3.580141E+03	-6.687832E+02	-3.3402	1.503903E+04	3.541109E+03	1.361826E+04
5	-5.000000E-02	-9.000000E+03	-7.095209E+02	1.350937E+02	89.0667	-7.073201E+02	-9.002200E+03	8.670206E+03
	5.000000E-02	9.000000E+03	7.095209E+02	-1.350937E+02	-.9333	9.002200E+03	7.073201E+02	8.670206E+03
6	-5.000000E-02	-3.000000E+03	7.205607E+02	-3.975338E+02	-83.9688	7.625621E+02	-3.042001E+03	3.486400E+03
	5.000000E-02	3.000000E+03	-7.205607E+02	3.975338E+02	6.0312	3.042001E+03	-7.625621E+02	3.486400E+03

Figure 4-17. Displacement and Stress Output

Small Deflection Assumption

New NX Nastran users are sometimes concerned that the displacement for each grid point deflection is exclusively in the Z-direction for this problem. Physically, you know that there's a displacement in the X-direction when the Z-displacement is as large as it is for this problem. However, linear analysis assumes small deflections, and as such, there's no deflection in the X-direction.

If the deflection of the plate is significant enough that the in-plane forces cannot be ignored, a nonlinear analysis may be warranted. Such is the case for the cantilever plate model where the deflections at the tip are over 5 inches. To show a comparison, this same problem was run using SOL 106, a nonlinear statics solution. A plot showing the linear and nonlinear results is given in Figure 4-18.

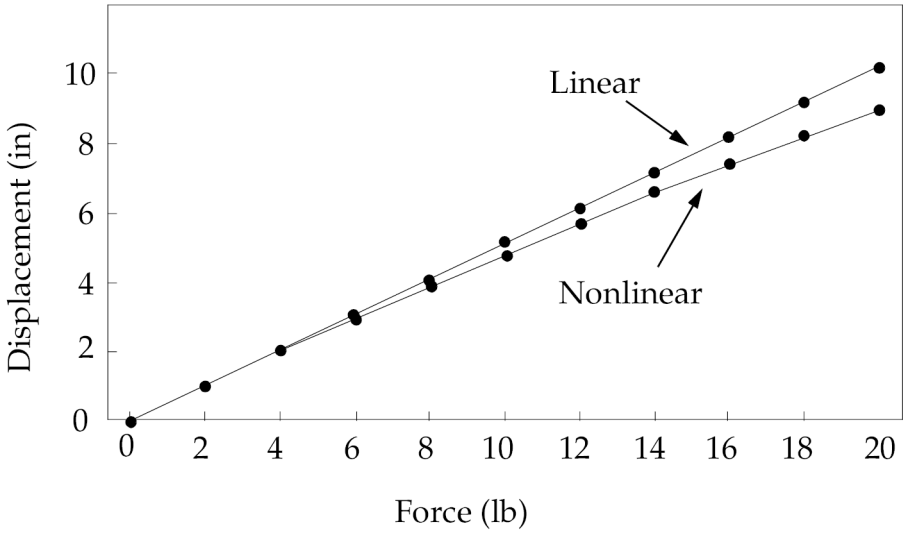


Figure 4-18. Nonlinear vs. Linear Results

This example is not intended to show you how to perform a nonlinear analysis; it serves only to remind you that linear analysis means small deflections and the superposition principle applies. The input file for the nonlinear run, “nonlin.dat”, is located on the delivery media if you wish to see how the nonlinear run is performed.

If you do not use the corner output, the stresses are output in the element coordinate system for the center of the element only. For this example, the maximum normal stress in the X—direction is 7500 psi. However, if you compute the stress at the fixed end using simple beam theory, the stress is 9000 psi. This discrepancy occurs because the 7500 psi stress is computed at a distance of 5 inches from the fixed edge. Although the example seems trivial, most users use a post-processor to view the results, as shown in Figure 4-19. The contour plot in the Figure 4-19 indicates the maximum stress as 7500 psi in the region close to the fixed edge. If you only look at the contour plot, as many people do, you can easily be misled. To help reduce interpretation errors, you can use the corner stress output. Grid point stresses and stress discontinuity checks are also available.

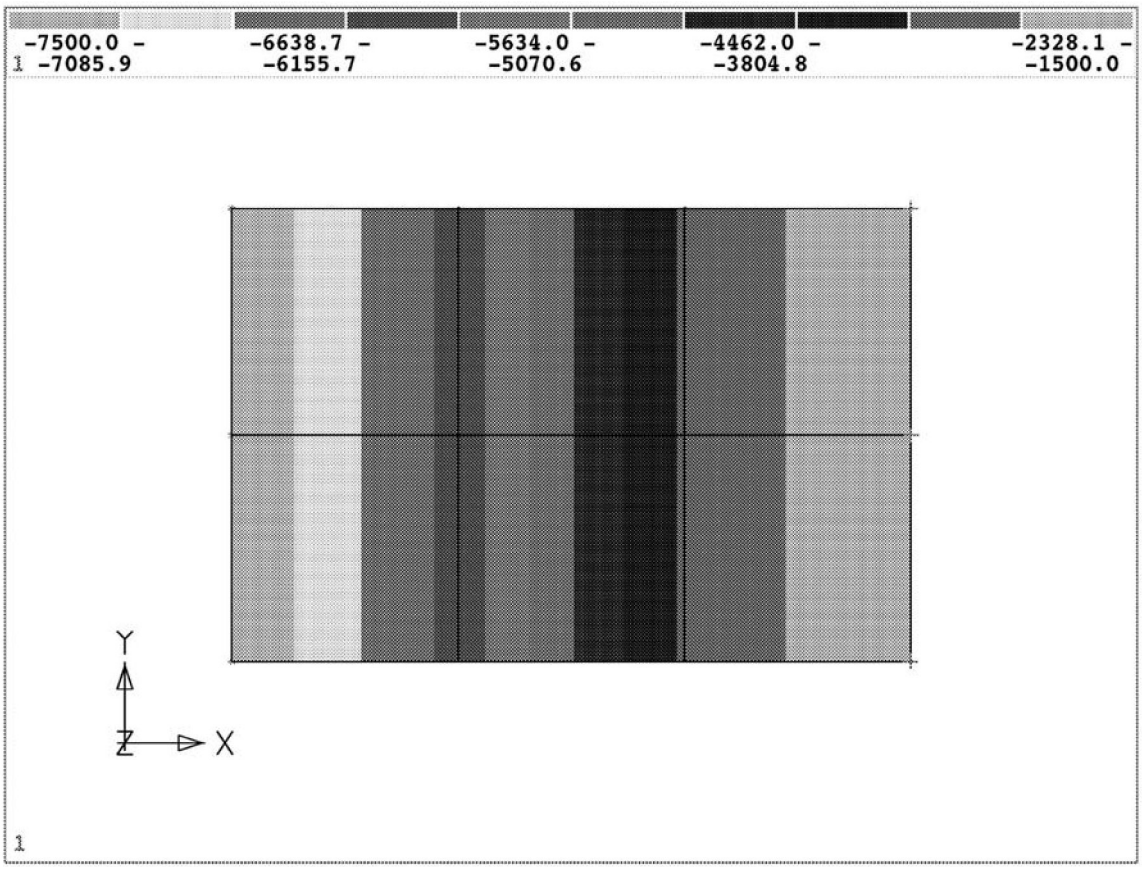


Figure 4-19. Cantilever Plate Stress Contour Plot

CQUAD4 Example 2

A 10 in x 10 in by 0.15 inch cantilevered plate is subjected to in-plane tensile loads of 300 lb_f and lateral loads of 0.5 lb_f at each free corner. Find the displacements, forces, and stresses in the plate. A single CQUAD4 element is used to model the plate as shown in Figure 4-20.

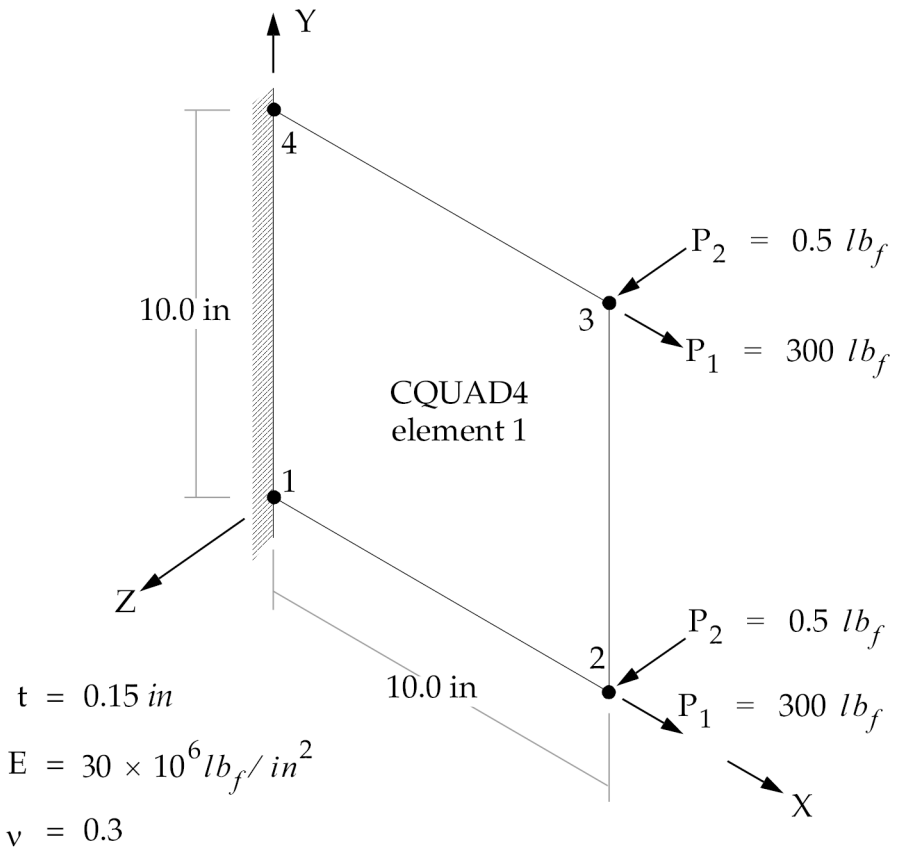


Figure 4-20. Cantilever Plate Example

The required Bulk Data entries are specified as follows:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	
GRID	1		0.	0.	0.		123456		
GRID	2		10.	0.	0.				
GRID	3		10.	10.	0.				
GRID	4		0.	10.	0.		123456		

CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
CQUAD4	1	5	1	2	3	4			

PSHELL	PID	MID1	T	MID2	$12I/T^3$	MID3	TS/T	NSM	
PSHELL	5	7	0.15	7		7			

MAT1	MID	E	G	NU	RHO	A	TREF	GE	
MAT1	7	30.E6		0.3					

The Case Control commands required to obtain the necessary output are as follows:

```
FORCE=ALL
DISP=ALL
STRESS=ALL
```

The grid point displacement output is shown in Figure 4-21.

1 CANTILEVER QUAD PLATE JULY 10, 2003 NX NASTRAN 07/10/2003 PAGE 11

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	0.0	0.0	0.0
2	G	1.301162E-04	2.927614E-05	3.737665E-03	9.460688E-05	-5.676413E-04	0.0
3	G	1.301162E-04	-2.927614E-05	3.737665E-03	-9.460688E-05	-5.676413E-04	0.0
4	G	0.0	0.0	0.0	0.0	0.0	0.0

Figure 4-21. Grid Point Displacement

Note the following when examining the deflections shown in Figure 4-21:

1. The maximum deflections of 3.738E-3 inches are due to the 0.5 lb_f lateral loads and occur at grid points 2 and 3 (the free edge), as expected.

2. The free edge deflections are identical since the structure and loadings are symmetric.

3. Grid points 1 and 4 have exactly zero displacement in all DOFs, since they were constrained to be fixed in the wall.

4. The lateral deflections occur in the T3 (+z) direction, which correspond with the direction of lateral loading. Note that these displacements are reported in the displacement coordinate system, not the element coordinate system.

5. The maximum lateral deflection of 3.738E-3 inches is much less than the thickness of the plate (0.15 in). Therefore, we are comfortably within the range of small displacement linear plate theory.

The CQUAD4 element force output is shown in Figure 4-22.

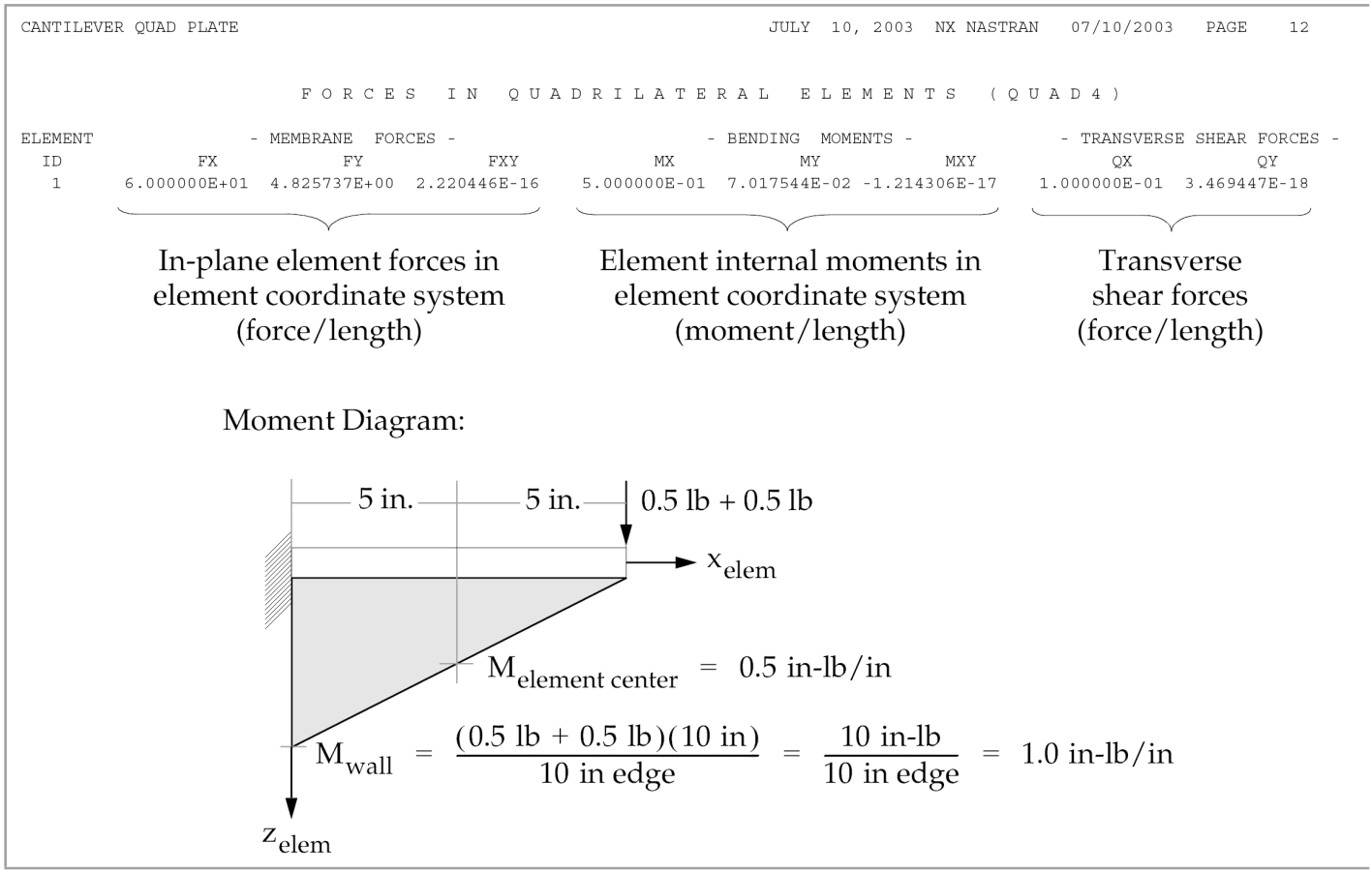


Figure 4-22. CQUAD4 Element Force and Moment Output

Note that numbers such as -1.214306E-17 (for bending moment MXY) are called “machine zeros” – they are zeros with slight errors due to computer numerical roundoff.

The CQUAD4 element stress output is shown in Figure 4-23.

1 CANTILEVER QUAD PLATE JULY 10, 2003 NX NASTRAN 7/10/2003 PAGE 14

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	PRINCIPAL STRESSES (ZERO SHEAR) MAJOR	MINOR	VON MISES
1	-7.500000E-02	5.333333E+02	5.088503E+01	-1.757853E-15	0.0000	5.333333E+02	5.088503E+01	5.097990E+02
	7.500000E-02	2.666667E+02	1.345813E+01	4.718447E-15	0.0000	2.666667E+02	1.345813E+01	2.601988E+02

Axial and bending stresses at the centroid of the CQUAD4 element are given by:

$$\sigma_{AXIAL} = \frac{P}{A} = \frac{300\ \text{lb} + 300\ \text{lb}}{(10\ \text{in})(0.15\ \text{in})} = 400\ \text{lb/in}^2$$

$$\sigma_{BENDING} = \frac{Mc}{I} = \frac{(.5\ \text{in-lb/in})(10\ \text{in edge})(.075\ \text{in})}{(10\ \text{in})(0.15\ \text{in})^3/12} = \pm 133.33\ \text{lb/in}^2$$

On the tensile surface,

$$\sigma_x^+ = \sigma_{AXIAL} + \sigma_{BENDING} = 400 + 133.33 = \underline{\underline{533.33\ \text{lb/in}^2}}$$

On the compressive surface,

$$\sigma_x^- = \sigma_{AXIAL} - \sigma_{BENDING} = 400 - 133.33 = \underline{\underline{266.67\ \text{lb/in}^2}}$$

The von Mises stress on the compressive surface is given by

$$\sigma_{von}^- = \sqrt{\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2} = \sqrt{(266.67)^2 - (266.67)(13.46) + (13.46)^2 + 3(4.718\text{E-}15)^2}$$

$$\sigma_{von}^- = 260.2\ lb_f/in^2$$

0

Figure 4-23. CQUAD4 Element Stress Output

Using the OMID Parameter to Output Shell Element Results

For the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 elements, you can use the PARAM,OMID,YES parameter to have NX Nastran output the element force, stress, and strain quantities in the material coordinate system in linear analyses. This capability is limited to element force, stress, and strain responses and provides output in the coordinate system defined using the THETA/MCID field on their associated bulk data entries. Both element center and element corner results are output in the material system.

You should use this capability with care because outputting element results in the material coordinate system can produce incorrect results in subsequent calculations that assume the element results are in the element coordinate system. By default, NX Nastran calculates the results for the shell elements in the element coordinate system. The element coordinate system is an artifact of the meshing option chosen to discretize the model and, typically, does not have a physical interpretation. On the other hand, the output of results in a specified coordinate system does have physical meaning

and it allows you to scan results across elements so you can ensure they have been output in a common coordinate system.

The parameter is applicable for all subcases and for static and dynamic analyses. You should not use PARAM,OMID,YES if grid point stresses are required or if you are going to examine your results in a post-processor. The calculation of grid point stresses is based on element stress results with the assumption that the element responses are in the element coordinate system. Post-processors assume element results are in the element coordinate system, thereby producing incorrect values if they are not in that system.

See Also

- "OMID" in the *NX Nastran Quick Reference Guide*.

Figure 4-24 contains the strain output from two different runs of the same model, one with PARAM,OMID,YES and the other with PARAM,OMID,NO. Differences are seen in the labeling that indicates how the NORMAL-X, NORMAL-Y, AND SHEAR-XY results are output and in the results themselves. In this case, the material axis is at a 45 degree angle with respect to the element axis and this value is reflected in the difference in the angle listed under the PRINCIPAL STRAINS heading. Similar differences can be seen in the stress output, whereas the force output has an additional label indicating that results are in the material coordinate system when OMID is set to YES. The standard force output does not include this label. (See Figure 4-25.)

S T R A I N S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRAINS IN MATERIAL COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRAINS (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	**4.380871E-04**	**3.387215E-04**	**8.973553E-04**	41.8406	8.398243E-04	-6.301572E-05	5.820263E-04
		5.000000E-02	**-4.380871E-04**	**-3.387215E-04**	**-8.973553E-04**	-48.1594	6.301572E-05	-8.398243E-04	5.820263E-04
1		-5.000000E-02	4.592682E-04	3.387215E-04	7.979897E-04	40.7048	8.025165E-04	-4.526860E-06	5.365264E-04
		5.000000E-02	-4.592682E-04	-3.387215E-04	-7.979897E-04	-49.2952	4.526860E-06	-8.025165E-04	5.365264E-04

a. Strain Output Fragment with OMID = YES (Key features in bold)

S T R A I N S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRAINS IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRAINS (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	**-6.027338E-05**	**8.370820E-04**	**9.936564E-05**	86.8406	8.398243E-04	-6.301573E-05	5.820264E-04
		5.000000E-02	**6.027338E-05**	**-8.370820E-04**	**-9.936564E-05**	-3.1594	6.301573E-05	-8.398243E-04	5.820264E-04
1		-5.000000E-02	-1.338338E-20	7.979897E-04	1.205468E-04	85.7048	8.025166E-04	-4.526860E-06	5.365264E-04
		5.000000E-02	1.338338E-20	-7.979897E-04	-1.205468E-04	-4.2952	4.526860E-06	-8.025166E-04	5.365264E-04

b. Strain Output Fragment with OMID=NO (Key features in bold)

Figure 4-24. Effect of Parameter OMID on Element Strain Output

SUBCASE 1

F O R C E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

RESULTS IN MATERIAL COORD SYSTEM

ELEMENT		- MEMBRANE FORCES -			- BENDING MOMENTS -			- TRANSVERSE SHEAR FORCES -	
ID	GRID-ID	FX	FY	FXY	MX	MY	MXY	QX	QY
1	CEN/4	.0	.0	.0	9.884681E+01	8.610762E+01	5.752278E+01	-1.243172E+01	2.000000E+02
1		.0	.0	.0	1.006119E+02	8.787272E+01	4.924231E+01	-1.243172E+01	2.000000E+02
7		.0	.0	.0	1.071273E+02	9.438809E+01	5.575769E+01	-1.243172E+01	2.000000E+02
8		.0	.0	.0	9.708171E+01	8.434253E+01	6.580325E+01	-1.243172E+01	2.000000E+02
5		.0	.0	.0	9.056634E+01	7.782716E+01	5.928787E+01	-1.243172E+01	2.000000E+02
2	CEN/4	.0	.0	.0	8.610762E+01	9.884681E+01	5.752278E+01	2.000000E+02	-1.243172E+01
2		.0	.0	.0	8.787272E+01	1.006119E+02	4.924231E+01	2.000000E+02	-1.243172E+01

Figure 4-25. Element Force Output Labeling with PARAM OMID=YES

You must keep track of the value of the OMID parameter when working with the .pch file because there is no labeling for the punch output. One suggestion is to include the value of this parameter in the job TITLE so that it appears at the head of the .pch file.

Guidelines and Limitations

The OMID parameter is limited to a subset of elements and has not been integrated with pre-processors and with grid point stress output that assumes that the results are in the element coordinate system. It is intended to provide a quick look at results that are in a consistent coordinate system so that one can easily and meaningfully scan these results across a number of elements. If these data are to be incorporated into design formulae, it removes the requirement that they first be transformed into a consistent coordinate system. Of course, the invariant principal stress/strain results have always provided mesh independent results.

The forces, stress, and strains in the material coordinate system may only be printed or punched and are not written to the .op2 or.xdb files. The usual element coordinate system results are written to the .op2 or .xdb file if they are requested (param,post,x).

Offsetting Shell Elements

You can offset the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR and CTRIAR elements relative to the mean plane of their connected grid points. There are three commonly used techniques to define these offsets:

- ZOFFS
- MID4
- RBAR

Note the following guidelines and recommendations for choosing a method:

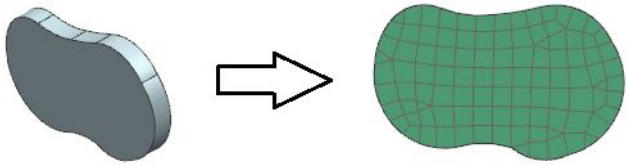
- Generally, you should use ZOFFS if you have a sufficiently fine mesh in the region where you want to define the offsets. If your mesh is more coarse, using RBARs to define offsets is generally more accurate.
- NX Nastran doesn’t modify the mass properties of an offset element to reflect the existence of the offset when you use the ZOFFS or MID4 method. If you need the weight or mass properties of an offset element for your analysis, use the RBAR method to create the offset.

- Regardless of which method you use to define the element offset, you must specify values for both MID1 and MID2 on the PSHELL entry that is referenced by element you are offsetting.

4.6 Plane Stress and Plane Strain Elements

- **Plane Stress Elements**

  Plane stress elements are two-dimensional elements with membrane only stiffness and in-plane loading. The plane stress element idealization has zero stress in the thickness direction. These elements represent structures that are thin relative to their lateral dimensions and planar.



  The entries CPLSTS4, CPLSTS8, CPLSTS3, and CPLSTS8 are plane stress elements.

  - They must be defined in either the XZ or the XY plane of the basic coordinate system. They must be in the XZ plane for SOL 601.
  - They are supported in all linear solutions expect cyclic solutions 114, 115, 116, 118 and aero solutions 144, 145, 146. They are supported in the nonlinear solution 601. They behave linearly in solutions 106 and 129.
  - All loads must be in-plane. The PLOADE1 entry can be used to define edge loads.
  - Properties are defined on the PPLANE entry when linear materials are applied, or the PLPLANE entry for hyperelastic (SOL 601 only). See “Understanding the MAT3 Bulk Entry” in the User’s Guide for orthotropic material information.
  - 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.

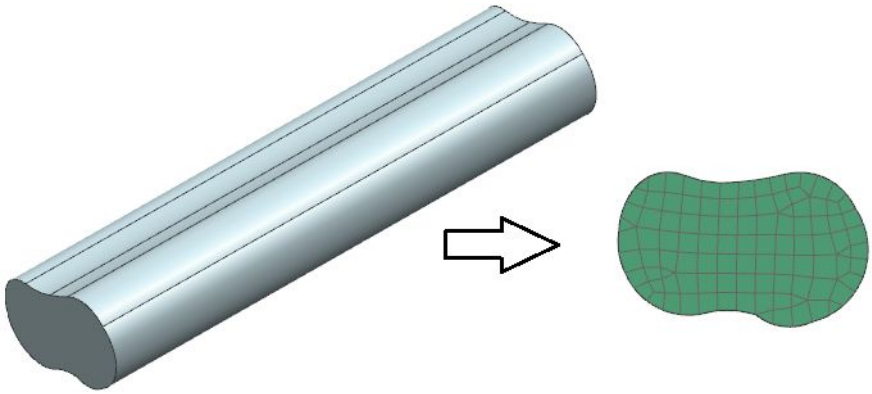
  The elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR can optionally be modeled as plane stress elements.

  - The associated PSHELL must only have MID1 defined.
  - The elements can be defined in any plane, but all must be defined in a consistent plane.

- All loads must be defined in-plane.
- They are supported in all solutions except SOLs 601 and 701.
- These elements have a slightly different stiffness formulation than the CPLSTSi elements. As a result, results can be slightly different.

- **Plane Strain Elements**

  Plane strain elements are two-dimensional elements with membrane only stiffness and in-plane loading. The plane strain element idealization has zero strain in the thickness direction. These elements represent structures that are very thick relative to their lateral dimensions.



  The entries CPLSTN4, CPLSTN8, CPLSTN3, and CPLSTN6 are plane strain elements.

  - They must be defined in either the XZ or the XY plane of the basic coordinate system. They must be in the XZ plane for SOL 601.
  - They are supported in all linear solutions expect cyclic solutions 114, 115 ,116, 118 and aero solutions 144, 145, 146. They are supported in the nonlinear solution 601. They behave linearly in solutions 106 and 129.
  - All loads must be in-plane. The PLOADE1 entry can be used to define edge loads.
  - Properties are defined on the PPLANE entry when linear materials are applied, or the PLPLANE entry for hyperelastic (SOL 601 only). See “Understanding the MAT3 Bulk Entry” in the User’s Guide for orthotropic material information.
  - 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.

The elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR can optionally be modeled as plane strain elements.

- The associated PSHELL must have MID2=-1 and MID1 must reference a MAT1 entry.
- The elements can be defined in any plane, but all must be defined in a consistent plane.
- They are supported in all solutions except SOL 701. In SOL 601, they must be defined on the XY plane.
- Only in-plane load components are applied to the element.
- In-plane loads 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.
- These elements have a slightly different stiffness formulation than the CPLSTNi elements. As a result, results can be slightly different.

CPLSTS and CPLSTN element stiffness

The formulation of the CPLSTS and CPLSTN elements provides for direct elastic stiffness for in-plane displacement DOF only. Consequently, the out-of-plane displacement DOF and all three rotational DOF have zero stiffness. This zero stiffness results in a singular stiffness matrix, which will prevent NX Nastran from solving your model.

To avoid this problem, you can constrain the out-of-plane displacement and rotational DOF either manually with an SPC bulk entry or automatically with the AUTOSPC parameter. If using the SPC method, ensure that you do not constrain either of the two in-plane displacement DOF that have stiffness.

See Also

- “Single-Point Constraints” in the *NX Nastran User’s Guide*.
- “Automatically Applying Single-Point Constraints” in the *NX Nastran User’s Guide*.

Plane stress and plane strain element output

The plane stress and plane strain elements support output of stress, strain, and grid point forces at the element center or corners as follows:

Element	Available Output
CPLSTN3, CPLSTS3	Stresses, strains, and grid point forces at the center only.
CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS4, CPLSTS6, and CPLSTS8	Stresses, strains, and grid point forces at the center and corners.
CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR modeled as plane stress or plane strain	See Understanding Plate and Shell Element Output.

Figure 4-26. Force, Stress and Strain Output in Linear Solutions

To request stress output, include a STRESS (or ELSTRESS) case control command. To request strain output, include a STRAIN case control command. To request grid point force output, include a GPFORCE case control command.

Stresses

Figure 4-27 shows the positive sense of stresses acting on a plane stress or plane strain element. The stresses shown are defined as follows:

σ_x, σ_z	Normal (membrane) stresses acting on the x- and z-faces per unit length.
τ_{xz}	In-plane shear stress per unit length.

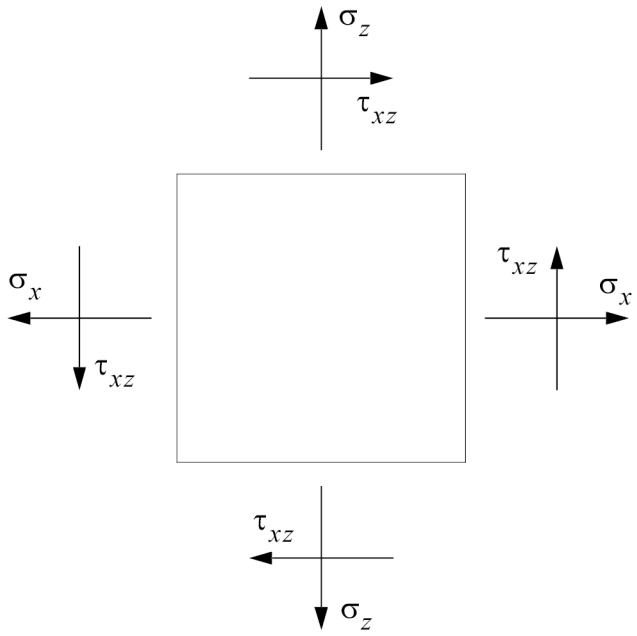


Figure 4-27. Stresses in Plane Stress and Plane Strain Elements

For plane stress elements, the normal stress acting through the thickness of the element is zero by definition. For the plane strain elements, the normal stress acting through the thickness of the element, σ_y, is typically nonzero. The normal stress acting through the thickness of the element is output by default when a STRESS case control command is used.

For both plane stress and plane strain elements, the von Mises stress is calculated and is output by default when a STRESS case control command is used. The von Mises stress is calculated from the stress components as follows:

$$\bar{\tau}_v = \left[1/2 \left\{ (\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 \right\} + 3\tau_{xy}^2 \right]^{1/2}$$

Equation 4-19.

Strains

Strain components are calculated from the in-plane nodal displacements, *u* and *w*, using the strain-displacement relations for plane elasticity:

$$\varepsilon_x = \frac{\partial u}{\partial x}; \quad \varepsilon_z = \frac{\partial w}{\partial z}; \quad \gamma_{xz} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}$$

Equation 4-20.

For plane stress elements, the normal stress acting through the thickness of the element is zero by definition. However, the normal strain acting through the thickness of the element, ε_y, is typically

nonzero due to Poisson's effect. The normal strain acting through the thickness of the element is output by default when a STRAIN case control command is used.

For plane strain elements, the normal strain acting through the thickness of the element is zero by definition.

For both plane stress and plane strain elements, the von Mises strain is calculated and is output by default when a STRAIN case control command is used. The von Mises strain is calculated from the strain components as follows:

$$\bar{\varepsilon}_v = \left[\frac{4}{9}(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x\varepsilon_y) + \frac{1}{3}\gamma_{xy}^2\right]^{1/2}$$

Equation 4-21.

In a linear static analysis, the strain output represents the total strain. That is, the mechanical strain plus the thermal strain.

Chapter 5: 3D Elements

- *Introduction to 3-Dimensional Elements*
- *Solid Elements (CTETRA, CPENTA, CPYRAM, CHEXA)*
- *Three-Dimensional Crack Tip Element (CRAC3D)*
- *Axisymmetric Solid Elements (CTRAX3, CQUADX4, CTRAX6, CTRIAX6, CTRIAX, CQUAD8, CQUADX)*

5.1 Introduction to 3-Dimensional Elements

You can use three-dimensional elements, commonly referred to as solid elements, to model structures that can't be modeled using beam or plate elements. For instance, a solid element is used to model an engine block because of the block's three-dimensional nature. If however, you're creating a model of the automobile hood, the best choice is one of the plate elements.

This chapter describes the following elements:

- CHEXA, CPENTA, CPYRAM, CTETRA
- CRAC3D (three-dimensional crack tip element)
- CTRIAX6, CTRAX3, CQUADX4, CTRAX6, CQUADX8 (axisymmetric elements that are used for axisymmetric analysis only)

Solid elements have only translational degrees of freedom. No rotational DOF are used to define the solid elements.

5.2 Solid Elements (CTETRA, CPENTA, CPYRAM, CHEXA)

NX Nastran includes four different solid polyhedron elements which are defined on the following bulk entries:

1. CTETRA — Four-sided solid (tetrahedral) element with 4 to 10 grid points
2. CPENTA — Five-sided solid (wedge) element with 6 to 15 grid points
3. CPYRAM — Five-sided solid (pyramid) element with 5 to 13 grid points
4. CHEXA — Six-sided solid (brick) element with 8 to 20 grid points

These elements differ from each other primarily in the number of faces and in the number of connected grid points. You can use these elements with all other NX Nastran elements except the axisymmetric elements. Connections are made only to displacement degrees-of-freedom at the grid points.

The CHEXA element is the most commonly used solid element in the NX Nastran element library. The CPENTA , CPYRAM, and CTETRA elements are used mainly for mesh transitions and in areas where the CHEXA element is excessively distorted.

Solid elements are commonly used to model an isotropic continuum for structural and thermal analyses using MAT1 and MAT4 bulk entries, respectively. However, solid elements can also be used to model:

- Anisotropic (MAT9) and orthotropic (MAT11) structural behavior, and anisotropic (MAT5) thermal behavior.
- Material nonlinear structural behavior including plasticity, nonlinear elasticity, hyperelasticity, and creep. The CPYRAM element is not supported for hyperelasticity.
- Geometric nonlinearity where the element stiffness must reformulate.
- Linear buckling where differential stiffness is calculated.

- Structural laminates using the PCOMPS bulk entry with CHEXA and CPENTA elements.
- Fluids for coupled fluid-structural analysis using MAT10 bulk entries.

Structural mass is calculated for all solid elements. The default mass procedure is lumped mass. You can request the coupled mass procedure by specifying PARAM,COUPMASS.

Defining Properties for Solid Elements

You can use the PSOLID, PLSOLID, and PCOMPS bulk entries to define the properties of solid elements.

- PSOLID bulk entries are used for most typical structural and thermal applications. PSOLID bulk entries can be referenced by all four solid elements. The PSOLID entry allows you to define a material coordinate system that can be used to define material properties and obtain stresses. Field 7 of the PSOLID entry provides you with some control over the integration technique used for the element. The CHEXA and CPENTA elements are modified isoparametric elements that use selective integration points for different components of strain. In addition to the standard isoparametric integration, there are two different networks of integration points available, depending on whether the element has midside nodes.
- PLSOLID bulk entries are only used to define hyperelastic materials. PLSOLID bulk entries can only be referenced by CHEXA, CPENTA, and CTETRA elements.
- PCOMPS bulk entries are only used to define solid element laminates. PCOMPS bulk entries can only be referenced by CHEXA and CPENTA elements.

See Also

- PSOLID in the *NX Nastran Quick Reference Guide*.
- PLSOLID in the *NX Nastran Quick Reference Guide*.
- PCOMPS in the *NX Nastran Quick Reference Guide*.

Solid Element Integration Types

Three types of integration are available for the CPENTA and CHEXA elements:

- Reduced shear integration with bubble functions
- Reduced shear integration without bubble functions
- Standard isoparametric integration

Only standard isoparametric integration is available for CTETRA elements. CPYRAM elements use Bedrosian integration.

If multiple types of integration are available for a given element, you can optionally override the default integration type by specifying an alternative integration type on the PSOLID entry.

Reduced shear integration minimizes shear locking and won't cause zero energy modes. If you select reduced shear integration with bubble functions, it minimizes Poisson's ratio locking which occurs in nearly incompressible materials and in elements under bending. In plastic analysis, bubble functions

are necessary to reduce locking caused by the plastic part in the material law. Therefore, reduced shear integration with bubble functions is the default option for the CPENTA and CHEXA elements. However, using bubble functions requires more computational effort.

In standard isoparametric integration, you can change the number of Gauss points or integration network to under integrate or over integrate the solid elements. Under integration may cause zero energy modes and over integration results in an element which may be too stiff. Standard isoparametric integration is more suited to non-structural problems.

Available Stress and Strain Output for Solid Elements

By default, the stress output for the solid elements is at the center and at each of the corner points. If no midside nodes are used, you may request the stress output at the Gauss points instead of the corner points by setting Field 6 of the PSOLID entry to “GAUSS” or 1.

The following stresses and strains are output upon request:

- Normal: σ_x , σ_y , σ_z , and ε_x , ε_y , ε_z
- Shear: τ_{xy} , τ_{yz} , τ_{zx} , and γ_{xy} , γ_{yz} , γ_{zx}
- Principal with magnitude and direction
- Mean pressure: $p_o = -\frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$

  Note

  Average normal stress and mean pressure are the same except that they have opposite signs.

- Octahedral shear stress:

$$\bar{\tau} = \frac{1}{3}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\tau_{yz}^2 + 6\tau_{zx}^2 + 6\tau_{xy}^2]^{1/2}$$

Equation 5-1.

or the von Mises equivalent (effective) stress:

$$\bar{\tau}_v = \left(\frac{3}{\sqrt{2}}\right)\bar{\tau}_o$$

Equation 5-2.

- Octahedral shear strain:

$$\bar{\varepsilon}_o = \left[\frac{1}{9}\left\{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2\right\} + \frac{1}{6}(\gamma_{xy}{}^2 + \gamma_{yz}{}^2 + \gamma_{zx}{}^2)\right]^{1/2}$$

Equation 5-3.

or the von Mises equivalent strain:

$$\bar{\varepsilon}_v = \left[\frac{2}{9}\left\{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2\right\} + \frac{1}{3}(\gamma_{xy}{}^2 + \gamma_{yz}{}^2 + \gamma_{zx}{}^2)\right]^{1/2}$$

Equation 5-4.

where the engineering strain components are defined as:

$$\varepsilon_x = \frac{\partial u}{\partial x};\ \varepsilon_y = \frac{\partial v}{\partial y};\ \varepsilon z = \frac{\partial w}{\partial z}$$

Equation 5-5.

$$\gamma_{xy} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right);\ \gamma_{yz} = \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right);\ \gamma_{zx} = \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)$$

Equation 5-6.

In nonlinear non-hyperelastic (small strain) analysis, engineering strain is output. In hyperelastic (large strain) analysis, logarithmic strain is output.

See Also

- Fully Nonlinear Hyperelastic Elements in the *NX Nastran Basic Nonlinear Analysis User's Guide*.

By default, the stresses and strains are evaluated in the basic coordinate system at each of the corner points and the centroid of the element. The stresses and strains may also be computed in the material coordinate system as defined in the CORDM field on the PSOLID entry. In addition, interpolated grid point stresses and mesh stress discontinuities are calculated in user-specified coordinate systems for grid points which connect solid elements. Only real stresses are available at the grid points. Mesh stress discontinuities are available in linear static analysis only.

See Also

- Introduction to Element Data Recovery in the *NX Nastran User's Guide*.

Solid elements contain stiffness only in the translation degrees of freedom at each grid point. Similar to the normal rotational degrees of freedom for the CQUAD4, you should be aware of the potential singularities due to the rotational degrees of freedom for the solid elements.You may either constrain the singular degrees of freedom manually or you can let NX Nastran automatically identify and constrain them for you using the AUTOSPC parameter. The parameter K6ROT doesn't affect solid

elements. Also any combination of the solid elements with elements that can transmit moments require special modeling. There are special considerations for mesh transitions.

See Also

- Creating Mesh Transitions in the *NX Nastran User's Guide*.
- Automatically Applying Single-point Constraints in the *NX Nastran User's Guide*.

Six-Sided Solid Element (CHEXA)

While the CHEXA element is recommended for general use, the CHEXA's accuracy degrades when the element is skewed. In some modeling situations, it has superior performance to other 3-D elements.

The CHEXA has eight corner grid points and up to twenty grid points if you include the twelve optional midside grid points. NX Nastran calculates element stresses (σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{zx}) at the element's center and Gauss points. These stresses are extrapolated to obtain the stresses at the corner grid points. The element's connection geometry is shown in Figure 5-1.

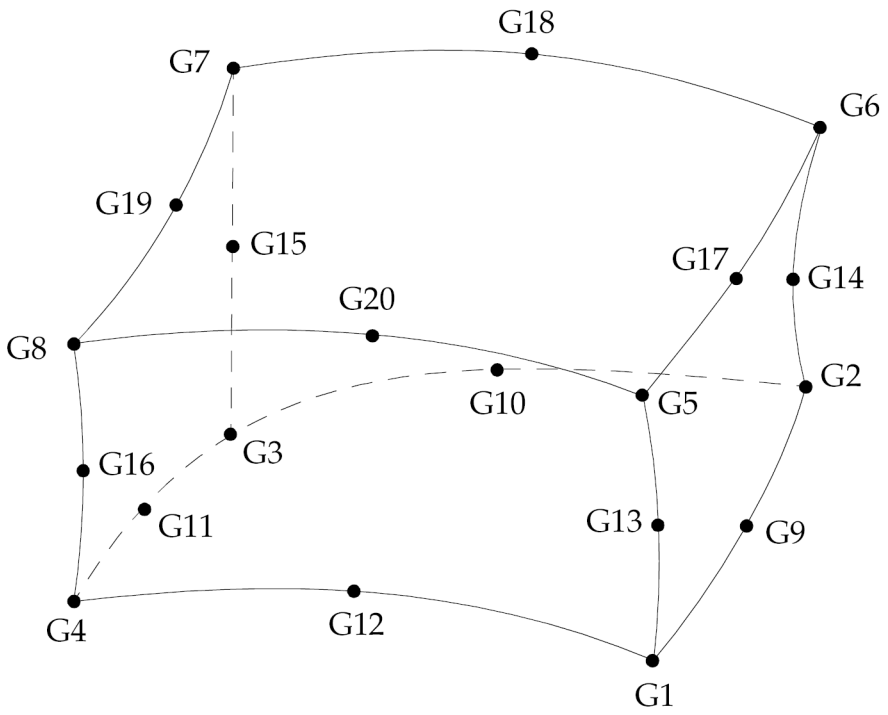


Figure 5-1. The CHEXA Element

You must always observe the ordering of the grid point IDs on the connectivity bulk data entries. For example, grid point G12 must lie between grid points G1 and G4. Deviation from the ordering scheme will result in a fatal error or incorrect answers. The midside grid points don't need to lie on a straight line between their respective corner grid points. However, you should try to keep the element's edges reasonably straight to avoid highly distorted elements.

You can delete any or all of the midside nodes for the CHEXA element. However, it is recommended that if midside nodes are used, then all midside grid points be included. Use the 8-noded or the 20-noded CHEXA in areas where accurate stress data recovery is required.

The CHEXA element coordinate system is shown in Figure 5-2.

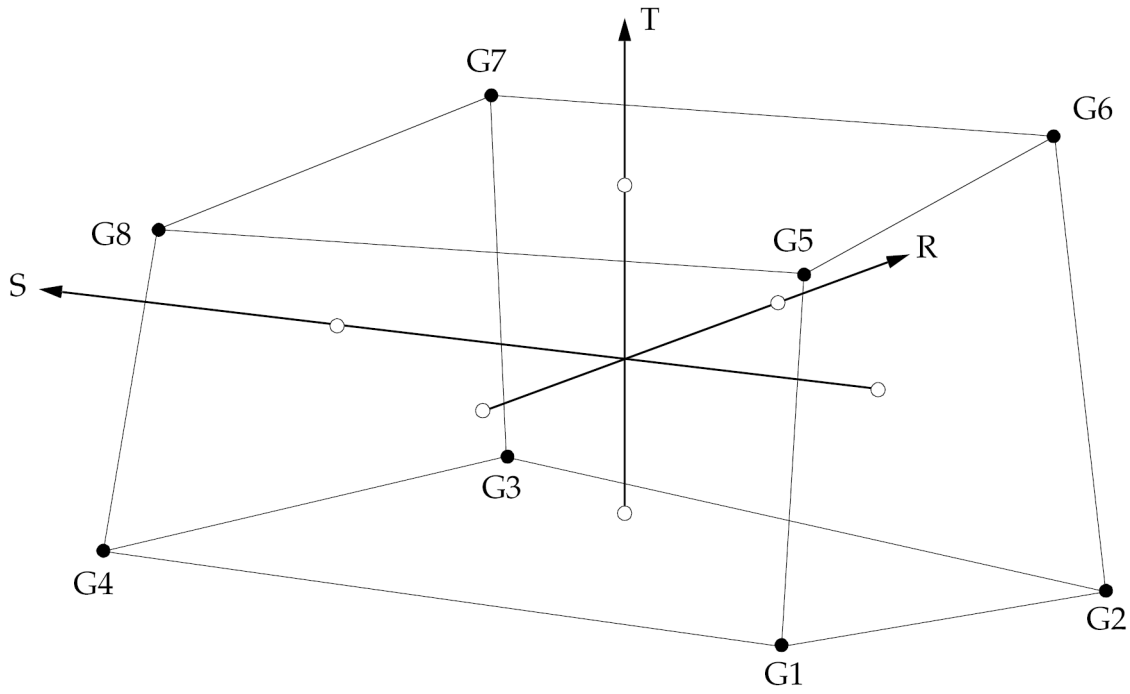


Figure 5-2. CHEXA Element Coordinate System

The CHEXA element coordinate system is defined in terms of vectors R, S, and T which join the centroids of opposite faces.

- The R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.
- The S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.
- The 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 three 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.

The R, S, and T vectors are not, in general, orthogonal to each other. They're used to define a set of orthogonal vectors R', S', and T' by performing an eigenvalue analysis. The element's x-, y-, and z-axes are then aligned with the same element faces as the R', S', and T' vectors. Because the software doesn't orient the RST vectors by the grid point IDs, a small perturbation in the geometry doesn't cause a drastic change in the element coordinate system.

CHEXA Format

The format of the CHEXA element entry 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			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 or blank)

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. The midside nodes, G9 to G20, are optional. If the ID of any midside node 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. Components of stress are output in the material coordinate system. The second continuation entry is optional.

See Also

- CHEXA in the *NX Nastran Quick Reference Guide*.

Five-Sided Solid Element (CPENTA)

The CPENTA element is commonly used to model transitions from solids to plates or shells. If the triangular faces are not on the exposed surfaces of the shell, excessive stiffness can result.

The CPENTA element uses from six to fifteen grid points (six with no midside grid points; up to fifteen using midside grid points). NX Nastran calculates element stresses (σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, and τ_{zx}) at the element's center and Gauss points. These stresses are extrapolated to obtain the stresses at the corner grid points. The element's connection geometry is shown in Figure 5-3.

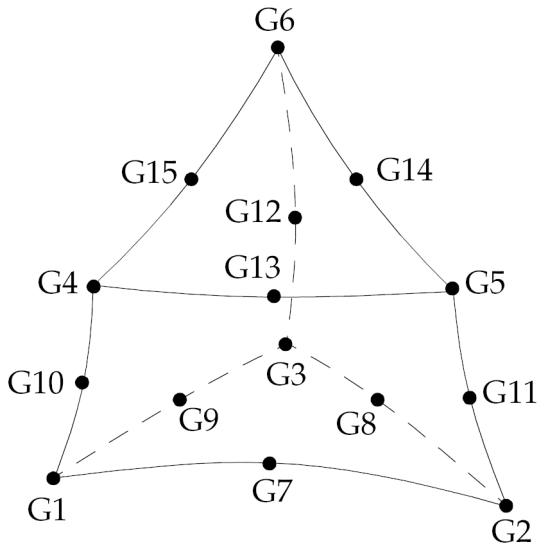


Figure 5-3. CPENTA Element Connection

The CPENTA element coordinate system is shown in Figure 5-4.

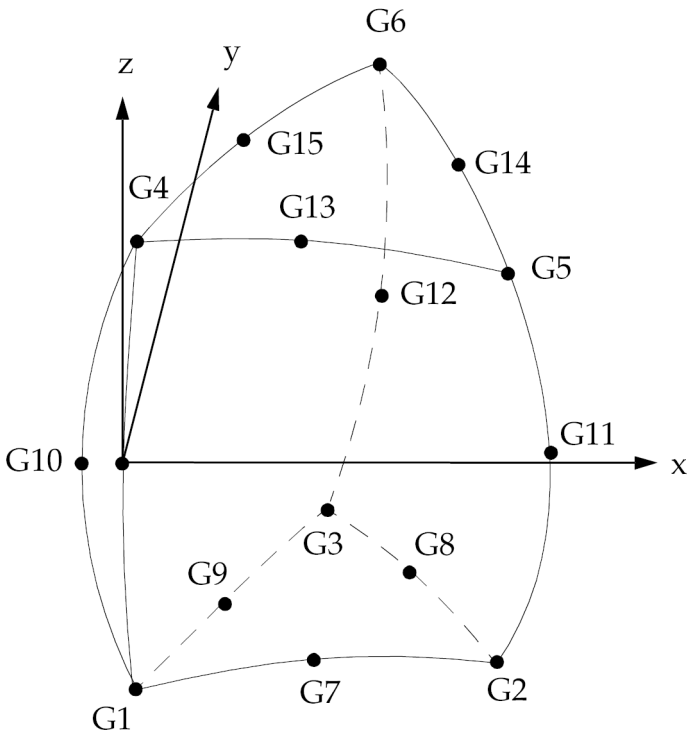


Figure 5-4. CPENTA Element Coordinate System

For non-hyperelastic elements, the origin of the CPENTA element 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 to G5 and G3 to G6, respectively.

CPENTA Format

The format of the CPENTA element entry is as follows

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								

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank)

Grid points G1, G2, and G3 define a triangular face. Grid points G1, G10, and G4 are on the same edge, etc. The midside nodes, G7 to G15, are optional. You can delete any or all midside nodes. The

continuations aren't required if you delete all midside nodes. Components of stress are output in the material coordinate system. You define the material coordinate system on the PSOLID entry.

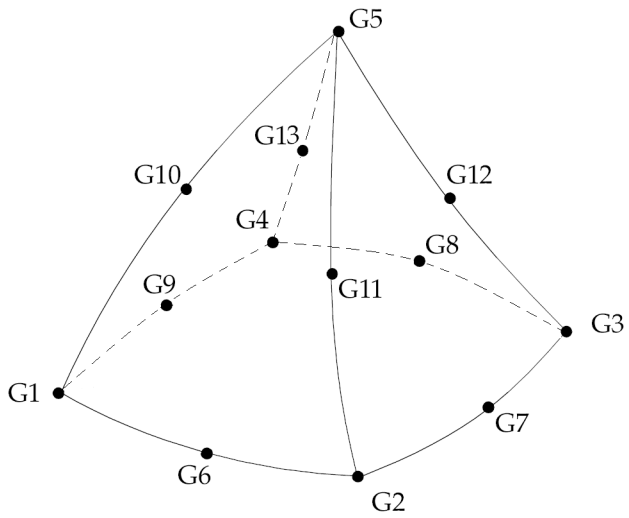
See Also

- CPENTA in the *NX Nastran Quick Reference Guide*.

Five-Sided Solid Element (CPYRAM)

The CPYRAM has five corner grid points and up to 13 grid points if you include the eight optional midside grid points.

The CPYRAM element is commonly used to model transitions from either linear or parabolic tetrahedral to linear hexahedral elements (although parabolic hexahedral elements could be used as well). It is needed because certain modeling features are best meshed with tetrahedral elements, while hexadedral elements are preferred where possible. NX Nastran calculates element stresses (σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{zx}) at the element's center and Gauss points. These stresses are extrapolated to obtain the stresses at the corner grid points. The element's connection geometry is shown in the following figure.



You can delete any or all of the midside nodes for the CPYRAM element. However, it is recommended that if midside nodes are used, then all midside grid points be included. Use the 13-noded CPYRAM in areas where accurate stress data recovery is required.

The CPYRAM element coordinate system is the same as the basic coordinate system.

CPYRAM Format

The format of the CPYRAM element entry is as follows:

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		

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank)

Grid points G1 through G4 define a quadrilateral face. The other four faces are triangles. G5 is the vertex and must be opposite with the quadrilateral face. The midside nodes, G6 to G13, are optional. If the ID of the midside node 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. Components of stress are output in the material coordinate system. You define the material coordinate system on the PSOLID entry.

See Also

- CPYRAM in the *NX Nastran Quick Reference Guide*.

Four-Sided Solid Element (CTETRA)

The CTETRA element is an isoparametric tetrahedron element with four vertex nodes and up to six additional midside nodes. If you use midside nodes, you should include all six nodes. The accuracy of the element degrades if some but not all the edge grid points are used.

The CTETRA solid element is used widely to model complicated systems (i.e. extrusions with many sharp turns and fillets, turbine blades). The element has a distinct advantage over the CHEXA when the geometry has sharp corners as you can have CTETRAs that are much better shaped than CHEXAs. However, you should always use CTETRAs with ten grid points for all structural simulations (e.g. solving for displacement and stress). The CTETRA with four grid points is overly stiff for these applications. In general, you should minimize your use of the 4-noded CTETRA, especially in the areas of high stress. However, CTETRA elements with four grid points are acceptable for heat transfer applications.

NX Nastran calculates element stresses (σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{zx}), at the element's center and Gauss points. These stresses are extrapolated to obtain the stresses at the corner grid points. The element's connection geometry is shown below.

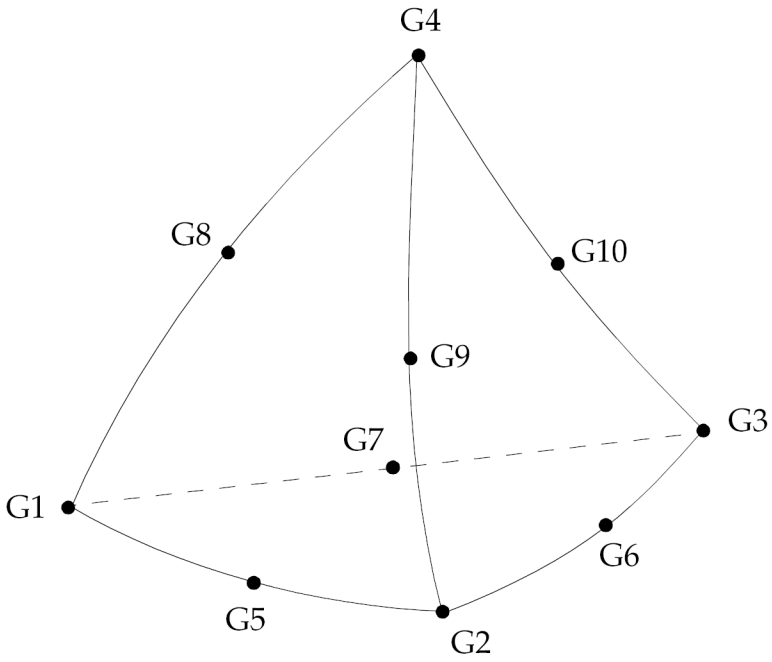


Figure 5-5. CTETRA Element Connection

The CTETRA element coordinate system is shown in Figure 5-6.

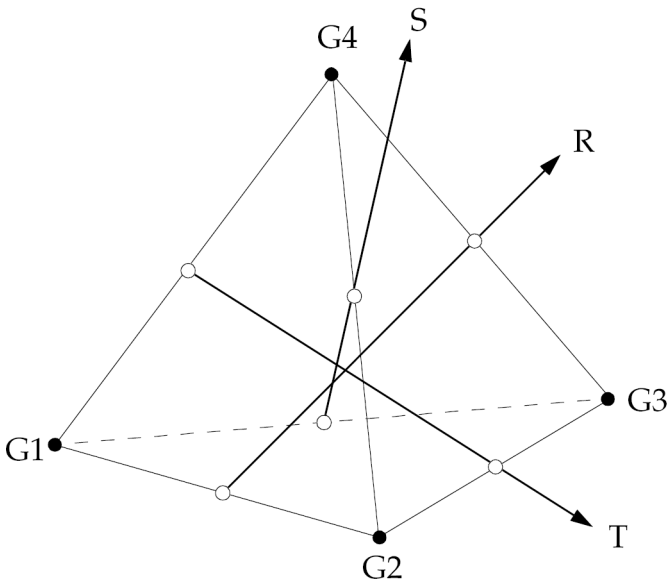


Figure 5-6. CTETRA Element Coordinate System

The CTETRA element coordinate system is derived from the three vectors R, S, and T which join the midpoints of opposite edges.

- The R vector joins the midpoints of edges G1-G2 and G3-G4.
- The S vector joins the midpoints of edges G1-G3 and G2-G4.

- The T vector joins the 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.

CTETRA Format

The format of the CTETRA element is as follows:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSOLID property entry. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer ≥ 0 or blank)

Grid points G1, G2, and G3 define a triangular face. The midside nodes, G5 to G10, must be located as shown on Figure 5-5. If you leave the ID of any midside node blank or set it to zero, the software adjusts the element's equations to give correct results for the reduced number of connections. You can't delete any of the element's corner grid points. Components of stress are output in the material coordinate system. You define the material coordinate system on the PSOLID property entry.

See Also

- CTETRA in the *NX Nastran Quick Reference Guide*.

5.3 Three-Dimensional Crack Tip Element (CRAC3D)

The three-dimensional crack tip element is used to model solids with a discontinuity due to a crack. Like the CRAC2D, the CRAC3D is a dummy element – you must specify the ADUM9 entry in the Bulk Data section. The connectivity is entered on the CRAC3D entry. The formats of the CRAC3D and ADUM9 entries are as follows:

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							

Field	Contents
EID	Element identification number.

Field	Contents
PID	Property identification number of a PRAC3D entry.
Gi	Grid point identification numbers of connection points.

1	2	3	4	5	6	7	8	9	10
ADUM9	64	0	6	0	CRAC3D				

You enter the properties of the CRAC3D element on the PRAC3D entry.

You have two options available, the brick option and the symmetric option. Figure 5-7 shows the 3-D brick and symmetric half-crack options with only the required connection points. Figure 5-8 shows the two options with all connection points.

- When you use the brick option, NX Nastran automatically subdivides the element into eight basic wedge elements. Grid points 1-10 and 19-28 are required, while grid points 11-18 and 29 -64 are optional. For the brick option, NX Nastran computes the stresses at the origin of the natural coordinates of wedges 4 and 5. It also computes the stress intensity factors K_i and K_{ii} from wedges 1 and 8.

- When you use the symmetric option, NX Nastran subdivides the element into four basic wedge elements. Grid points 1-7 and 19-25 are required. When you use this option, the stress is computed from wedge 4, and the stress intensity factor K_i is computed from wedge 1 only.

The CRAC3D element is based upon a 3-D formulation. Both of the faces (formed by grid points 1 through 18 and grid points 19 through 36) and that of the midplane (grid 37 through 46) should be planar. If there's any significant deviation in the element, the software issues an error message.

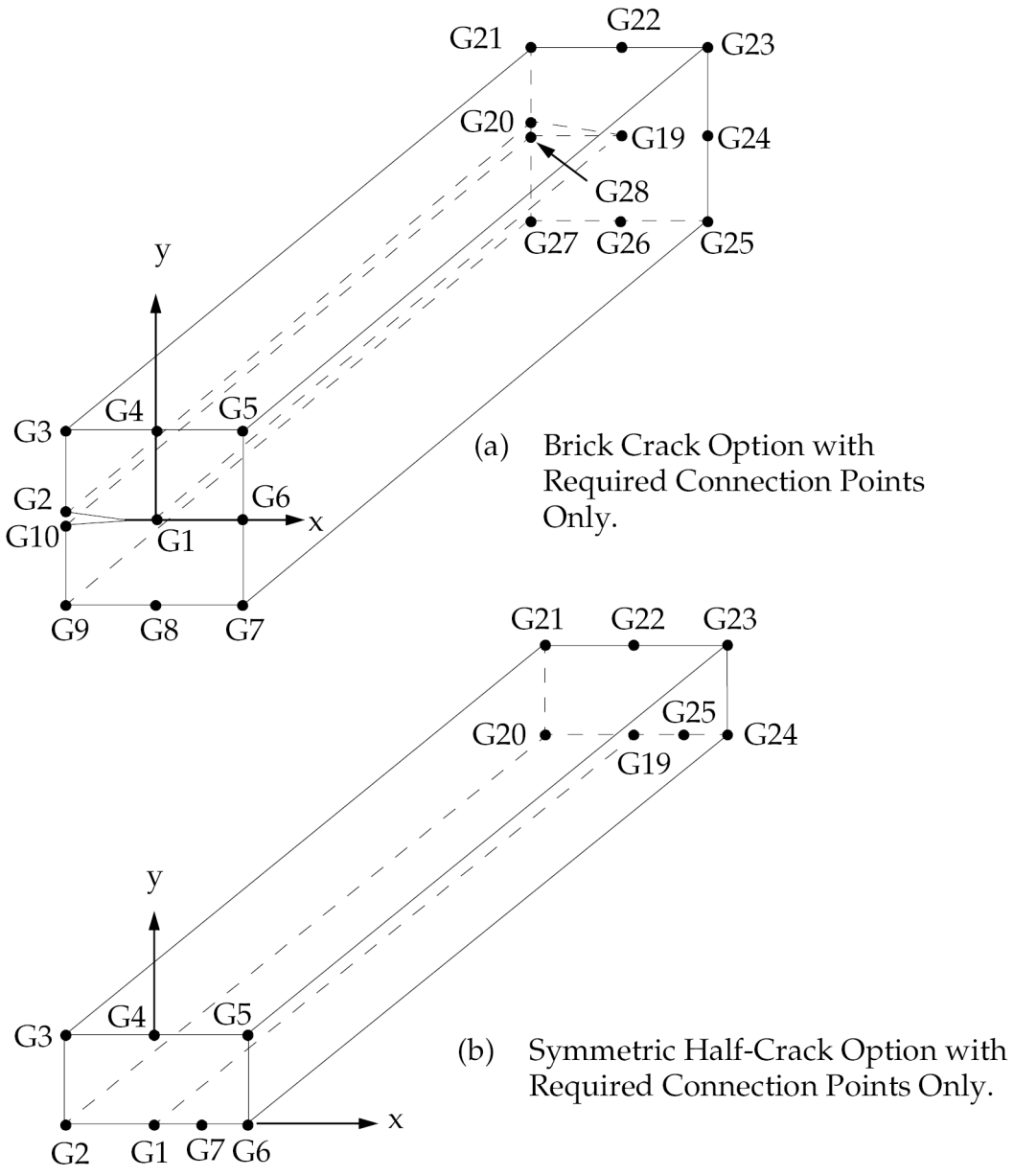
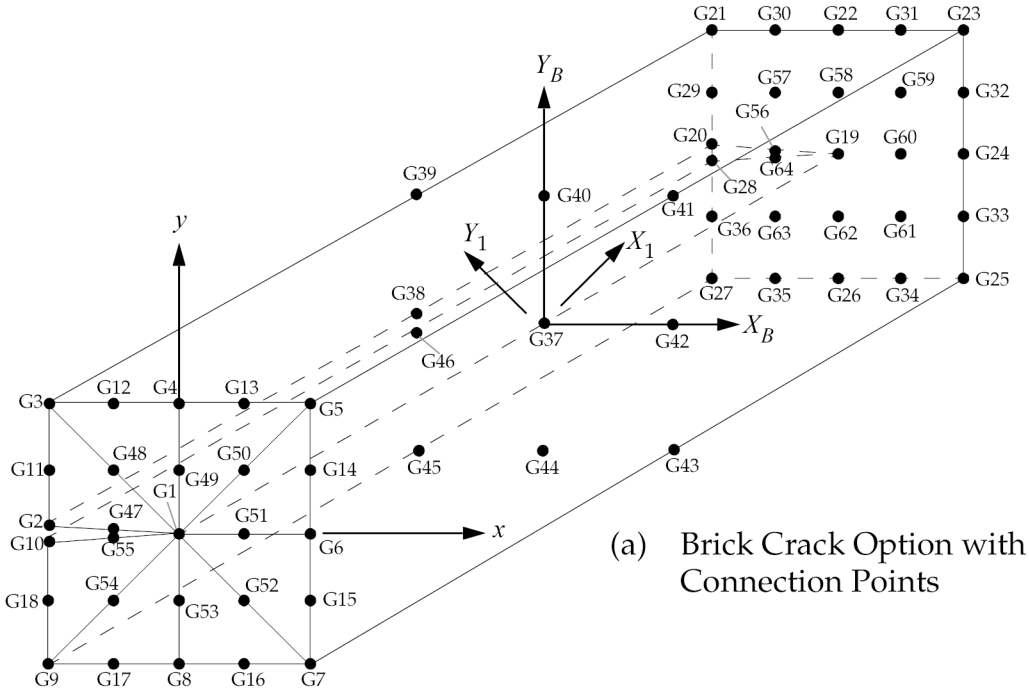
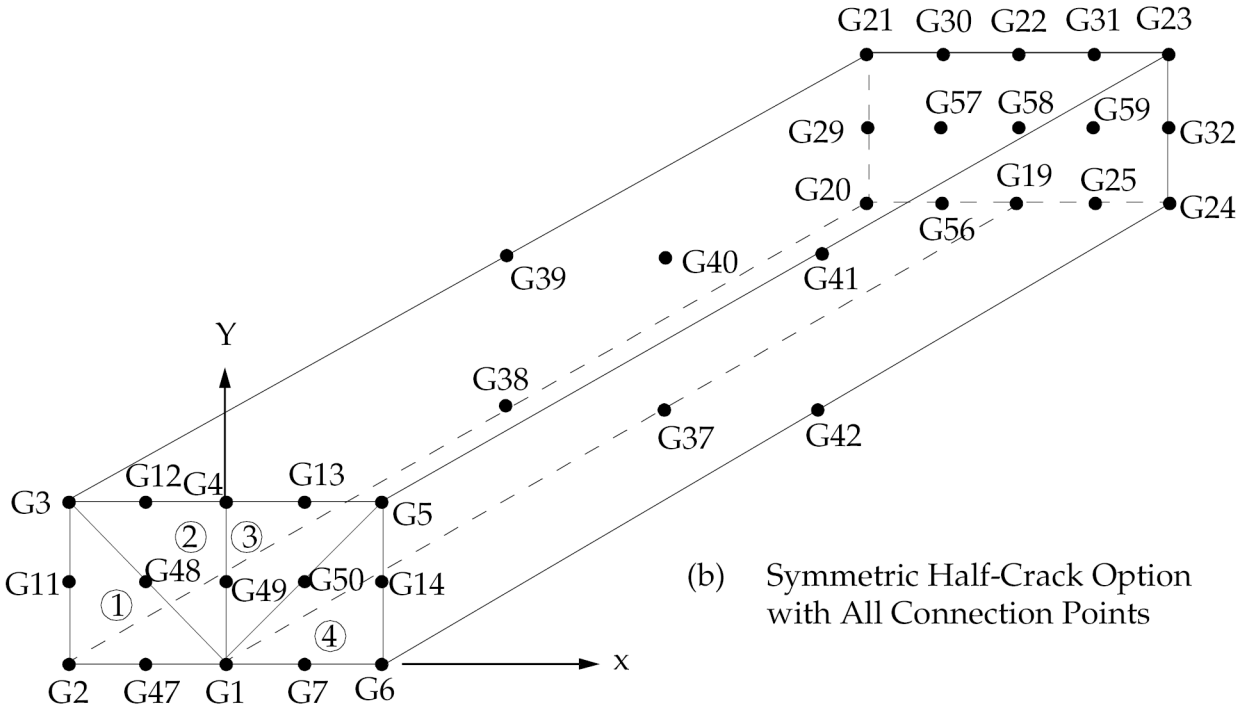


Figure 5-7. CRAC3D Solid Crack Tip Element



(a) Brick Crack Option with All Connection Points



(b) Symmetric Half-Crack Option with All Connection Points

Figure 5-8. CRAC3D Solid Crack Tip Element (continued)

S1	S2	S3	S4	S5	S6	S7	S8	S9
σ_x	σ_y	σ_z	τ_{xy}	τ_{yz}	τ_{zx}	K_I	K_{II}	K_{III}

- For the brick crack option, the software reports stresses at the average of the origin of the natural coordinate of wedges 4 and 5.
- For the symmetric crack option, the software reports stresses at the origin of the natural coordinate of wedge 4.

See Also

- PRAC3D in the *NX Nastran Quick Reference Guide*.
- CRAC3D in the *NX Nastran Quick Reference Guide*.
- ADUMi in the *NX Nastran Quick Reference Guide*.

5.4 Axisymmetric Solid Elements

The axisymmetric elements CTRAX3, CQUADX4, CTRAX6, CQUADX8, CTRIAX6, CTRIAX, and CQUADX define a solid ring by sweeping a surface defined on a plane through a circular arc. Loads are constant with azimuth for these elements; that is, only the zeroth harmonic is considered. There may be innovative modeling techniques that allow coupling this class of axisymmetric element with other elements, but there are no features to provide correct automatic coupling.

The axisymmetric elements in NX Nastran can be summarized into the following categories:

1. The recommended axisymmetric elements CTRAX3, CQUADX4, CTRAX6, and CQUADX8 support all capabilities of the elements described in categories 2 and 3 below, they can be defined in either the X-Z or X-Y basic coordinate system, and are supported in solutions 101, 103, 105, 106 (hyperelastic), 107, 108, 109, 110, 111, 112, 153, 159, 401, and 601. They support the materials linear isotropic (MAT1/MATT1), linear orthotropic (MAT3/MATT3), isotropic heat transfer (MAT4/MATT4), thermal material properties for anisotropic materials (MAT5/MATT5), and MATHP or MATHE for hyperelastic solutions.

2. The CTRIAX6 element is a linear axisymmetric element defined in the X-Z basic coordinate system, it is supported in SOLs 101, 103, 107, 108, 109, 110, 111, 112, 153, 159, and it supports the materials linear isotropic (MAT1/MATT1), linear orthotropic (MAT3/MATT3), isotropic heat transfer (MAT4/MATT4), and thermal material properties for anisotropic materials (MAT5/MATT5).

3. The CTRIAX and CQUADX elements are defined in the X-Y basic coordinate system, they are fully nonlinear elements supported in solution types SOL 106 and 601 but only support hyper-elastic material defined on the MATHP bulk data entry. The CTRIAX and CQUADX axisymmetric elements are described in the Hyperelastic Axisymmetric Elements section.

CTRAX3, CTRAX6, CQUADX4, and CQUADX8

These axisymmetric elements are defined in either the X-Z plane, or the X-Y plane of the basic coordinate system, and are supported in solutions 101, 103, 105, 106 (hyperelastic), 107, 108, 109, 110, 111, 112, 153, 159, 401 and 601.

Assumptions

- When defined on the X-Z plane, the Z axis is the axial direction and the X axis is the radial direction. When defined on the X-Y plane, the X axis is the axial direction and the Y axis is the radial direction.

- All grids must lie either in the X-Z plane, or in the X-Y plane of the basic coordinate system.
- All loads must be in the plane in which the elements are oriented.
- XY and YZ shear stresses and strains are zero.
- Orthotropic material properties defined with a MAT3 entry are given in the (X_m, Z_m) coordinate system shown in Figure 5-10 when the element is defined on the XZ plane, or in the (X_m, Y_m) coordinate system shown in Figure 5-11 when the element is defined on the XY plane. Note that a positive orientation angle (THETA) rotates the material coordinate system in a radial-to-axial direction.
- Gravity vectors must be parallel to the axis of symmetry.
- All angular velocity vectors must lie on the axis of symmetry.
- Grid points G1 through G8 must be numbered as shown below (midside grids on the CTRAX6 and CQUADX8 cannot be eliminated).
- 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.

Materials

The supported materials are linear isotropic (MAT1/MATT1), linear orthotropic (MAT3/MATT3), isotropic heat transfer (MAT4/MATT4), and thermal material properties for anisotropic materials (MAT5/MATT5). For hyperelastic solutions, MATHP or MATHE can be used. Material properties (if defined on a MAT3 entry) are given in the (rm, zm) coordinate system as shown in the figures below.

Properties

The CTRAX3, CTRAX6, CQUADX4, and CQUADX8 use the property PSOLID, unless the solution is hyperelastic, then a PLSOLID is used.

Loads

- By defauilt, 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

  = 424.115 Newtons

See Axisymmetric Element Sectional Properties.

- Pressures can be applied using the PLOADX1 bulk data entry.
- Gravity vectors must be parallel to the axis of symmetry.

Grid Point DOF

Three translational degrees of freedom are assigned to each grid. Translations are in the directions of the nodal displacement coordinate system and have the dimensions of length. The out of plane DOF must be restraint by specifying an SPC, or PARAM,AUTOSPC,YES.

Output

- Displacements
- Stress
- Strain
- Reaction Forces
- Elastic Strain Energy (per radian)

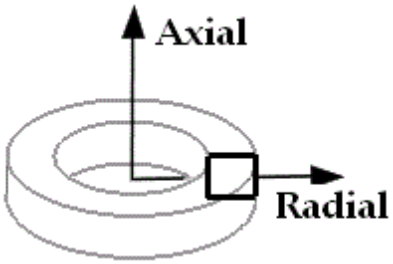


Figure 5-9. General Axisymmetric Element Idealization

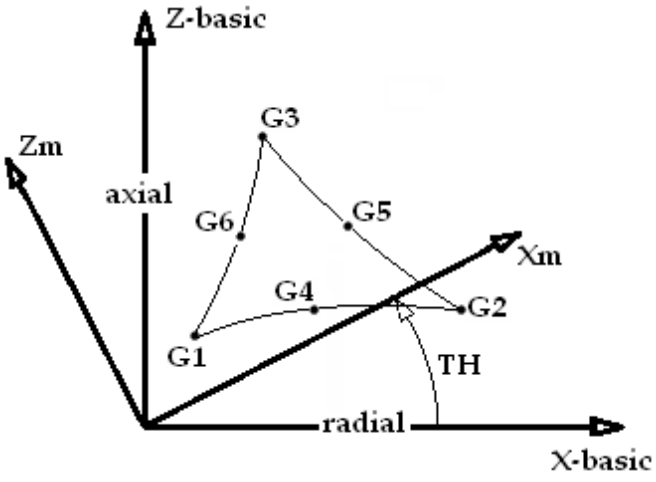


Figure 5-10. Axisymmetric Element Coordinate Systems, XZ-Plane

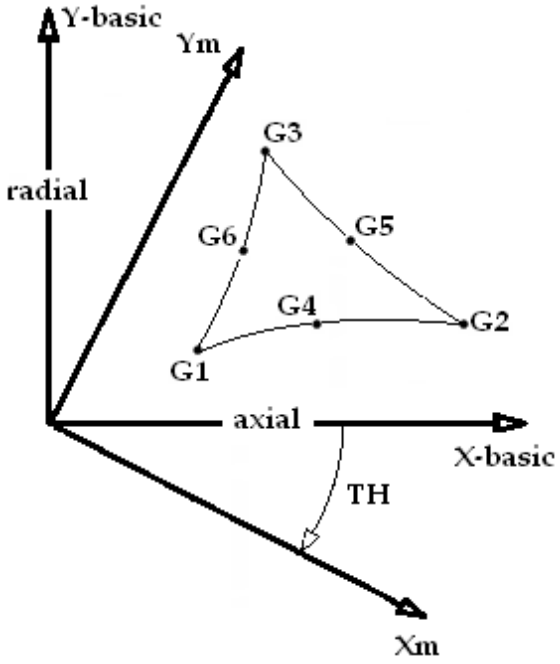


Figure 5-11. Axisymmetric Element Coordinate Systems, XY-Plane

CTRAX3, CTRAX6, CQUADX4, CQUADX8 Element Formulation

The default formulation for this element uses area coordinates for displacement interpolation functions to achieve a linear variation of displacements within the element. Details for this element may be found in Zienkiewicz and Taylor, 1989, or in Cook et al, 1989.

A mean dilatational formulation may also be selected. This formulation is useful for eliminating element locking which occurs when modelling nearly incompressible materials (Poisson's ratio greater than .49). The standard displacement interpolation functions are used, but the strain-displacement matrix is split into dilatational and deviatoric parts. The dilatational part is replaced with the mean dilatational strain-displacement matrix for the element. This technique is called the strain projection or B technique. For details see Hughes, 1987.

A single value for the mean stress or pressure is obtained across the whole element. Therefore, this formulation is not recommended unless locking due to incompressibility is a problem.

See Tables 17-6 and 17-7 under the PSOLID bulk data entry in the *NX Nastran Quick Reference Guide*.

References:

- Cook, R. D., Malkus, D. S., and Plesha, M. E., Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, 1989, pp 114-115, 293-297.

- Hughes, T. J. R., The Finite Element Method, Linear Static and Dynamic Finite Element Analysis, Prentice-Hall, Englewood Cliffs, NJ, 1987, pp 232-237.

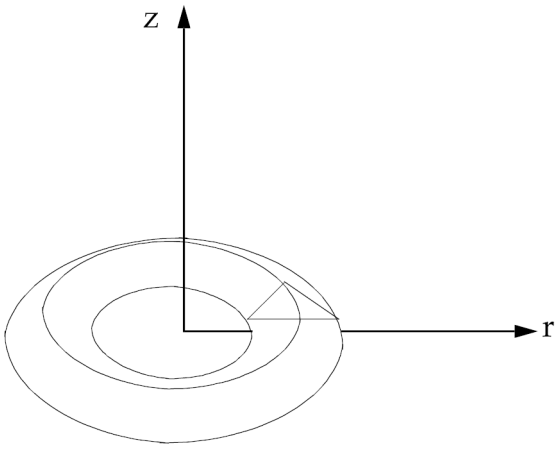
- Zienkiewicz, O. C. and Taylor, R. L., The Finite Element Method, Volume 1, Basic Formulation and Linear Problems, 4th Edition, McGraw-Hill, London, 1989, pp 72-80, 128-130.

CTRIAX6

The CTRIAX6 is a linear isoparametric element with an axisymmetric configuration that is restricted to axisymmetric applied loading. It is used for the modeling of axisymmetric, thick-walled structures of arbitrary profile. This element is not designed to be used with any other elements. Otherwise, CTRIAX6 is used in a conventional manner, and except for its own connection entry (and a pressure load entry, PLOADX1) it doesn't require special Bulk Data entries. There is no property bulk data entry for CTRIAX6. You define the material property ID for the element directly on the CTRIAX entry.

Note: You should only use the CTRIAX6 element for analyzing axisymmetric structures with axisymmetric loads. This application should not be confused with the cyclic symmetric capability within NX Nastran, which can handle both axisymmetric and nonaxisymmetric loading.

The coordinate system for the CTRIAX6 element is shown in Figure 5-12. Cylindrical anisotropy is optional. Orientation of the orthotropic axes in the (*r* , *z*) plane is specified by the angle θ. Deformation behavior of the element is described in terms of translations in the *r* and *z* directions at each of the six grid points. All other degrees-of-freedom should be constrained.



The CTRIAX6 element is a "solid of revolution" element with a triangular cross section. You must supply the three corner grid points and may optionally supply up to three midside nodes.

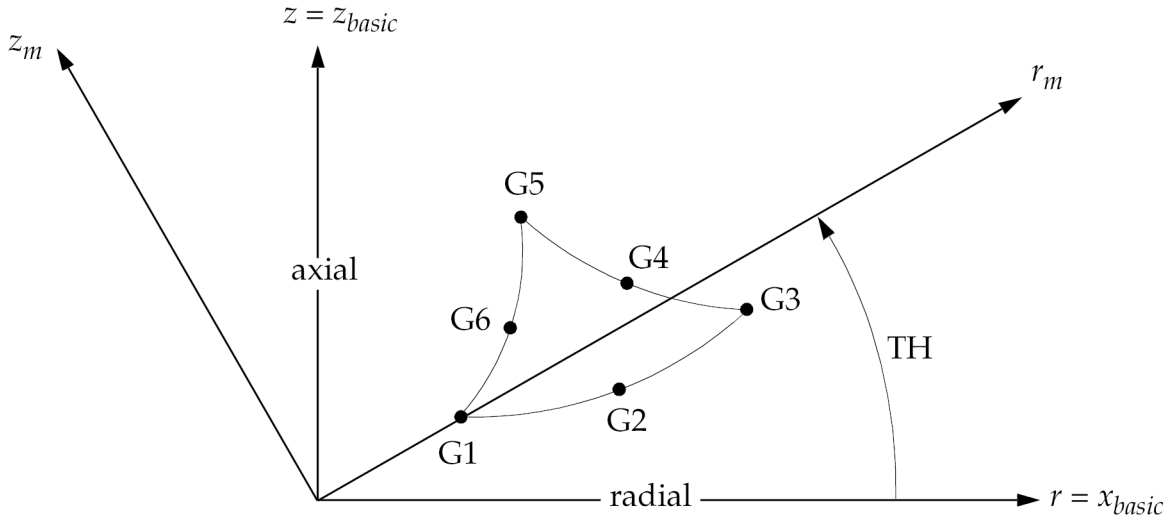


Figure 5-12. CTRIAX6 Element Coordinate System

You can request that NX Nastran output the following stresses, which it evaluates at the three vertex grid points and the element's centroid:

- σ_r – stress in r_m direction of material coordinate system.
- σ_θ – stress in azimuthal direction.
- σ_z – stress in z_m direction of material coordinate system.
- τ_{rz} – shear stress in material coordinate system.
- Maximum principal stress.
- Maximum shear stress.
- von Mises equivalent or octahedral shear stress.

See Also

- CTRIAX6 in the *NX Nastran Quick Reference Guide*.

Axisymmetric Element Sectional Properties

Before NX Nastran 10, the stiffness, mass and loads for the CTRAX3, CQUADX4, CTRAX6, CQUADX8, CTRIAX, and CQUADX elements were based on a per radian section basis.

Beginning in NX Nastran 10, the stiffness, mass and loads for these elements is now based on a 2*PI section basis. For example, a concentrated load (e.g., FORCE entry) defined on a grid point is the total of the force around the circumference.

The system cell 587 can optionally be set to 1 to revert to the per radian section basis. Note that the CTRAX3, CQUADX4, CTRAX6, and CQUAD8 elements remain on a 2*PI basis in a heat transfer analysis, regardless of the system cell setting.

The following table summarizes the output and input items which change, or do not change, when system 587 =1 is defined. You should adjust the inputs listed here to be consistent with the section basis selected.

Item	Where Specified	Category	Changes	Description
BCRESULTS	Case Control	Output	YES	Edge contact force output changes. Pressure output does not change.
BGRESULTS	Case Control	Output	YES	Edge glue force output changes. Pressure output does not change.
EKE	Case Control	Output	YES	Element kinetic energy changes.
ELSUM	Case Control	Output	NO	*Always* computed in a 2*PI basis regardless of the system cell 587 setting.
ESE	Case Control	Output	YES	Element strain energy changes.
GPFORCE	Case Control	Output	YES	Grid point force changes.
GPKE	Case Control	Output	NO	Grid point kinetic energy *does not* change.

MEFFMASS	Case Control	Output	YES	Modal effective mass changes.
MPCFORCES	Case Control	Output	YES	MPC constraint forces change.
OLOAD	Case Control	Output	YES	Applied load vector output changes.
SPCFORCES	Case Control	Output	YES	SPC force output changes.
GROUNDCHECK	Case Control	Output	YES	Strain Energy output changes.
WEIGHTCHECK	Case Control	Output	YES	Weight / Mass output changes.
External Work	f06 file Output	Output	YES	Work output changes.
GRID POINT WEIGHT GENERATOR	f06 file Output	Output	YES	Grid point weight output changes.
K2GG	Case Control	Input	YES	DMIG stiffness input changes.
K2PP	Case Control	Input	YES	DMIG stiffness input changes.
K42GG	Case Control	Input	YES	DMIG stiffness input changes.
M2GG	Case Control	Input	YES	DMIG mass input changes.
M2PP	Case Control	Input	YES	DMIG mass input changes.
P2G	Case Control	Input	YES	Direct input load matrices change.
BOLTFOR	Bulk Data	Input	YES	Preload bolt force (LOAD) changes.
CELASi	Bulk Data	Input	YES	Spring element input changes.
CMASSi	Bulk Data	Input	YES	Mass element input changes.
PELAS	Bulk Data	Input	YES	Elastic stiffness for changes.
PMASS	Bulk Data	Input	YES	Scalar mass for changes.
CONM*	Bulk Data	Input	YES	Concentrated mass input changes.
CPLSTS*	Bulk Data	Input	YES	Plane stress thickness input changes.
FORCE	Bulk Data	Input	YES	GRID point force magnitude changes.
FORCE1	Bulk Data	Input	YES	GRID point force magnitude changes.
FORCE2	Bulk Data	Input	YES	GRID point force magnitude changes.
PPLANE	Bulk Data	Input	YES	Thickness of plane stress element changes.
SLOAD	Bulk Data	Input	YES	Static load changes.

Chapter 6: Special Element Types

- *Introduction to the Special Element*
- *General Element Capability (GENEL)*
- *Bushing (CBUSH) Elements*
- *CWELD Connector Element*
- *Gap and Line Contact Elements*
- *Concentrated Mass Elements (CONM1, CONM2)*
- *p-Elements*
- *Hyperelastic Elements*
- *Interface Elements*

This section describes special elements in NX Nastran. These include:

- general (GENEL) elements
- bushing elements
- CWELD elements
- gap and line contact elements
- concentrated mass elements
- p-elements
- hyperelastic elements
- interface elements

6.1 General Element Capability (GENEL)

In NX Nastran, you use the GENEL entry to create general elements whose properties are defined in terms of deflection, influence coefficients, or stiffness matrices which can be connected between any number of grid points. One of the important uses of the general element is the representation of part of a structure by means of experimentally measured data. No output data is prepared for the general element.

The GENEL element is really not an element in the same sense as the CBAR or CQUAD4 element. There are no properties explicitly defined and no data recovery is performed. The GENEL element is very useful when you want to include in your model a substructure that is difficult to model using the standard elements.You can use the GENEL element to describe a substructure that has an arbitrary number of connection grid points or scalar points. You can derive the input data entered for the GENEL element from a hand calculation, another computer model, or actual test data.

The general element is a structural stiffness element connected to any number of degrees-of-freedom, as you specify. In defining the form of the externally generated data on the stiffness of the element, there are two major options:

1. Instead of supplying the stiffness matrix for the element directly, you provide the deflection influence coefficients for the structure supported in a non-redundant manner. The associated matrix of the restrained rigid body motions may be input or may be generated internally by the program.

2. You can input the stiffness matrix of the element directly. This stiffness matrix may be for the unsupported body, containing all the rigid body modes, or it may be for a subset of the body's degrees-of-freedom from which some or all of the rigid body motions are deleted. In the latter case, the option is given for automatic inflation of the stiffness matrix to reintroduce the restrained rigid body terms, provided that the original support conditions did not constitute a redundant set of reactions. An important advantage of this option is that, if the original support conditions restrain all rigid body motions, the reduced stiffness matrix need not be specified by the user to high precision in order to preserve the rigid body properties of the element.

The defining equation for the general element when written in the flexibility form is

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^T & 0 \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix}$$

Equation 6-1.

where:

$[Z]$	=	is the matrix of deflection influence coefficients for coordinates $\{u_i\}$ when coordinates $\{u_d\}$ are rigidly restrained.
$[S]$	=	is a rigid body matrix whose terms are the displacements $\{u_i\}$ due to unit motions of the coordinates $\{u_d\}$, when all $f_i = 0$.
$[f_i]$	=	are the forces applied to the element at the $\{u_i\}$ full coordinates.
$[f_d]$	=	are the forces applied to the element at the $\{u_d\}$ coordinates. They are assumed to be statically related to the $\{f_i\}$ forces, i.e., they constitute a nonredundant set of reactions for the element.

The defining equation for the general element when written in the stiffness form is

$$\begin{Bmatrix} f_i \\ f_d \end{Bmatrix} = \begin{bmatrix} k & -k\,S \\ -S^T k & S^T k\,S \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix}$$

Equation 6-2.

where all symbols have the same meaning as in Eq. 6-1 and $[k] = [Z]^{-1}$, when $[k]$ is nonsingular. Note, however, that it is permissible for $[k]$ to be singular. Eq. 6-2 derivable from Eq. 6-1 when $[k]$ is nonsingular.

Input data for the element consists of lists of the u_i and u_d coordinates, which may occur at either geometric or scalar grid points; the values of the elements of the $[Z]$ matrix, or the values of the elements of the $[k]$ matrix; and (optionally) the values of the elements of the $[S]$ matrix.

You may request that the program internally generate the $[S]$ matrix. If so, the u_i and coordinates occur only at geometric grid points, and there must be six or less u_d coordinates that provide a nonredundant set of reactions for the element as a three-dimensional body.

The $[S]$ matrix is internally generated as follows. Let $\{u_b\}$ be a set of six independent motions (three translations and three rotations) along coordinate axes at the origin of the basic coordinate system. Let the relationship between $\{u_d\}$ and $\{u_b\}$.

$$\{u_d\} = [D_d]\{u_b\}$$

Equation 6-3.

The elements of $[D_d]$ are easily calculated from the basic (x,y,z) geometric coordinates of the grid points at which the elements of $\{u_d\}$ occur, and the transformations between basic and global (local) coordinate systems. Let the relationship between $\{u_i\}$ and $\{u_b\}$ be

$$\{u_i\} = [D_i]\{u_b\}$$

Equation 6-4.

where $[D_i]$ is calculated in the same manner as $[D_d]$. Then, if $[D_d]$ is nonsingular,

$$[S] = [D_i][D_d]^{-1}$$

Equation 6-5.

Note that, if the set $\{u_d\}$ is not a sufficient set of reactions, $[D_d]$ is singular and $[S]$ cannot be computed in the manner shown. When $\{u_d\}$ contains fewer than six elements, the matrix $[D_d]$ is not directly invertible but a submatrix $[a]$ of rank r, where r is the number of elements of $\{u_d\}$, can be extracted and inverted.

A method which is available only for the stiffness formulation and not for the flexibility formulation will be described. The flexibility formulation requires that $\{u_d\}$ have six components. The method is as follows. Let $\{u_d\}$ be augmented by 6–r displacement components $\{u_d'\}$ which are restrained to zero value. We may then write

$$\begin{Bmatrix} u_d \\ \hline d_d' \end{Bmatrix} = \begin{bmatrix} D_d \\ \hline D_d' \end{bmatrix} \{u_b\} = [\bar{D}]\{u_b\}$$

Equation 6-6.

The matrix $[D_d]$ is examined and a nonsingular subset $[a]$ with r rows and columns is found. $\{u_b\}$ is then reordered to identify its first r elements with $\{u_d\}$. The remaining elements of $\{u_b\}$ are equated to the elements of $\{u_d\}$. The complete matrix $[\bar{D}]$ then has the form

$$[\bar{D}] = \left[\begin{array}{c|c} a & b \\ \hline 0 & I \end{array}\right]$$

Equation 6-7.

with an inverse

$$[\bar{D}]^{-1} = \left[\begin{array}{c|c} a^{-1} & -a^{-1}b \\ \hline 0 & I \end{array}\right]$$

Equation 6-8.

Since the members of $\{u_d'\}$ are restrained to zero value,

$$\{u_b\} = [D_r]\{u_d\}$$

Equation 6-9.

where $[D_r]$ is the $(\times r)$ partitioned matrix given by

$$[D_r] = \begin{bmatrix} a^{-1} \\ \hline 0 \end{bmatrix}$$

Equation 6-10.

The $[D_i]$ matrix is formed as before and the $[S]$ matrix is then

$$[S] = [D_i][D_r]$$

Equation 6-11.

Although this procedure will replace all deleted rigid body motions, it is not necessary to do this if a stiffness matrix rather than a flexibility matrix is input. It is, however, a highly recommended procedure because it will eliminate errors due to non-satisfaction of rigid body properties by imprecise input data.

The stiffness matrix of the element written in partitioned form is

$$[K_{ee}] = \begin{bmatrix} K_{ii} & K_{id} \\ \hline K_{id}^T & K_{dd} \end{bmatrix}$$

Equation 6-12.

When the flexibility formulation is used, the program evaluates the partitions of $[K_{ee}]$ from $[Z]$ and $[S]$ as follows:

$$[K_{ii}] = [Z]^{-1}$$

Equation 6-13.

$$[K_{id}] = [Z]^{-1}[S]$$

Equation 6-14.

$$[K_{dd}] = [S]^{T}[Z]^{-1}[S]$$

Equation 6-15.

If a stiffness matrix, [*k*], rather than a flexibility matrix is input, the partitions of [K_{ee}] are

$$[K_{ii}] = [k]$$

Equation 6-16.

$$[K_{id}] = -[k][S]$$

Equation 6-17.

$$[K_{dd}] = -[S]^{T}[k][S]$$

Equation 6-18.

No internal forces or other output data are produced for the general element.

There are two approaches that you may use to define the properties of a GENEL element: (1) the stiffness approach, in which case you define the stiffness for the element; and (2) the flexibility approach, in which case you define the flexibility matrix for the element.

1. The stiffness approach:

$$\begin{Bmatrix} f_i \\ f_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ -S^{T}K & S^{T}KS \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix}$$

2. The flexibility approach:

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^{T} & O \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix}$$

where:

$$u_i = [u_{i1}, u_{i2}, \ldots, u_{im}]^T$$

$$u_d = [u_{d1}, u_{d2}, \ldots, u_{dn}]^T$$

$$KZ = [K] = [Z] = \begin{bmatrix} KZ11 & \ldots & \ldots & \ldots \\ KZ21 & KZ22 & \ldots & \ldots \\ KZ31 & KZ32 & \ldots & \ldots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \ldots & \ldots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ]$$

$$S = [S] = \begin{bmatrix} S11 & \ldots & S_{1n} \\ S21 & \ldots & \ldots \\ S31 & \ldots & \ldots \\ \vdots & \vdots & \vdots \\ S_{m1} & \ldots & S_{mn} \end{bmatrix}$$

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body whose rigid body motions are defined by $\{u_i\} = [S]\{u_d\}$.

The required input is the redundant displacement set $\{u_i\}$ list and the lower triangular portion of $[K]$ or $[Z]$ (note: $[Z] = [K]^{-1}$). Additional input may include the determinant $\{u_d\}$ 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 (translation or rotation, no scalar points). If the $\{u_d\}$ set contains exactly six degrees of freedom, then the $[S]$ matrix computed internally describes the rigid motion at $\{u_i\}$ due to unit values of the components of $\{u_d\}$.

When the $[S]$ matrix is omitted, the data describing the element is in the form of a stiffness matrix (or flexibility matrix) for a redundant subset of the connected degrees of freedom, that is, all of the degrees of freedom over and above those required to express the rigid body motion of the element. In this case, extreme precision is not required because only the redundant subset is input, not the entire stiffness matrix. Using exactly six degrees of freedom in the $\{u_d\}$ set and avoiding the $[S]$ matrix is easier and is therefore recommended. An example of defining a GENEL element without entering an $[S]$ is presented later.

See Also

- GENEL in the *NX Nastran Quick Reference Guide*

GENEL Example: Robotic Arm

For an example of the GENEL element, consider the robotic arm shown in Figure 6-1. The arm consists of three simple bar members with a complex joint connecting members 1 and 2. For the problem at hand, suppose that the goal is not to perform a stress analysis of the joint but rather to compute the deflection of the end where the force is acting. You can make a detailed model of the

joint, but it will take a fair amount of time and the results will still be questionable. The ideal choice is to take the joint to the test lab and perform a static load test and use those results directly.

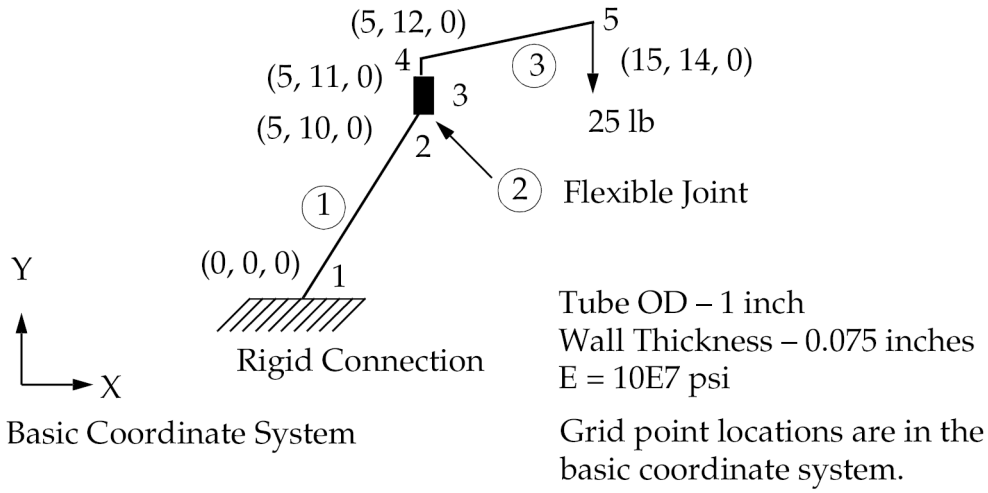


Figure 6-1. Robotic Arm with Joint

The GENEL element is the ideal tool for this job. The joint is tested in the test lab by constraining one end of the joint and applying six separate loads to the other end as shown in Figure 6-2.

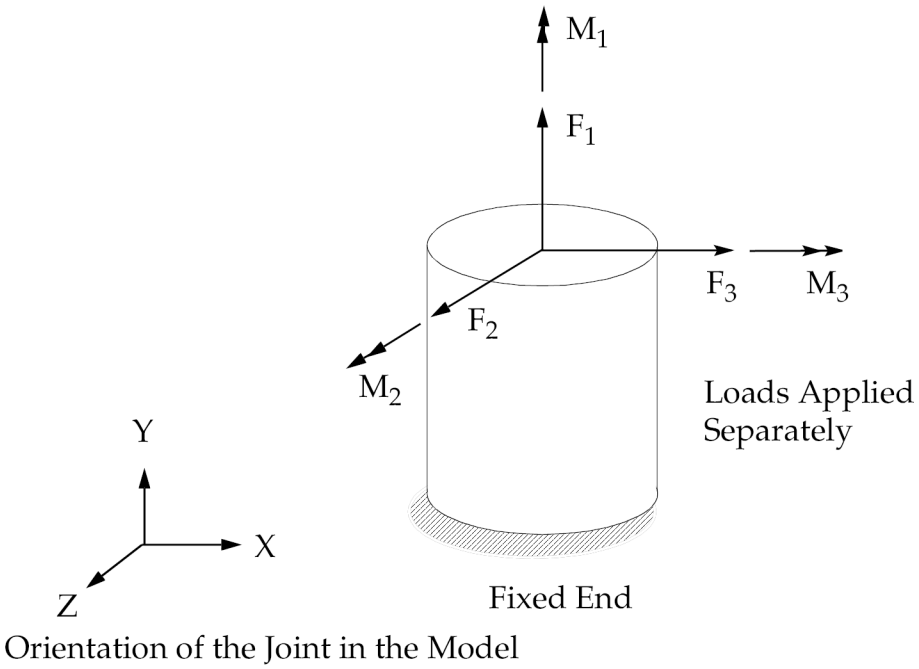


Figure 6-2. The Test Arrangement to Obtain the Flexibility Matrix

Table 6-1 summarizes the displacements measured for each of the applied loads.

Table 6-1. Test Results for the Joint

Deflection (10^{-7}) Due to Unit Loads						
	F_1	F_2	F_3	M_1	M_2	M_3
F_1 DISP	2.2	0	0	0	0	0
F_2 DISP	0	1.58	0	0	0	2.38
F_3 DISP	0	0	1.58	0	-2.38	0
M_1 Rotation	0	0	0	6.18	0	0
M_2 Rotation	0	0	-2.38	0	4.75	0
M_3 Rotation	0	2.38	0	0	0	4.75

Table 6-1 also represents the flexibility matrix of the joint with the rigid body properties removed. By specifying all of the degrees of freedom at grid point 2 as being in the dependent set $\{u_d\}$ and the $[S]$ degrees of freedom at grid point 3 as being in the independent set $\{u_i\}$, the matrix is not required. The input file showing the model with the GENEL is shown in Listing 6-1.

```
$
$FILENAME - GENEL1.DAT
$
CORD2R  1         0       0.      0.      0.      1.      0.      0.
        0.        1.      0.
GENEL   99                2       1       2       2       2       3
        2       4         2       5       2       6
        UD                3       1       3       2       3       3
        3       4         3       5       3       6
        Z       2.20E-7   0.0     0.0     0.0     0.0     0.0     1.58E-7
        0.0     0.0       0.0     2.38E-7 1.58E-7 0.0     -2.38E-70.0
        6.18E-7 0.0       0.0     4.75E-7 0.0     4.75E-7
$
GRID    1                 0.0     0.0     0.0
GRID    2                 5.      10.     0.0      1
GRID    3                 5.      11.     0.0      1
GRID    4                 5.      12.     0.0
GRID    5                 15.     14.     0.0
$
CBAR    1       1         1       2       1.0     1.0     0.0
CBAR    2       1         3       4       1.0     1.0     0.0
CBAR    3       1         4       5       1.0     1.0     0.0
$
FORCE   1       5         0       25.     0.0     -1.     0.0
SPC     1       1         123456  0.0
PBAR    1       1         .2179   .02346  .02346  .04692
$
MAT1    1       1.+7              .3
```

Listing 6-1. GENEL Element Input File

The flexibility matrix generated from the test data was properly aligned with the model geometry with the use of a local coordinate (note the CD field of grid points 2 and 3). Using this local coordinate system, the output displacement X-axis corresponds to the F1 direction, etc.

As an alternative approach, you can use direct matrix input (DMIG entries) to input structure matrices.

See Also

- "Direct Matrix Input" in the *NX Nastran User's Guide*

6.2 Bushing (CBUSH) Elements

The bushing (generalized spring and damper) elements consist of the following:

- CBUSH
- CBUSH1D

The generalized spring-damper element CBUSH is a structural scalar element connecting two noncoincident grid points, or two coincident grid points, or one grid point with an associated PBUSH entry. This combination is valid for any structural solution sequence. To make frequency dependent the PBUSH need only have an associated PBUSHT Bulk Data entry. The PBUSHT entry for frequency dependency is only used in SOL 108 and SOL 111. You can also use the PBUSHT entry to define load-displacement dependency in SOL 106.

Figure 6-3 shows some of the advantages of using the CBUSH element over CELASi elements. For example, if you use CELASi elements and the geometry isn't aligned properly, internal constraints may be induced. The CBUSH element contains all the features of the CELASi elements plus it avoids the internal constraint problem. The following example demonstrates the use of the CBUSH element as a replacement for scalar element for static analysis. The analysis joins any two grid points by user-specified spring rates, in a convenient manner without regard to the location or the displacement coordinate systems of the connected grid points. This method eliminates the need to avoid internal constraints when modeling.

The model shown in Figure 6-3 has two sheets of metal modeled with CQUAD4 elements. The sheets are placed next to each other. There are grid points at the common boundary of each sheet of metal, which are joined by spot welds. The edge opposite the joined edge of one of the sheets is constrained to ground. The grid points at the boundary are slightly misaligned between the two sheets due to manufacturing tolerances. There is a nominal mesh size of 2 units between the grid points, with 10 elements on each edge. The adjacent pairs of grid points are displaced from each other in three directions inside a radius of 0.1 units in a pattern that maximizes the offset at one end, approaches zero at the midpoint, and continues to vary linearly to a maximum in the opposite direction at the opposite end.

CELASi elements are used in the first model, and CBUSH elements are used in a second, unconnected model. PLOTEL elements are placed in parallel with the CELAS2 elements to show their connectivity. The second model is identical to the first model with respect to geometry, constraints, and loading.

A static loading consisting of a point load with equal components in all three directions on the center point opposite the constrained edge is applied. The first loading condition loads only the first model and the second loading condition loads only the second model, allowing comparison of the response for both models in one combined analysis. The input file bushweld.dat is in the test problem library.

In modal frequency response, the basis vectors (system modes) [φ] will be computed only once in the analysis and will be based on nominal values of the scalar frequency dependent springs. In general, any change in their stiffness due to frequency will have little impact on the overall contribution to the structural modes.

The stiffness matrix for the CBUSH element takes the diagonal form in the element system:

$$K_o = \begin{bmatrix} k_u & & & & & \\ & k_v & & & & \\ & & k_w & & & \\ & & & k_{\theta_x} & & \\ & & & & k_{\theta_y} & \\ & & & & & k_{\theta_z} \end{bmatrix}$$

For the B matrix replace the k terms with b.

When transformed into the basic system, there is coupling between translations and rotations, thus ensuring rigid body requirements.

The element axes are defined by one of the following procedures:

- If a CID is specified 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. The options GO or (X1,X2,X3) have no meaning and will be ignored. Then $[T_{ab}]$ is computed directly from CID.

- For noncoincident grids (GA ≠ GB), if neither options GO or (X1,X2,X3) is specified and no CID is specified, then the line AB is the element x-axis. No y-axis or z-axis need be specified. This option is only valid when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified but K2, K3, K5, K6 (or B2, B3, B5, B6) are not specified. If K2, K3, K5, K6 (or B2, B3, B5, B6) are specified, a fatal message will be issued. Then $[T_{ab}]$ is computed from the given vectors like the beam element.

DMAP Operations

Direct Frequency Response

Nominal Values

The following matrices are formed only once in the analysis and are based on the parameter input to EMG of /' '/ implying that the nominal values only are to be used for frequency dependent springs and dampers.

$$K_{dd} = (1 + ig)K^1_{dd} + K^{2x}_{dd} + iK^4_{dd}$$

$$M_{dd} = M^1_{dd} + M^2_{dd}$$

$$B_{dd} = B^1_{dd} + B^{2x}_{dd}$$

Equation 6-19.

ESTF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTF'/

$$\Delta K_{dd}^{2f} = ig\Delta K_{dd}^{1f}$$

$$\Delta K_{dd}^{4f} = g_e^f[\Delta k_j^f] \quad j = 1 \rightarrow 6$$

$$\Delta B_{dd}^{2f} = \Delta B_{dd}^{1f}$$

ESTNF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTNF'/

$$\Delta K_{dd}^{4f\,nominal} = [\Delta g_e^f \; k_j^{f\,nominal}] \quad j = 1 \rightarrow 6$$

and j runs through the stiffness values specified for the CBUSH element or j = 1 for the CELAS1 and CELAS3 elements.

Then at each frequency form:

$$K_{dd} \leftarrow K_{dd} + \Delta K_{dd}^f + i\Delta K_{dd}^{4f} + i\Delta K_{dd}^{4f\,nominal}$$

$$B_{dd} \leftarrow B_{dd} + \Delta B_{dd}^f$$

Then the equation to be solved is:

$$[-\omega^2 \; M_{dd} + i\omega \; B_{dd} + K_{dd}][u_d] = [P_d]$$

Modal Frequency Response

Basis Vector and Nominal Values

The basis vector matrix [φ] (system modes) is formed only once in the analysis using nominal values for frequency dependent elements.

The following matrices are formed only once in the analysis and are based on the parameter input to EMG of /' '/ implying that the nominal values only are to be used for frequency dependent springs and dampers.

$$K_{dd}^2 = K_{dd}^{2x} + igK_{dd}^1 + iK_{dd}^4$$

$$B_{dd}^2 = B_{dd}^1 + B_{dd}^{2x}$$

ESTF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTF'/

$$\Delta K_{dd}^{2f} = ig\Delta K_{dd}^{1f}$$

$$\Delta K_{dd}^{4f} = g_e^f[\Delta k_j^f] \quad j = 1 \rightarrow 6$$

$$\Delta B_{dd}^{2f} = \Delta B_{dd}^{1f}$$

ESTNF

The following matrices are formed at each frequency in the analysis and are based on the parameter input to EMG of /'ESTNF'/

$$\Delta K_{dd}^{4f\,nominal} = [\Delta g_e^f \; k_j^{f\,nominal}] \quad j = 1 \rightarrow 6$$

and j runs through the stiffness values specified for the CBUSH element or j = 1 for the CELAS1 and CELAS3 elements.

Then at each frequency form:

$$K_{dd}^2 \leftarrow K_{dd}^2 + \Delta K_{dd}^f + i\Delta K_{dd}^{4f} + i\Delta K_{dd}^{4f\,nominal}$$

$$B_{dd}^2 \leftarrow B_{dd}^2 + \Delta B_{dd}^f$$

The GKAM module will then produce:

$$K_{hh} = [k] + [\Phi]^T[K_{dd}^2][\Phi]$$

$$B_{hh} = [b] + [\Phi]^T[B_{dd}^2][\Phi]$$

$$M_{hh} = [m] + [\Phi]^T[M_{dd}^2][\Phi]$$

Then the equation to be solved is:

$$[-\omega^2 M_{hh} + i\omega B_{hh} + K_{hh}][u_h] = [P_h]$$

Element force calculation:

Frequency:

$$F_e = [(1 + ig_t)K_e + i\omega B_e]U_e$$

where

$$g_t = g + g_e$$

Static

$$F_e = K_e U_e$$

Transient

$$F_e = K_e U_e(t)$$

CBUSH1D Element

The CBUSH1D is a one dimensional version of the BUSH element (without the rigid offsets) and supports large displacements. You define the element with the CBUSH1D and a PBUSH1D entry. You can define several spring or damping values on the PBUSH1D property entry. It is assumed that springs and dampers work in parallel. The element force is the sum of all springs and dampers.

The CBUSH1D element has axial stiffness and axial damping. The element includes the effects of large deformation. The elastic forces and the damping forces follow the deformation of the element axis if there is no element coordinate system defined. The forces stay fixed in the x-direction of the element coordinate system if you define such a system. Arbitrary nonlinear force-displacement and force-velocity functions are defined with tables and equations. A special input format is provided to model shock absorbers.

CBUSH1D Benefits

The element damping follows large deformation. You can conveniently model arbitrary force deflection functions. When the same components of two grid points must be connected, you should use force-deflection functions with the CBUSH1D element instead of using NOLINi entries. The CBUSH1D element produces tangent stiffness and tangent damping matrices, whereas the NOLINi entries do not produce tangent matrices. Therefore, CBUSH1D elements are expected to converge better than NOLINi forces.

CBUSH1D Output

The CBUSH1D element puts out axial force, relative axial displacement and relative axial velocity. It also puts out stress and strain if stress and strain coefficients are defined. All element related output (forces, stresses) is requested with the STRESS case control command for SOL 106.

CBUSH1D Guidelines

CBUSH1D is available in all solution sequences. In static and normal modes solution sequences, the damping is ignored. In linear dynamic solution sequences, the linear stiffness and damping is used. In linear dynamic solution sequences, the BUSH1D damping forces aren't included in the element force output.

In nonlinear solution sequences, the linear stiffness and damping is used for the initial tangent stiffness and damping. When nonlinear force functions are defined and the stiffness needs to be updated, the tangents of the force-displacement and force-velocity curves are used for stiffness and damping. The BUSH1D element is considered to be nonlinear if a nonlinear force function is

defined or if large deformation is turned on (PARAM,LGDISP,1). For a nonlinear BUSH1D element, the element force output is the sum of the elastic forces and the damping forces. The element is considered to be a linear element if only a linear stiffness and a linear damping are defined and large deformation is turned off.

CBUSH1D Limitations

- The BUSH1D element nonlinear forces are defined with table look ups and equations. Equations are only available if the default option ADAPT on the TSTEPNL entry is used, equations are not available for the options AUTO and TSTEP.
- The table look ups are all single precision. In nonlinear, round-off errors may accumulate due to single precision table look ups.
- For linear dynamic solution sequences, the damping forces are not included in the element force output.
- The “LOG” option on the TABLED1 is not supported with the BUSH1D.

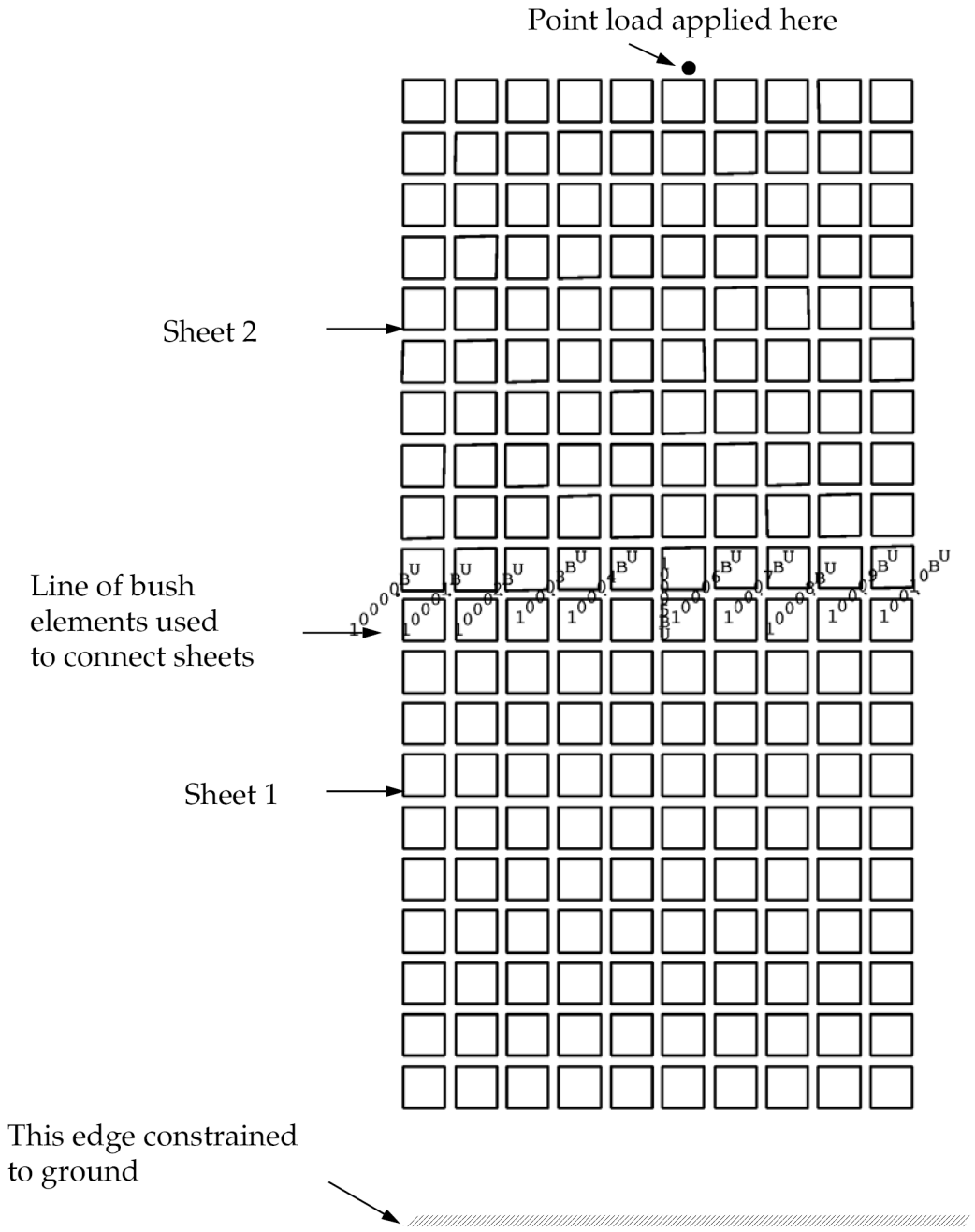


Figure 6-3. Spot-Weld Comparison Model

To use CELASi elements properly, you must account for the offsets between the grid points. The most practical method may be to define coordinate systems, which align with a line between each pair of grid points, and then input the CELASi elements along these coordinate systems, which is a tedious, error-prone task. If the grid points are located in non-Cartesian systems or several Cartesian coordinate systems, the task is even more tedious and error prone. Such small misalignment errors are ignored in this model, and the CELAS2 elements are input in the basic coordinate system. The consequence is that internal constraints are built into the model when, for example, the elements

are offset in the y-direction, are joined by stiffness in the x- and z-directions, and the element has a rotation about the x or z axis.

CBUSH elements are used to join the plates in the second model. The coordinate system of the elastomer must be defined for each element. The option for defining the elastomer coordinate system, which is useable for all geometry including coincident and very close grid points, is the CID option in field 9 of the CBUSH entry. Since the welds are regarded as isotropic, the basic coordinate system is selected by a value of 0. No other consideration of geometry is needed regardless of the coordinate systems used to define grid point locations or displacement system directions.

Use of PARAM,EST,YES provides the length of each element, a modeling check to ensure that the wrong grid points have not been joined or that the misalignment has not been modeled correctly because the length between connected points is greater than the manufacturing tolerance of 0.1 units. If wrong (nonadjacent) points are inadvertently joined by CELASi elements, large internal constraints can be generated that can be difficult to diagnose. CBUSH elements also appear on structure plots.

The input entries for a spot weld are shown for each modeling method in Listing 6-2. The CBUSH element requires one concise line of nonredundant data per weld, plus a common property entry for all elements, while the CELAS2-based model requires six lines per weld. It would require even more input per weld if the geometry were modeled properly.

```
$ INPUT STREAM FOR ELAS2 MODEL OF ONE SPOT WELD
$ SPOT WELD THE EDGES WITH CELAS ELEMENTS
CELAS2,   1,    1.+6,    111000,   1,     210000,  1
CELAS2,   2,    1.+6,    111000,   2,     210000,  2
CELAS2,   3,    1.+6,    111000,   3,     210000,  3
CELAS2,   4,    1.+6,    111000,   4,     210000,  4
CELAS2,   5,    1.+6,    111000,   5,     210000,  5
CELAS2,   6,    1.+6,    111000,   6,     210000,  6
$ INPUT STREAM FOR BUSH MODEL OF ONE SPOT WELD
$ WELD THE EDGES WITH BUSH ELEMENTS
CBUSH,    10000, 1,      311000,   410000,    ,     ,     ,     0
PBUSH,     1,    K       1.+6,     1.+6   1.+6     1.+6   1.+6  1.+6
PARAM, EST, 1 $ PRINT THE MEASURE OF ALL ELEMENTS
```

Listing 6-2. Spot-Weld Models

The OLOAD resultants and the SPC-force resultants for the two models (Figure 6-4) illustrate the effects of internal constraints caused by the misaligned CELASi elements. The first load case is for CELASi modeling and the second line is for CBUSH modeling. When there are no internal constraints, the constraint resultants are equal and opposite to the load resultants. Any unbalance is due to internal constraints.

```
OUTPUT OF LOAD AND SPCFORCE RESULTANTS
SUBCASE 1 IS FOR ELAS ELEMENTS, SUBCASE 2 IS FOR BUSH ELEMENTS

                                                 OLOAD    RESULTANT
                T1             T2             T3             R1             R2             R3
        1   4.0000000E+00  4.0000000E+00  4.0000000E+00  1.6000000E+02 -4.0000000E+01 -1.2000000E+02
        2   4.0000000E+00  4.0000000E+00  4.0000000E+00  1.6000000E+02 -4.0000000E+01 -1.2000000E+02
. . .
                                                 SPCFORCE RESULTANT
                T1             T2             T3             R1             R2             R3
        1  -4.0000000E+00 -4.0000000E+00 -4.0000000E+00 -1.5909122E+02  3.9545681E+01  1.1954554E+02
        2  -4.0000000E+00 -4.0000000E+00 -4.0000000E+00 -1.6000000E+02  4.0000000E+01  1.2000000E+02
```

Figure 6-4. Output of Load and SPC-Force Resultants

The resultants in the CBUSH spot weld model balance to the degree of accuracy shown in the printout. The CELASi-based model resultants match to the degree of accuracy shown in the printout for forces, but only to two or three digits for moments. A review of grid point force balance output shows that the grid points attached to CELASi elements are in balance even though they contain internal constraints. This shows the difficulty in diagnosing elements with internal constraints in model check-out condition activities. Evidence of internal constraints are apparent in static analysis when the resultants are not in balance. The analysis does not isolate the elements with internal constraints; instead it merely states that some internal constraints must exist.

Internal constraints provide plausible results for some loading conditions but may provide unexpected results for other loadings or eigenvectors. There is no direct evidence of internal constraints in element forces, grid point forces, SPC forces or other commonly used output data. The internal constraints are hidden SPC moments, which do not appear in SPC force output. Their effect on this model is to reduce the maximum deflection for this loading condition by 0.65%, and the external work (UIM 5293) by 1% when comparing the CELASi model with the CBUSH model due to the (false) stiffening effect of the internal constraints on rotation at the interface between the two sheets of metal.

Another method to model spot welds is through the use of RBAR elements. RBAR elements do not have internal constraints. The CBUSH element should result in lower computation time than the RBAR element when the model contains many spot welds.

In dynamic analysis, the CBUSH elements can be used as vibration control devices that have impedance values (stiffness and dampling) that are frequency dependent.

6.3 CWELD Connector Element

The CWELD element and corresponding PWELD property entries let you establish connections between points, elements, patches, or any of their combinations. Although there are a number of different ways to model structural connections and fasteners in NX Nastran, such as with CBUSH or CBAR elements or RBE2s, CWELDS are generally easy to generate, less error-prone, and always satisfy the condition of rigid body invariance.

CWELD Connectivity Definition

You define the connectivity for a weld element with the CWELD Bulk Data entry. You can create connections conventionally, from point-to-point, or in a more advanced fashion between elements and/or patches of grid points. In the case of elements and patches, actual weld attachment points

will usually occur within element domains or patches and will be computed automatically, with corresponding automatic creation of necessary grid points and degrees-of-freedom. Element- and patch-based connections, moreover, eliminate the need for congruent meshes. Reference grids that determine spot weld spacing, for example, can be defined beforehand which, when projected (using the CWELD entry) through the surfaces to be attached, uniquely determine the weld elements' location and geometry.

See Also

- "CWELD" in the *NX Nastran Quick Reference Guide*

With the CWELD element, you can choose between three different conenctivity options:

- patch-to-patch
- point-to-patch
- point-to-point

Using the Patch-to-Patch Method

In the context of CWELD element definition, a patch is a surface to which the weld element will connect. Actually, two patches must be defined on the CWELD entry in order to define a valid connection. The attachment locations on these surfaces are determined by vector projection from a single grid point, GS, also referenced on the CWELD entry. The patches are defined either by reference to a shell element ID or by an ordered list of grid points on a surface. See Figure 6-5.

The "ELEMID" option defines a connection between two shell elements:

```
CWELD, EWID, PWID, GS, "ELEMID", , , , +CWE1
+CWE1, SHIDA, SHIDB
```

EWID is the identification number of the CWELD element and PWID is the identification number of the corresponding PWELD property entry. The projection of a normal vector through grid point GS determines the connection locations on the shell elements SHIDA and SHIDB. The grid point GS does not need to lie on either of the element surfaces.

Instead of shell element IDs, an ordered list of grids could have been used to define a surface patch as seen in the following CWELD example:

```
CWELD, EWID, PWID, GS, "GRIDID", , , "QT", , +CWG1
+CWG1, GA1, GA2, GA3, GA4, , , , ,+CWG2
+CWG2, GB1, GB2, GB3
```

The "GRIDID" option affords a more general approach to surface patch definition, based on an ordered list of grids. Such patches can either be triangular or rectangular with anywhere from three to eight grids points. Ordering and numbering conventions follow directly from the CTRIA3 and CTRIA6 entries for triangular patches, and the CQUAD4 and CQUAD8 entries for rectangular patches (mid side nodes can be omitted). Since the program is generally unable to tell a TRIA6 with two mid side nodes deleted from a QUAD4, it becomes necessary to also indicate the nature of the patches involved, hence the string "QT" in the preceding CWELD example, for "quadrilateral to triangular surface patch connection." (See Remark 5 on the Bulk Data entry, "CWELD" in the *NX Nastran Quick Reference Guide* entry for other options.)

With the patch-to-patch connection, non-congruent meshes of any element type can be connected. Patch-to-patch connections are recommended when the cross sectional area of the connector is larger than 20% of the characteristic element face area.

The patch-to-patch connection can also be used to connect more than two layers of shell elements. For example, if three layers need to be connected, a second CWELD element is defined that refers to the same spot weld grid GS as the first CWELD. Patch B of the first CWELD is repeated as patch A in the second CWELD.

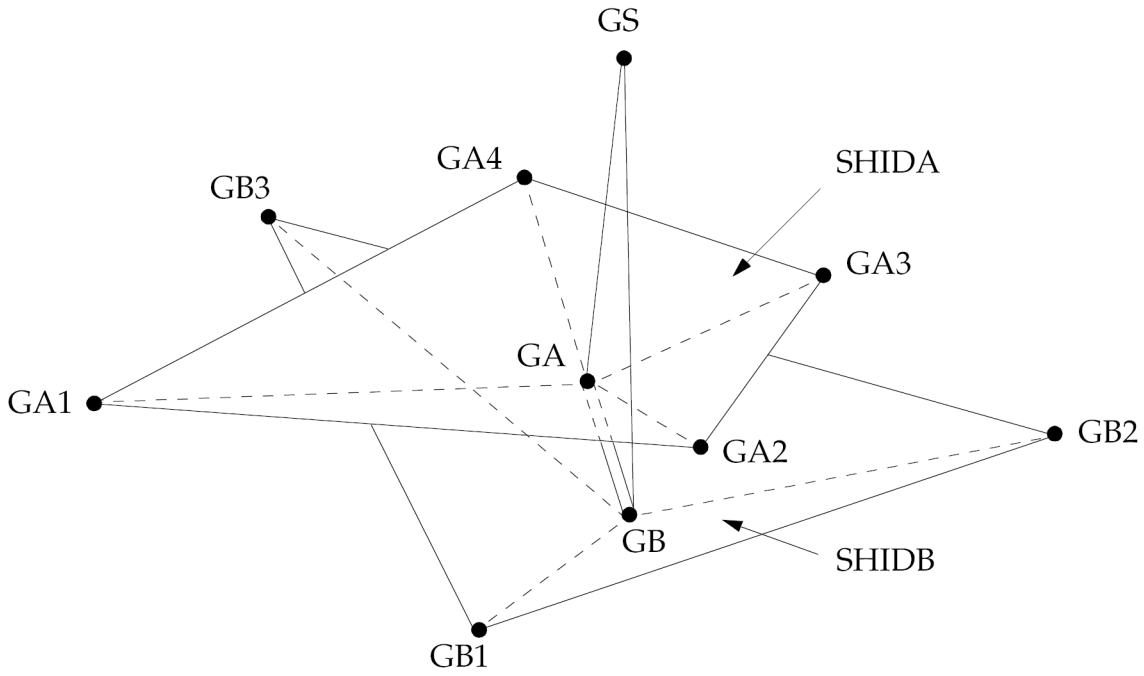


Figure 6-5. Patch-to-Patch Connection Defined with Format GRIDID or ELEMID

Using the Point-to-Patch Option

The “GRIDID” and “ELEMID” formats can also be used to define the connection of a point to a surface patch and again, is a useful method of joining non-congruent meshes. As before, the patch can either be triangular or quadrilateral:

```
CWELD, EWID, PWID, GS, "GRIDID", , , "Q", , +CWP1
+CWP1, GA1, GA2, GA3, GA4
```

For a point-to-patch connection, the vertex grid point GS of a shell element is connected to a surface patch, as shown in Figure 6-6. The patch is either defined by grid points GAi or a shell element SHIDA. A normal projection of grid point GS on the surface patch A creates grid GA. The vector from grid GA to GS defines the element axis and length. A shell normal in the direction of the element axis is automatically generated at grid GS. Point-to-patch connections are recommended if a flexible shell (GS) is connected to a stiff part (GAi).

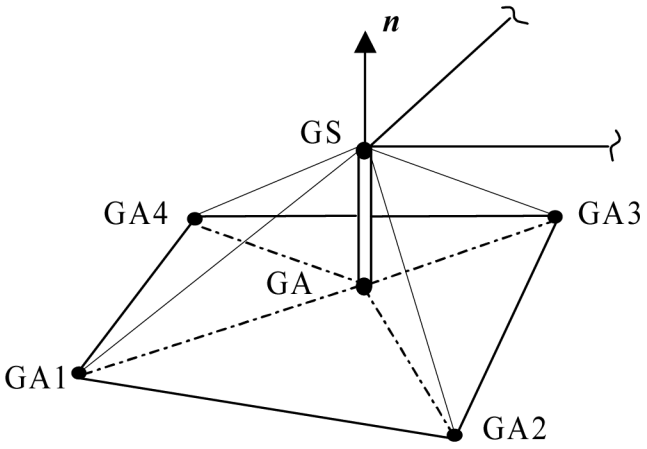


Figure 6-6. Point-to-Patch Connection

Using the Point-to-Point Method

The “ALIGN” format defines a point-to-point connection:

```
CWELD, EWID, PWID, GS, “GRIDID”, , , “Q”, , +CWP1
+CWP1, GA1, GA2, GA3, GA4
```

In the point-to-point connection, two shell vertex grids, GA and GB, are connected as shown in Figure 6-7. The vector from GA to GB determines the axis and length of the connector. Two shell normals are automatically generated for the two shell vertex grids, and both normals point in the direction of the weld axis.

You can only use the point-to-point connection type for shell elements when the two layers of shell meshes are nearly congruent. The element axis (the vector from GA to GB) must be nearly normal to the shell surfaces. You should only use the point-to-point connection when the cross sectional area of the connector doesn’t exceed 20% of the characteristic shell element area.

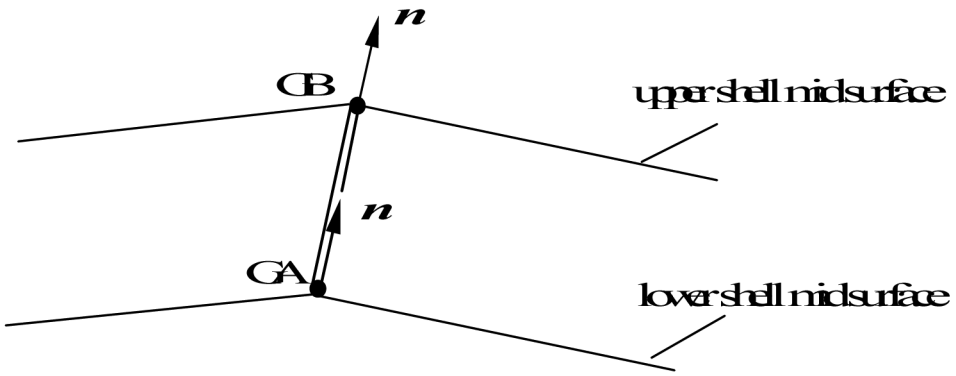


Figure 6-7. Point-to-Point Connection

Defining the Properties of the CWELD

MID references a material entry, D is the diameter of the element, and MSET is used to indicate whether the generated constraints will explicitly appear in the *m* -set or be reduced out at the element level prior to global stiffness matrix assembly. Further parameters and flags will more than likely be introduced in the future as additional options are implemented.

CWELD element constraint generation for patch-based connections are under some measure of your control. When MSET=ON is defined on the PWELD entry, the constraints are expressed as multi-point constraint equations. For output purposes, the additional *m* -set constraints are labeled “RWELD”, with identification numbers beginning with the ID specified on the PARAM, OSWELM entry.

Constraint equations can be incorporated directly into the element stiffness matrices. The MSET=OFF option (default) instead applies the CWELD element constraint set locally, without generating additional *m* -set equations. The local constraint effects are reduced out prior to system stiffness matrix assembly resulting in a more compact set of equations, and the expected benefits in numerical performance and robustness. The MSET = OFF option is only available for the patch-to-patch connection (GRIDID or ELEMID).

TYPE identifies the type of connection: “Blank” for a general connector, and “SPOT” for a spot weld connector. The connection type influences effective element length in the following manner: If the format on the CWELD entry is “ELEMID”, and TYPE= “SPOT”, then regardless of the distance GA to GB, the effective length L_e of the spot weld is set to $L_e = 1/2(t_A + t_B)$ where t_A and t_B are the thickness of shell *A* and *B* , respectively. If TYPE is left blank (a general connector), then the length of the element is not modified 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 $L_e = 0.2D$. If *L* is above this range, the effective length is instead set to $L_e = 5.0D$.

Finite Element Representation of the Connector

For all connectivity options, the CWELD element itself is modeled with a special shear flexible beam-type element of length L and a finite cross-sectional area which is assumed to be a circle of diameter D, as shown in Figure 6-8. Other cross sections will be added in the future. The length L is the distance from GA to GB. An effective length is computed if TYPE=“SPOT” or if grids GA and GB are coincident. The element has twelve (2x6) degrees-of-freedom.

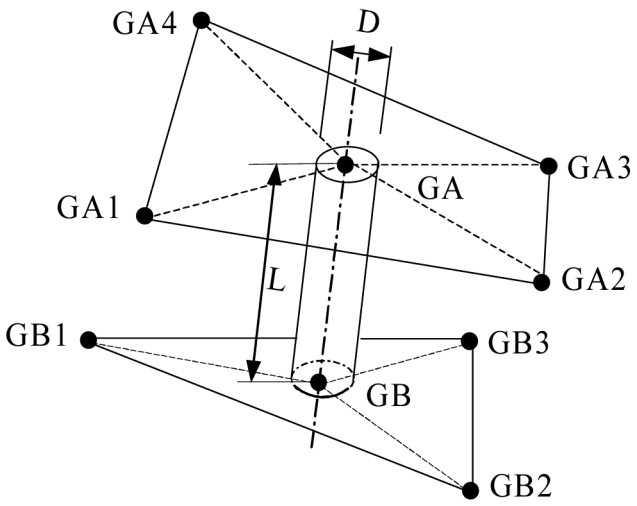


Figure 6-8. CWELD Element

For the point-to-patch and patch-to-patch connection, the degrees-of-freedom of the spot weld end point GA are constrained as follows: the 3 translational and 3 rotational degrees-of-freedom are connected to the 3 translational degrees-of-freedom of each grid GAi with constraints from Kirchhoff shell theory,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_A = \sum N_i(\xi_A, \eta_A) \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_i$$

Equation 6-20.

$$\theta_x^A = \frac{\partial w}{\partial y} = \sum N_{i,y} \cdot w_i$$

$$\theta_y^A = -\frac{\partial w}{\partial x} = -\sum N_{i,x} \cdot w_i$$

$$\theta_z^A = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = \frac{1}{2}\left(\sum N_{i,x} \cdot v_I - \sum N_{i,y} u_i\right)$$

Equation 6-21.

These 6 equations are written in the local tangent system of the surface patch at point GA. The two tangent directions are x and y, and the normal direction is z. N_i are the shape functions of the surface patch; ξA and ηA are the normalized coordinates of GA; u, v, w are the displacements; and θx, θy, θz are the rotations in the local tangent system at GA. For the patch-to-patch connection, another set of 6 equations similar to Eq. 6-20 and Eq. 6-21 is written to connect grid point GB to GBi. The patch-to-patch connection results in 12 constraint equations.

In summary, with MSET="ON", the CWELD element consists of a two node element with 12 degrees-of-freedom. In addition, 6 constraint equations are generated for the point-to-patch

connection or 12 constraint equations for the patch-to-patch connection. The degrees-of-freedom for GA and GB are put into the dependent set (*m* -set).

If a patch-to-patch connection is specified and MSET="OFF", the 12 constraint equations are included in the stiffness matrix instead, and the degrees-of-freedom for GA and GB are condensed out. No *m* -set degrees-of-freedom are generated and the subsequent, sometimes costly, *m* -set constraint elimination is avoided. The resulting element is 3xN degrees-of-freedom, where N is the total number of grids GAi plus GBi. This maximum total number of grids is 16, yielding an element with a maximum 48 degrees-of-freedom.

CWELD Results Output

The element forces are output in the element coordinate system, see Figure 6-9. The element x-axis is in the direction of GA to GB. 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 CWELD element force output and sign convention is the same as for the CBAR element.

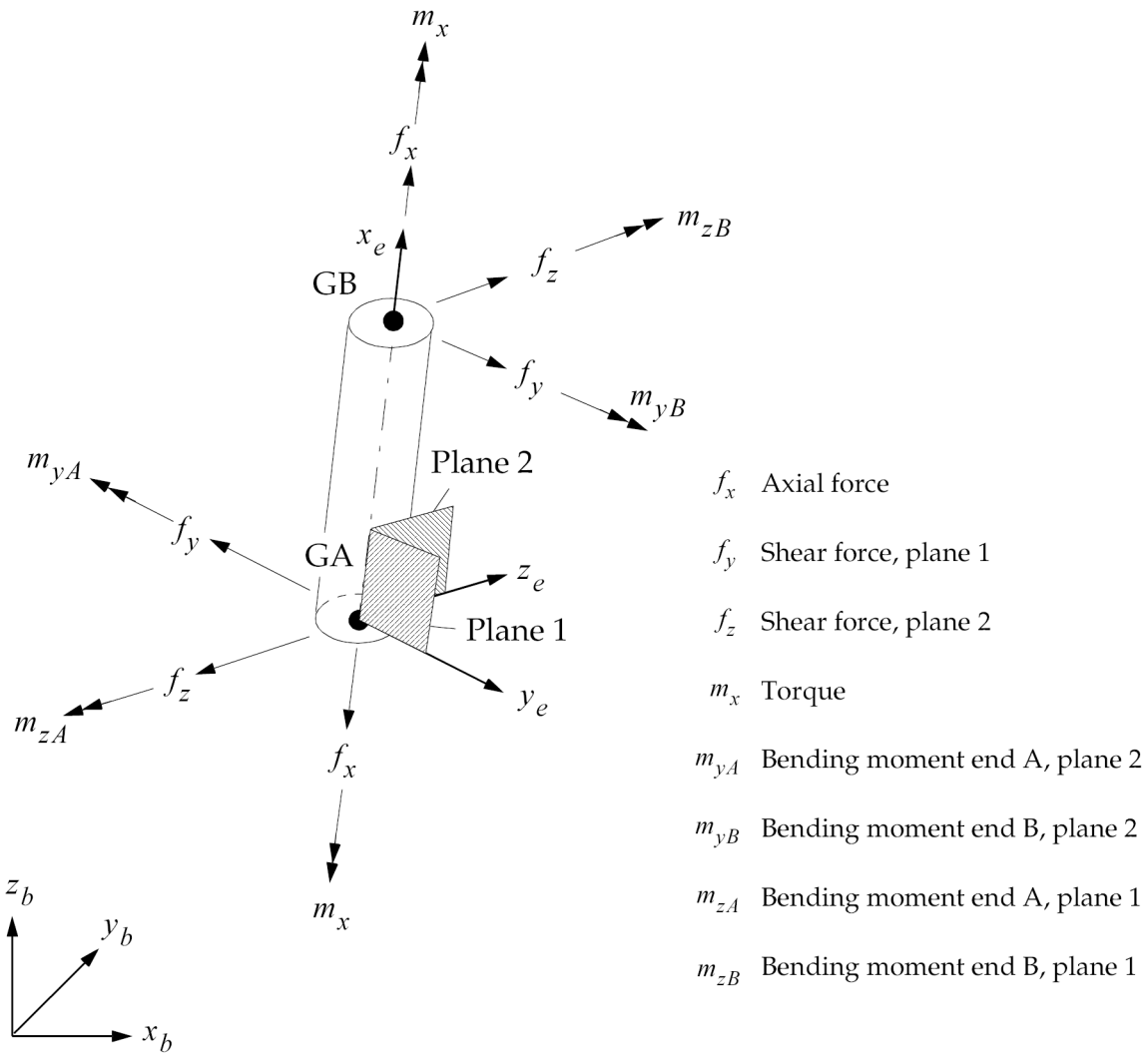


Figure 6-9. Element Coordinate System and Sign Convention of Element Forces

Example (CWELD101a-b.DAT)

In the following example, two cylindrical shell segments are connected with 4 CWELD elements at the 4 corners of the overlapping sections, see Figure 6-10. The results of the patch-to-patch connection are compared to the results of the point-to-point connection. For the patch-to-patch connection, we place the four CWELD elements on the corner shells of the overlapping area. For the point-to-point connection, we take the inner vertex points of the corner shells. The deflection at grid point 64 for the patch-to-patch connection is lower (stiffer) than from the point-to-point connection (1.6906 versus 1.9237). The difference is significant in this example because of the coarse mesh and because the connection of the two shells is modeled with only 4 welds. In most practical problems, the patch-to-patch connection produces stiffer results than the point-to-point connection.

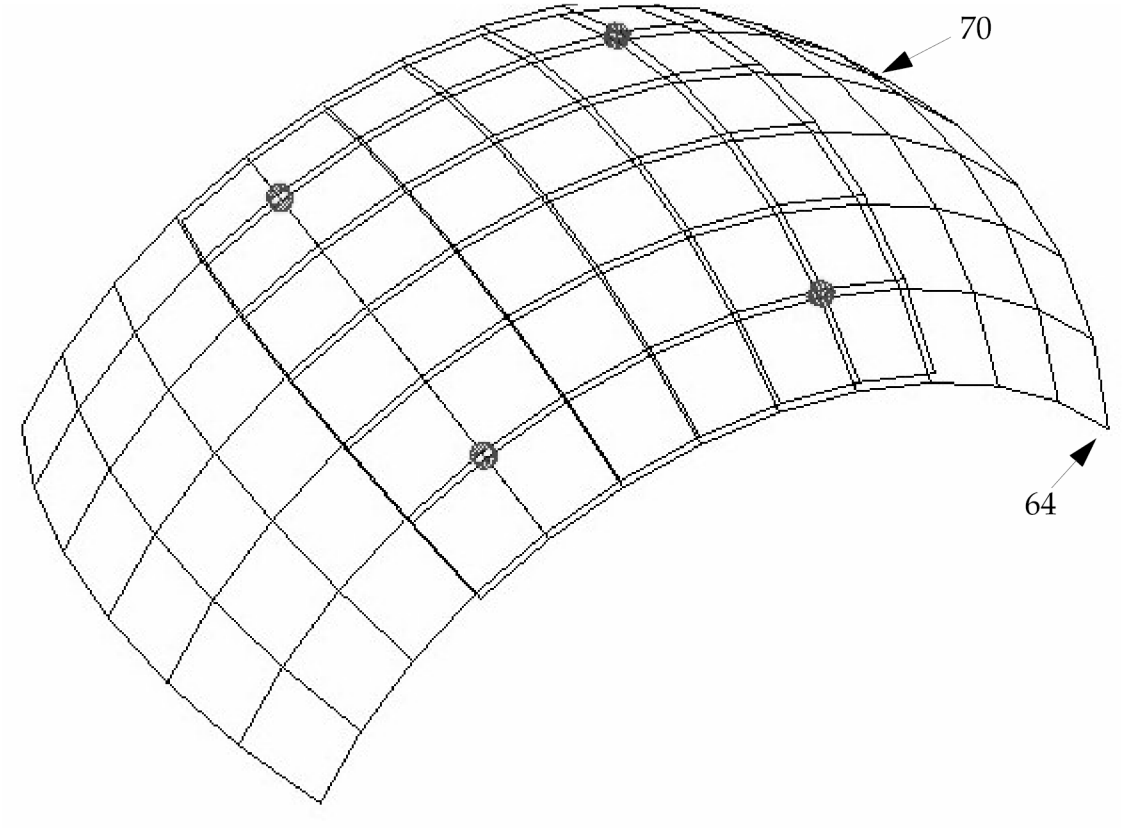


Figure 6-10. Two Spherical Segments with Spot Welds at the Four Corners

Common Input

```
$
$ Nodes of the connected elements
$
GRID     1               21.6506 37.5   -25.     1
GRID     2               23.4923 40.6899-17.101  1
GRID     6               23.4923 40.6899 17.101  1
GRID     7               21.6506 37.5    25.     1
GRID     8               14.8099 40.6899-25.     1
GRID     9               16.0697 44.1511-17.101  1
GRID     13              16.0697 44.1511 17.101  1
GRID     14              14.8099 40.6899 25.     1
GRID     36             -14.81   40.6899-25.     1
GRID     37             -16.0697 44.1511-17.101  1
GRID     41             -16.0697 44.1511 17.101  1
GRID     42             -14.81   40.6899 25.     1
GRID     43             -21.6507 37.5   -25.     1
GRID     44             -23.4924 40.6899-17.101  1
GRID     48             -23.4924 40.6899 17.101  1
GRID     49             -21.6507 37.5    25.     1
GRID     92              22.1506 38.3661-25.     1
GRID     93              23.9923 41.5559-17.101  1
GRID     97              23.9923 41.5559 17.101  1
```

```
GRID     98              22.1506 38.366  25.     1
GRID     99              15.1519 41.6296-25.     1
GRID     100             16.4117 45.0908-17.101  1
GRID     104             16.4117 45.0908 17.101  1
GRID     105             15.1519 41.6296 25.     1
GRID     127            -15.152  41.6296-25.     1
GRID     128            -16.4118 45.0908-17.101  1
GRID     130            -17.4431 47.9243 2.435-5 1
GRID     131            -17.1833 47.2105 8.68244 1
GRID     132            -16.4118 45.0908 17.101  1
GRID     133            -15.152  41.6296 25.     1
GRID     134            -22.1506 38.366 -25.     1
GRID     135            -23.9923 41.5559-17.101  1
GRID     139            -23.9923 41.5559 17.101  1
GRID     140            -22.1506 38.366  25.     1
GRID     141             0.      0.      0.      2
$
$ Grid points GS for spot weld location
$
GRID     142             16.2407 44.6209-17.101
GRID     143             16.2407 44.6209 17.101
GRID     144            -16.2407 44.6209-17.101
GRID     145            -16.2407 44.6209 17.101
$
$ Referenced Coordinate Frames
$
CORD2C   1               0.      0.      0.      0.      0.      1.
1.217-8 1.      0.
CORD2R   2               0.      0.      0.      0.      0.      1.
1.      0.      0.
$
$ CQUAD4 connectivities
$
CQUAD4   1       1       1       2       9       8
CQUAD4   6       1       6       7       14      13
CQUAD4   31      1       36      37      44      43
CQUAD4   36      1       41      42      49      48
CQUAD4   73      1       92      93      100     99
CQUAD4   78      1       97      98      105     104
CQUAD4   103     1       127     128     135     134
CQUAD4   108     1       132     133     140     139
$
$ shell element property t=1.0
$
PSHELL   1       1       1.      1               1
$
$ Material Record : Steel
$
MAT1     1       210000.         .3      7.85-9
$
$ PWELD property with D= 2.0
$
PWELD    200     1 2.
```

Input for the Patch-to-Patch Connection

```
$
$ CWELD using ELEMID option, outer diagonal elements
$
CWELD     109     200     142     ELEMID
          1       73
CWELD     110     200     143     ELEMID
          6       78
CWELD     111     200     144     ELEMID
          31      103
CWELD     112     200     145     ELEMID
          36      108
```

Input for the Point-to-Point Connection

```
$
$CWELD  ALIGN option
$
CWELD    109        200            ALIGN  9       100
CWELD    110        200            ALIGN  13      104
CWELD    111        200            ALIGN  37      128
CWELD    112        200            ALIGN  41      132
```

Output of Element Forces

0 SUBCASE 1

FORCES IN WELD ELEMENTS (CWELD)

ELEMENT	BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL	
ID	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX
109	-2.135744E+02	1.586358E+02	7.728133E+01	1.390482E+01	-2.908656E+02	-1.447359E+02	-8.345229E+01	0.0
110	2.135769E+02	1.586297E+02	-7.728750E+01	1.390158E+01	2.908643E+02	-1.447281E+02	-8.346558E+01	0.0
111	-2.363556E+02	-3.536731E+02	-2.396099E+02	-2.361487E+02	3.254667E+00	1.175355E+02	-1.198793E+01	0.0
112	2.363570E+02	-3.536719E+02	2.396101E+02	-2.361489E+02	-3.253387E+00	1.175340E+02	-1.198794E+01	0.0

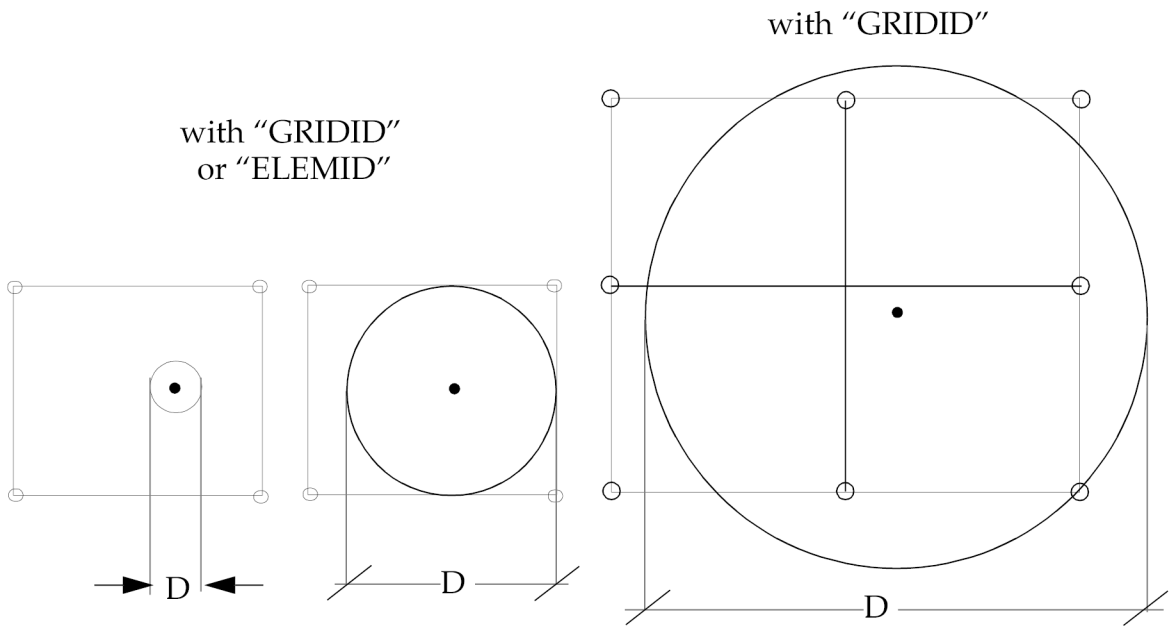


Figure 6-11. Range of Cross-sectional Area versus Element Size for the Patch-to-Patch Connection

- For the patch-to-patch and the point-to-patch connection, the projected grid points GA or GB may lie on an edge of the surface patches or may coincide with a grid point GAi or GBi. The connection is valid as long as GA and GB lie within the surface A and B, respectively. If GA or GB lie outside the surface but inside a tolerance of 5% of the element length, then they are moved on to the surface. In extreme cases, the patch-to-patch type connects elements that are not overlapping (see Figure 6-12). Although the connection is valid, the CWELD may become too stiff.

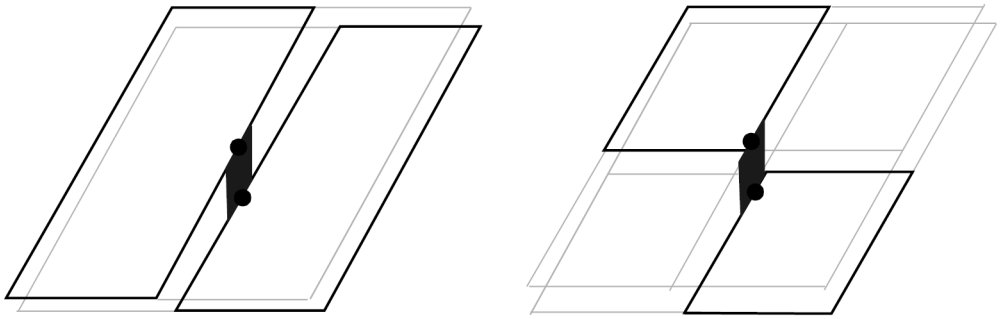


Figure 6-12. Non-overlaping Elements in the Patch-to-Patch Connection

CWELD Guidelines and Remarks

- The new CWELD element is available in all solution sequences.
- CWELD elements are not supported in material or geometric nonlinear analyses.
- SNORM Bulk Data entries aren't necessary.

- PARAM,K6ROT isn't necessary.
- The patch-to-patch connection is sufficiently accurate if the ratio of the cross sectional area to the surface patch area is between 10% and 100%, see Figure 6-11.
- With the GRIDID or ELEMID option, the cross section of the connector may cover up to eight grid points if a quadrilateral surface patch with mid side nodes is defined, see Figure 6-11.

6.4 Gap and Line Contact Elements

In NX Nastran, you can define gap and friction elements with a CGAP entry. The element coordinate system and nomenclature are shown in Figure 6-13. CID is required, if it is used to define the element coordinate system. Otherwise, the *X* -axis of the element coordinate system, x_{elem} , is defined by a line connecting GA and GB of the gap element. The orientation of the gap element is determined by vector $\vec{v}$ similar to the definition of the beam element, which is in the direction from grid points GA to GO or defined by (X1, X2, X3).

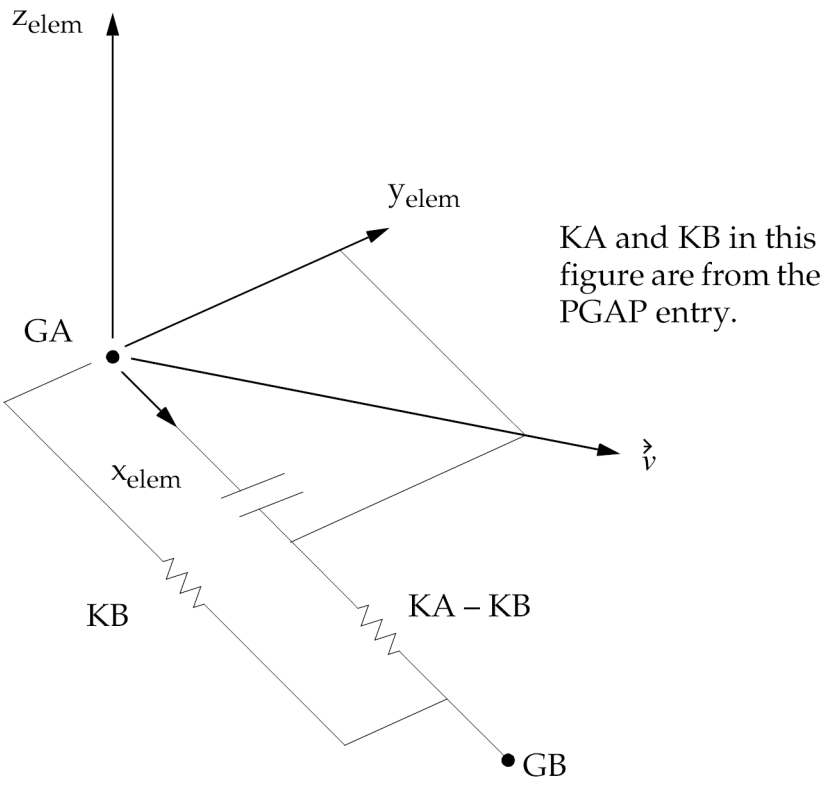


Figure 6-13. Gap Element Coordinate Systems

The properties for the gap elements are defined on the PGAP entry. The initial gap opening is defined by U_0 . If the gap is closed ($U_A - U_B \geq U_0$), the axial stiffness (*KA*) has a very large value (relative to the adjacent structure). When the gap is open, there is a small stiffness *KB* in the axial direction.

NX Nastran includes two types of gap elements: nonadaptive and adaptive. When you use the nonadaptive GAP element, you specify the anisotropic coefficients of friction (μ_1 and μ_2) for the

frictional displacements. Also, the anisotropic coefficients of friction are replaced by the coefficients of static and dynamic friction μ_s and μ_k. On the PGAP continuation entry, the allowable penetration limit T_{max} should be specified because there is no default. In general, the recommended allowable penetration T_{max} is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected by GA and GB. When T_{max} is set to zero, the penalty values will not be adjusted adaptively.

Gap element forces (or stresses) and relative displacements are requested by the STRESS or FORCE Case Control command and computed in the element coordinate system. A positive axial force F_x indicates compression. For the element with friction, the magnitude of the slip displacement is always less than the shear displacement after the slip starts. For the element without friction, the shear displacements and slip displacements have the same value.

See Also

- "CGAP" in the *NX Nastran Quick Reference Guide*
- "Performing a 3-D Slide Line Contact Analysis" in the *NX Nastran Basic Nonlinear Analysis User's Guide*

6.5 Concentrated Mass Elements (CONM1, CONM2)

You can use the concentrated mass elements to define a concentrated mass at a grid point. NX Nastran supports two forms of input of concentrated mass:

- CONM1
- CONM2

The CONM1 allows a general 6 × 6 symmetric mass matrix in a specified coordinate system to be assigned to a geometric grid point.

The CONM2 element allows a concentrated mass about its center of gravity to be specified. CONM2 lets you specify the offset of the center of gravity of the concentrated mass relative to grid point location, a reference coordinate system, the mass and a 3 × 3 symmetric matrix of mass moments of inertia measured from its center of gravity.

See Also

- "CONM1" in the *NX Nastran Quick Reference Guide*
- "CONM2" in the *NX Nastran Quick Reference Guide*

6.6 p-Elements

p-elements are elements that have variable degrees-of-freedom. You can specify the polynomial order for each element (Px, Py, and Pz), and NX Nastran generates the degrees-of-freedom required.

There are a total of six different forms of p-elements you can define with connection entries:

CTETRA	Four-sided solid element specified by 4 corner grid points
CPENTA	Five-sided solid element specified by 6 corner grid points

CHEXA	Six-sided solid element specified by 8 corner grid points
CTRIA	Three-sided curved shell element specified by 3 corner grid points
CQUAD	Four-sided curved shell element specified by 4 corner grid points
CBEAM	A curved beam element specified by its 2 end grid points

You specify the properties of p-elements using the PSOLID, PSHELL, PBEAM, or PBEAML entry. These elements may use either isotropic materials as defined on the MAT1 entry or anisotropic materials as defined on the MAT9 entry. The material coordinate system can use the basic system (0), any defined system (integer > 0), or the element coordinate system (-1 or blank).

See Also

- “p-Elements” in the *NX Nastran User’s Guide*

p-Element Geometry

You can model the geometry of the p-elements by:

- Supplying additional POINT entries to define the geometry of the edges. For example, in the FEEDGE entry, you need to specify GEOMIN = POINT. In this case, when the FEEDGE entry contains the ID of one point, then the FEEDGE is considered to have a quadratic geometry. When the FEEDGE entry contains the ID of two points, then the FEEDGE is considered to have a cubic geometry. If there are no additional points specified on the FEEDGE entry, then the FEEDGE is considered to have a linear geometry (see Figure 6-14).

- Supplying the actual geometry, using the GMCURV and GMSURF entries, to define the geometry of the edges. You need to specify GEOMIN = GMCURV in the FEEDGE entry and providing a SURFID on the FEFACE entry (see Figure 6-15).

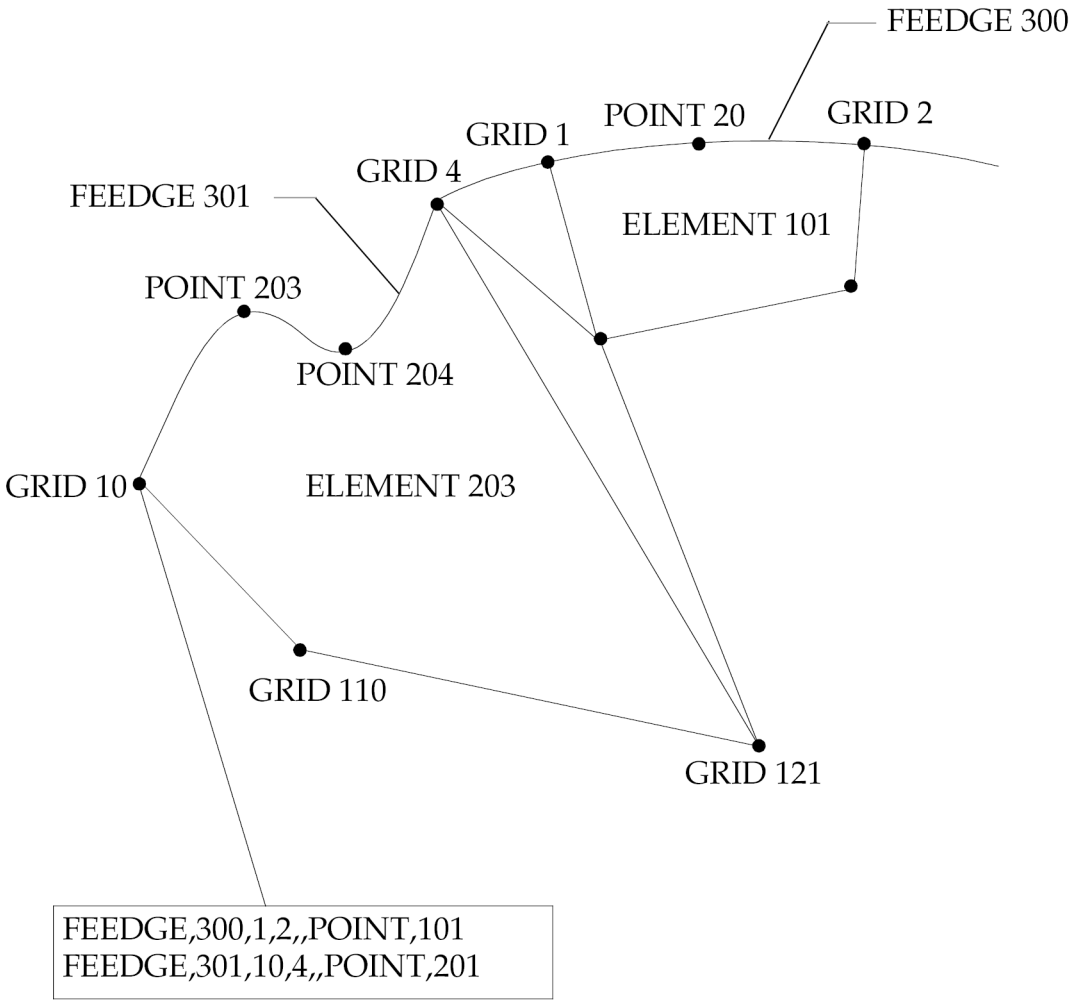


Figure 6-14. Specifying Geometry Using GEOMIN = POINT Method

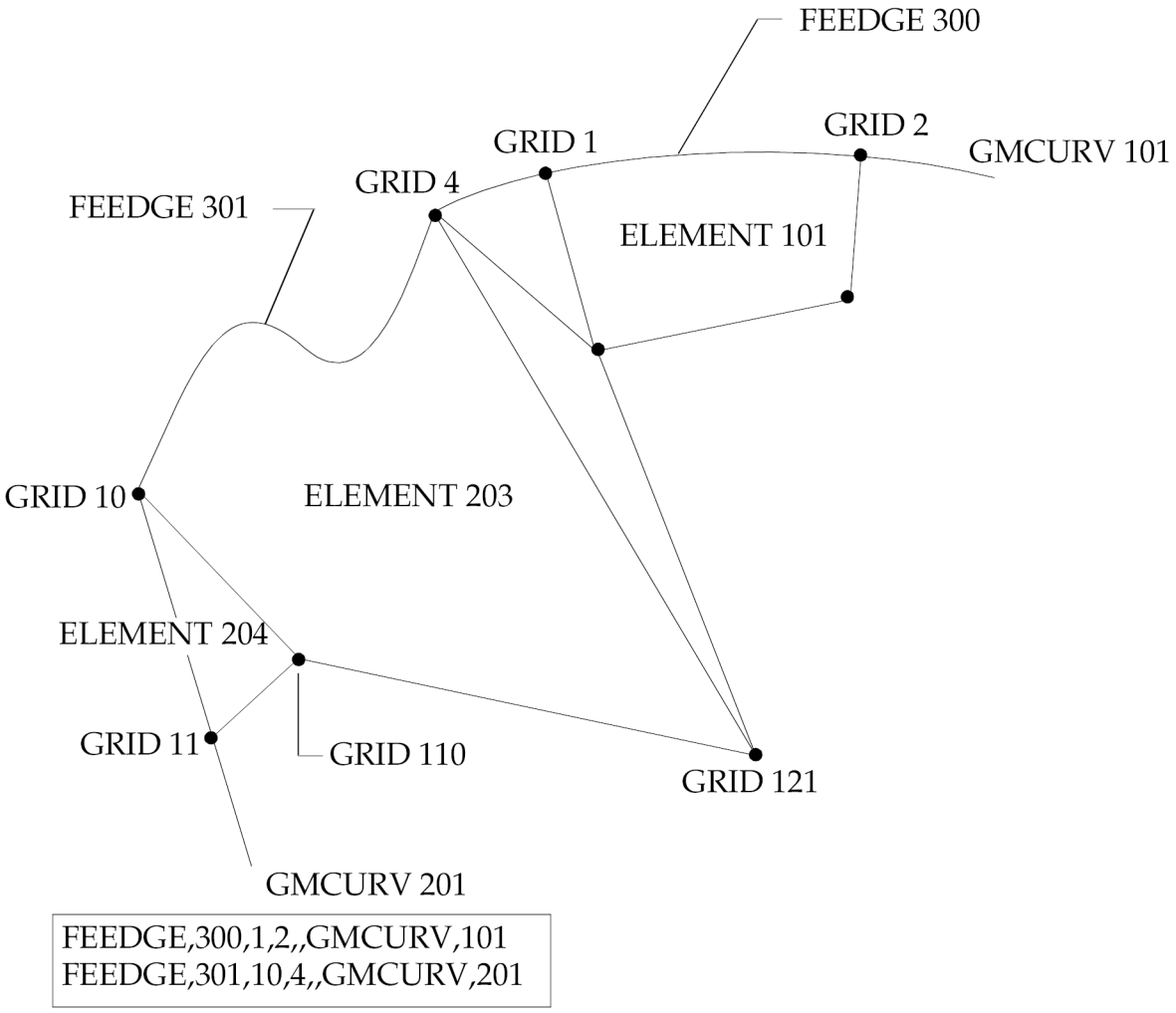


Figure 6-15. Specifying Geometry Using GEOMIN = GMCURV Method

In general, the geometry specification for any edge using a FEEDGE entry overrides the geometry specification for that edge via the FEFACE entry. In other words, the geometry of the edges belonging to both a FEEDGE and a FEFACE will be calculated using the data supplied on the FEEDGE entry. In addition, whenever two or more GMSURF are intersecting and FEFACE entries referring to the individual GMSURF entries are supplied, an additional FEEDGE entry must be supplied for the edges that are common to the multiple surfaces.

Bulk entries defining p-elements with p-valve greater than 1 are:

ADAPT	GMSPC
DEQATN	GMSURF
FEEDGE	OUTPUT
FEFACE	OUTRCV
GMBC	PSET
GMCORD	PVAL
GMCURV	TABLE3D
GMLOAD	TEMPF

6.7 Hyperelastic Elements

The hyperelastic elements are intended for fully nonlinear (finite deformation) analysis including the effect of large strain and large rotation. Geometric nonlinearity is a subset of this type of analysis. In addition, the elements are especially designed to handle nonlinear elastic materials at the nearly incompressible limit. Volumetric locking avoidance is provided through a mixed formulation, based on a three field variational principle, with isoparametric displacement and discontinuous pressure and volumetric strain interpolations. Shear locking avoidance is provided through the use of second order elements.

You define the hyperelastic elements on the same connection entries as the other shell and solid elements. They are distinguished by their property entries. A PLPLANE or PLSOLID entry defines a hyperelastic element. The hyperelastic material, which is characterized by a generalized Rivlin polynomial form of order 5, applicable to compressible elastomers, is defined on the MATHP entry.

See Also

- "Elements for Nonlinear Analysis" in the *NX Nastran Basic Nonlinear Analysis User's Guide*

Hyperelastic Solid Elements

The following elements are available:

- CTETRA – Four-sided solid element with 4 to 10 nodes.
- CPENTA – Five-sided solid element with 6 to 15 nodes.
- CHEXA – Six-sided solid element with 8 to 20 nodes.

There's no element coordinate system associated with the hyperelastic solid elements. All output is in the basic coordinate system. The following quantities are output at the Gauss points:

- Cauchy stresses

$$\sigma_x,\ \sigma_y,\ \sigma_z,\ \tau_{xy},\ \tau_{yz},\ \tau_{zx}$$

- Pressure

$$p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

- Logarithmic strains

$$\varepsilon = \sum_{l=1}^{3} ln\gamma_l N_l N_l^T$$

- Volumetric strain

$$J - 1 = \frac{dV - dV_0}{dV_0}$$

See Also

- “Solid Elements (CTETRA, CPENTA, CHEXA)”

Hyperelastic Plane Elements

These are plane strain elements defined on the following connectivities:

- CQUAD – Quadrilateral element with 4 to 9 nodes. When the center node is missing, this element may also be specified on a CQUAD8 connectivity entry. When all edge nodes are missing, the CQUAD4 connectivity may be used.
- CTRIA3 – Triangular element with 3 nodes.
- CTRIA6 – Triangular element with 3 to 6 nodes.

Figure 6-16 shows the element connectivity for the CQUAD element.

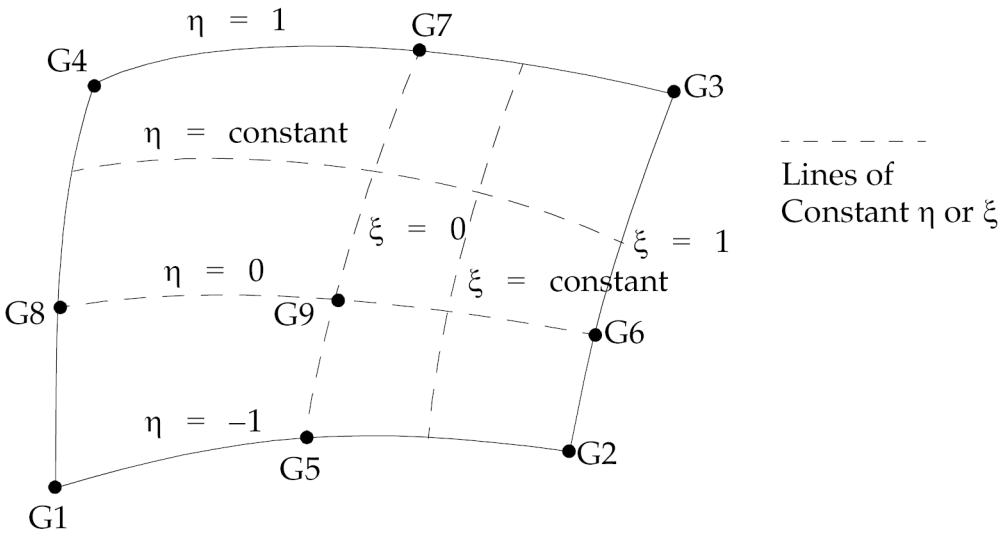


Figure 6-16. CQUAD Element

Note, however, that there is no element coordinate system associated with the hyperelastic plane elements. All output is in the CID coordinate system. Cauchy stresses σ_x, σ_y, σ_z, τ_{xy}, pressure $p = 1/3(\sigma_x + \sigma_y + \sigma_z)$, logarithmic strains and volumetric strain are output at the Gauss points.

The plane of deformation is the XY plane of the CID coordinate system, defined on the PLPLANE property entry. The model and all loading must lie on this plane, which, by default, is the XY plane of the basic coordinate system. The displacement along the Z axis of the CID coordinate system is zero or constant.

Hyperelastic Axisymmetric Elements

The axisymmetric hyperelastic elements are defined on the following connectivity entries:

- CTRAX3, CTRAX6, CQUADX4, CQUADX8 elements (described under the 3D element chapter).
- CQUADX – Quadrilateral axisymmetric element with 4 to 9 nodes.

- CTRIAX – Triangular axisymmetric element with 3 to 6 nodes.

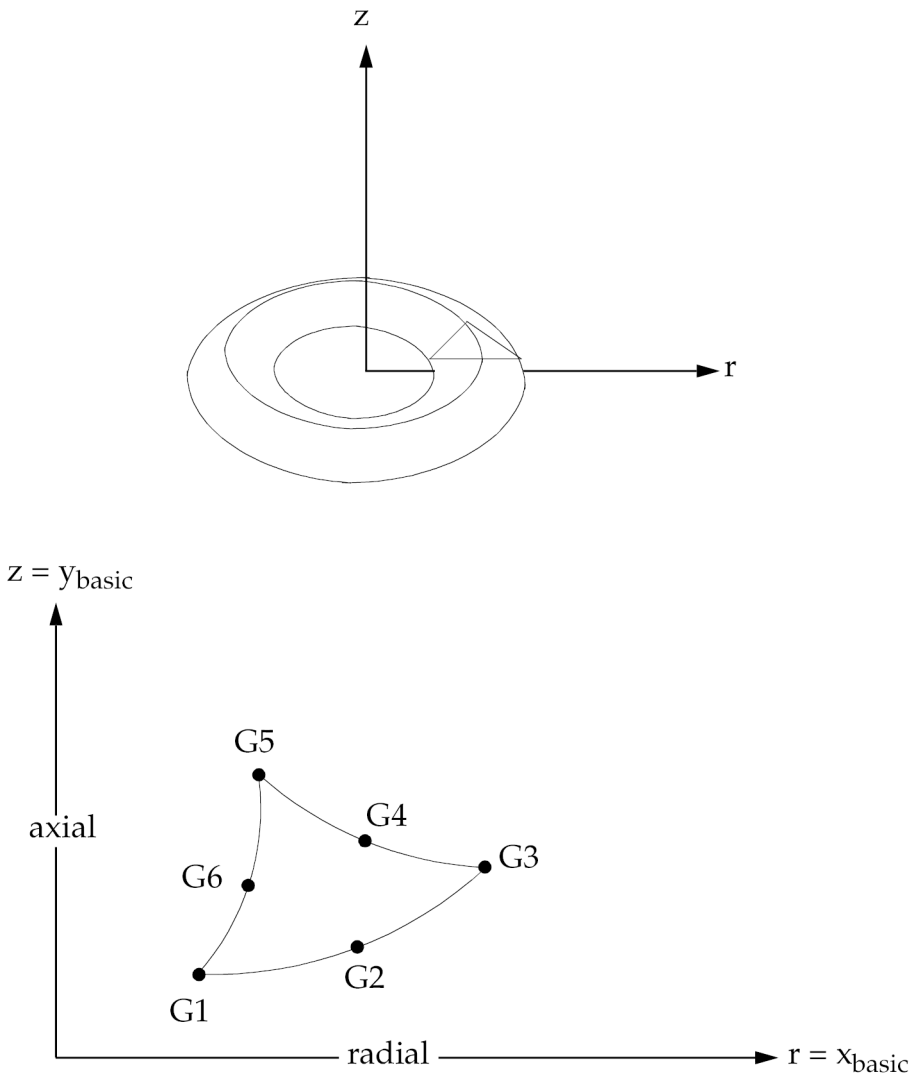


Figure 6-17. The CTRIAX Hyperelastic Coordinate System Element

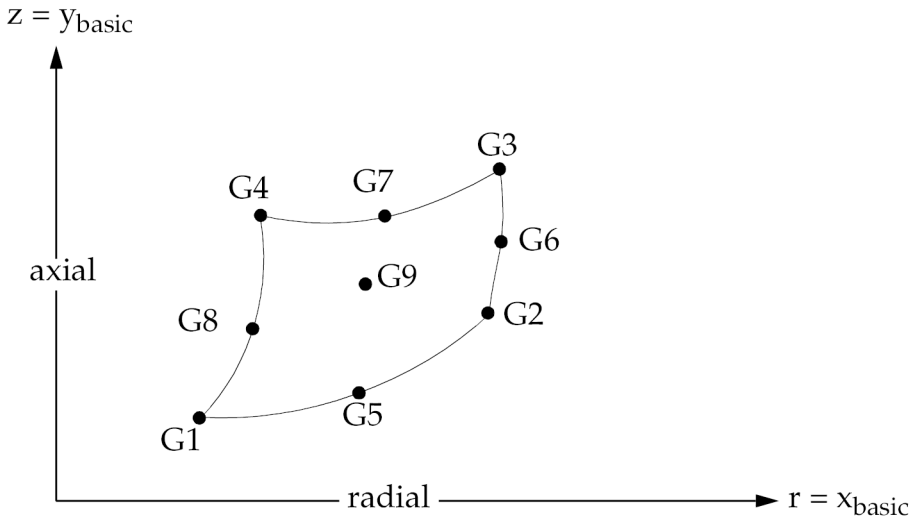


Figure 6-18. The CQUADX Hyperelastic Coordinate System Element

CTRIAX and CQUADX

The plane of deformation is the XY plane of the basic coordinate system, and Y is the axisymmetry axis. The model and all loading must lie on this plane.

All output is in the basic coordinate system. Cauchy stresses σ_x (radial), pressure σ_y (axial), σ_z (circumferential), τ_{xy} , pressure $p = 1/3(\sigma x + \sigma y + \sigma z$, logarithmic strains and volumetric strain are output at the Gauss points.

Pressure loads, specified on the PLOAD4 and PLOADX1 Bulk Data entries respectively, with follower force characteristics, are available for the solid and axisymmetric elements. A pressure load may not be specified on the plane strain elements.

Temperature loads may be specified for all hyperelastic elements on the TEMP and TEMD entries. The hyperelastic material, however, may not be temperature dependent. Temperature affects the stress-strain relation.

GPSTRESS and FORCE (or ELFORCE) output isn't available for hyperelastic elements.

6.8 Interface Elements

The interface elements are primarily used when performing global local analyses. The current implementation is for p-elements.

For curve interfaces you use the following bulk data entries:

- GMBNDC – Geometric Boundary - Curve
- GMINTC – Geometric Interface - Curve
- PINTC – Properties of Geometric Interface - Curve

For surface interfaces you use the following bulk data entries:

- GMBNDS – Geometric Boundary - Surface

- GMINTS – Geometric Interface - Surface
- PINTS – Properties of Geometric Interface - Surface

The interface elements use a hybrid variational formulation with Lagrange multipliers, developed by NASA Langley Research Center. There are displacement variables defined on the interface element, in order to avoid making the interface too stiff, such as a rigid element. There are also Lagrange multipliers defined on each boundary, which represent the forces between the boundaries and the interface element. This formulation is energy-based and results in a compliant interface.

Curve Interface Elements

Interface elements allow you to connect dissimilar meshes over a common geometric boundary, instead of using transition meshes or constraint conditions. Primary applications where you might specify the interface elements manually include: facilitating global-local analysis, where a patch of elements may be removed from the global model and replaced by a denser patch for a local detail, without having to transition to the surrounding area; and connecting meshes built by different engineering organizations, such as a wing to the fuselage of an airplane.

Primary applications where the interface elements could be generated automatically are related to automeshers, which may be required to transition between large and small elements between mesh regions; and h-refinement, where subdivided elements may be adjacent to undivided elements without a transition area.

Dissimilar meshes can occur with global-local analysis, where part of the structure is modeled as the area of primary interest in which detailed stress distributions are required, and part of the structure is modeled as the area of secondary interest through which load paths are passed into the area of primary interest.

Generally the area of primary interest has a finer mesh than the area of secondary interest and, therefore, a transition area is required. Severe transitions generally produce elements that are heavily distorted, which can result in poor stresses and poor load transfer into the area of primary interest. An example of using interface elements to avoid such transitions is shown in Figure 6-19. Similarly, a patch of elements may be removed from the global model and replaced by a denser patch for local detail.

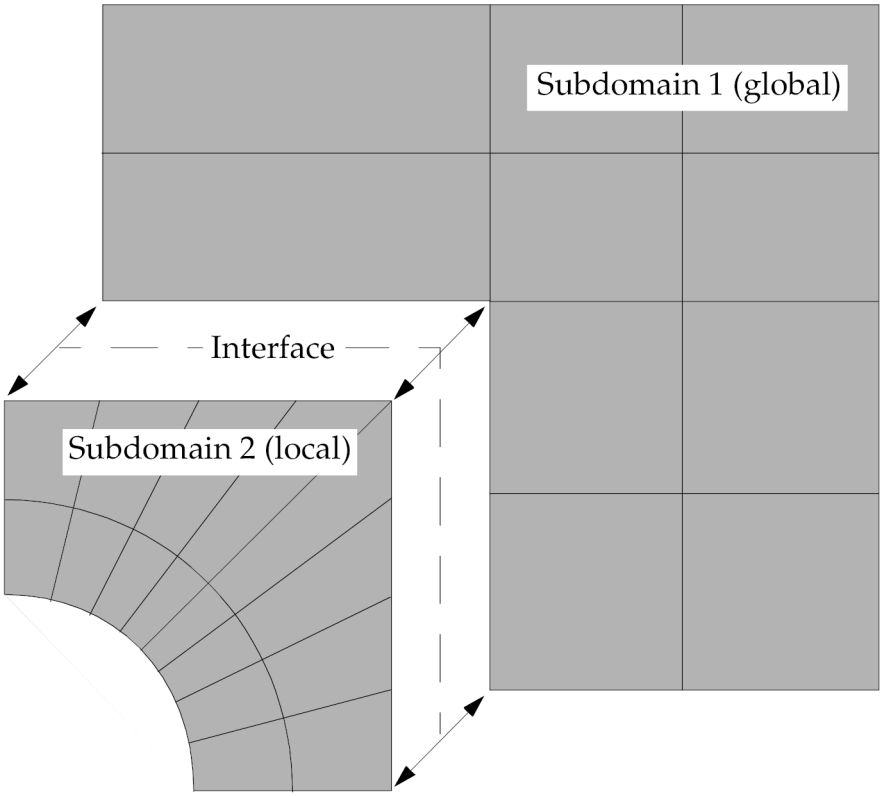


Figure 6-19. Example of Interface Elements (Exploded View)

In large system assemblies, different analysts or even different organizations may have created different components of the model, such as the wing and the fuselage of an airplane. Unless they have carefully coordinated their efforts, the finite element meshes of the different components may not match at the interfaces.

Dissimilar meshes generated by the analysis program can also arise with automeshers, which may be required to transition between large elements and small elements in a small area. Many automeshers generate tetrahedral meshes for solids, and distorted tetrahedra may be more susceptible to poor results. Interface elements are particularly useful when a transition is needed between large and small elements within a small area.

When mesh refinement is performed, subdivided elements may be adjacent to undivided elements with no room for a transition area. Without some kind of interface element, the subdivision would need to be carried out to the model boundary or otherwise transitioned out.

It is important to note that the interface elements provide a tool for connecting dissimilar meshes, but they do not increase the accuracy of the mesh. As with any interface formulation, the hybrid variational technology, which imposes continuity conditions in a weak form, can not increase the accuracy of the adjacent subdomains. For instance, if a single element edge on one boundary is connected to many element edges on the other boundary, the analysis is going to be limited to the accuracy of the boundary containing the single element edge. This restriction should be considered when deciding how close to the areas of primary interest to put the interface elements.

Guidelines

- The interface elements use the geometry of the boundary with the least number of degrees-of-freedom, which consist of cubic polynomials. If the other boundaries have widely varying geometry, poor answers may result. Warnings may be issued, but no geometrical adjustment is performed.

- Sharp corners within the interface element may degrade accuracy. The preferred alternative is to specify multiple interface elements. An example of multiple interface elements is solved as the second example.

- Connecting few elements to many will not improve the accuracy along the interface. The limiting factor on the accuracy will be those few elements. For example, if one boundary has one element and the other boundary has four, the accuracy will be limited to that one element.

- Interface elements only connect common displacement fields of different element types. No kinematic constraints are enforced. Shell p-elements have five fields, and solid p-elements have three. Therefore the rotational fields of the shell p-elements will not be connected to the solid p-elements.

- The sparse solver for linear statics and the Lanczos eigensolver for normal modes should be used. The sparse solver is the default solver for linear statics.

- The value of epsilon, which is the residual from the linear solution, and the shapes of modes of primary interest, which can best be evaluated graphically, should be checked to detect unstable results. Plots of displacements and stresses may also indicate unstable results. This would be visible as discontinuities in the displacements or stresses across the interface, which would imply a poor solution in that area.

- Since the boundaries are physically distinct, certain functions, such as shell normals and stress discontinuities in the error estimator, will not be applied across the interface.

- Contour plots may show differences across the interface because of the different view meshes. However, this is an indication of the results processing, not the original solution. A denser view mesh would reduce the differences.

Surface Interface Elements

You are often need to connect dissimilar meshes when you're refined specific regions of a mesh. One method of connecting these dissimilar meshes is to use interface elements. An example in which patches of elements have been removed from the global model and replaced by denser patches for local details is shown in Figure 6-20, where the boundaries of the patches are bold.

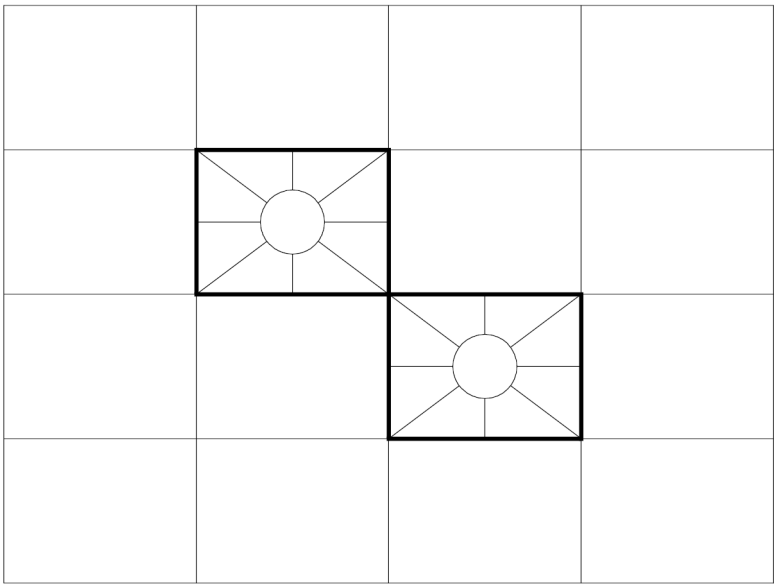


Figure 6-20. Example of Dissimilar Mesh from Global/Local Analysis

Implementation

Surface interface elements have been implemented for p-elements:

- The surface interface elements connect p-element faces. These can be either faces of solid p-elements or shell p-elements, and the faces may be either quadrilateral or triangular.
- The interface elements are geometry-based. They use the same geometry as the p-element boundaries.
- The interface elements connect only corresponding displacement fields. They do not have kinematic constraints, such as connecting shell rotations to solid translations.
- There are three methods of defining the subdomain boundaries of solid or shell p-element faces (GMBNDS). For the surface interface, each boundary may be defined using the GMSURF with which the finite element faces are associated; the FEFACEs defining the finite element faces; or in the most basic form, the GRIDs over the finite element faces.
- Once the boundaries have been defined, they must be associated with the interface elements. This is accomplished by referencing the boundaries in the interface element definition (GMINTS).
- Since the interface elements consist only of the differences in displacement components weighted by the Lagrange multipliers, there are no conventional element or material properties. The property Bulk Data entry (PINTS) specifies a tolerance for the interface elements, which defines the allowable distance between the subdomain boundaries; and a scaling factor, which may improve the conditioning of the Lagrange multipliers.

Output

The interface elements have no output of their own. However, they do cause changes in the customary output:

- Internal node, edge, and face degrees-of-freedom are generated for the interface elements. Because of their formulation, use of the parameter AUTOSPC, which is the default in the allowable solution sequences, detects them as singular. This can make the Grid Point Singularity Table (GPST) larger than expected. These degrees-of-freedom will also appear in the USET table.

- The interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other model errors, these messages may be ignored and PARAM, BAILOUT, –1 may be used to continue.

Guidelines

- The interface elements use the geometry of the boundary with the least number of degrees-of-freedom, which consist of cubic polynomials. If the other boundaries have widely varying geometry, poor answers may result. Warnings may be issued, but no geometrical adjustment is performed.

- Sharp corners within the interface element may degrade accuracy. The preferred alternative is to specify multiple interface elements. An example of multiple interface elements is solved as the second example.

- Connecting few elements to many will not improve the accuracy along the interface. The limiting factor on the accuracy will be those few elements. For example, if one boundary has one element and the other boundary has four, the accuracy will be limited to that one element.

- Interface elements only connect common displacement fields of different element types. No kinematic constraints are enforced. Shell p-elements have five fields, and solid p-elements have three. Therefore the rotational fields of the shell p-elements will not be connected to the solid p-elements.

- The sparse solver for linear statics and the Lanczos eigensolver for normal modes should be used. The sparse solver is the default solver for linear statics.

- The value of epsilon, which is the residual from the linear solution, and the shapes of modes of primary interest, which can best be evaluated graphically, should be checked to detect unstable results. Plots of displacements and stresses may also indicate unstable results. This would be visible as discontinuities in the displacements or stresses across the interface, which would imply a poor solution in that area.

- Since the boundaries are physically distinct, certain functions, such as shell normals and stress discontinuities in the error estimator, will not be applied across the interface.

- Contour plots may show differences across the interface because of the different view meshes. However, this is an indication of the results processing, not the original solution. A denser view mesh would reduce the differences.

Limitations

- Constraining the boundaries may lead to unstable results in certain cases. The most common constraint on a boundary is at an endpoint, such as a symmetry condition. Cases with constraints include multiple interfaces at the same point, such as a sharp corner; and endpoints of different boundaries connected together, such as an interior interface. Such unstable results are

indicated by high epsilons or non-physical modes. They are also indicated by irregularities in the displacements or stresses.

- Superelements aren’t supported.

Chapter 7: R-Type Elements

- *Introduction to R-Type Elements*
- *Rigid element processing options*
- *The RROD Element*
- *The RBAR Element*
- *The RBE2 Element*
- *The RBE3 Element*

7.1 Introduction to R-Type Elements

An R-type element is an element that imposes fixed constraints between components of motion at the grid points or scalar points to which they are connected. Thus, an R-type element is mathematically equivalent to one or more multipoint constraint equations. Each constraint equation expresses one dependent degree of freedom as a linear function of the independent degrees of freedom.

Although sometimes the R-type elements are referred to collectively as rigid elements, the name “rigid” is misleading. The R-type elements that are rigid consist of the RROD, RBAR, RBE1, RBE2, and RTRPLT. The RBE3 and RSPLINE are interpolation elements and aren’t rigid. The RBAR, RBE2, and RBE3 elements are the most commonly used R-type elements in the NX Nastran element library. In addition these R-type elements, you can use the RSSCON element to transition between plate and solid elements.

Note

Using rigid elements will cause incorrect results in buckling and differential stiffness analyses because the large displacement effects are not calculated. Exceptions (rigid elements for which there is no error) are zero length elements (to simulate a hinge) and rigid elements constrained so that they don’t rotate.

Overview of Available Element Types

Table 7-1 lists the rigid body elements available in NX Nastran. These elements require only the specification of the degrees-of-freedom that are involved in the constraint equations. All coefficients in these equations of constraint are calculated internally in NX Nastran.

Table 7-1. Rigid Element and MPC Entries

Name	Description	*m* = Dependent Degrees-of-freedom
RROD	A open-ended rod which is rigid in extension	$m = 1$
RBAR	Rigid bar with six degrees-of-freedom at each end.	$1 \le m \le 6$
RTRPLT	Rigid triangular plate with six degrees-of-freedom at each vertex.	$1 \le m \le 12$
RBE1	A rigid body connected to an arbitrary number of grid points. The independent and dependent degrees-of-freedom can be arbitrarily selected by the user.	$m \ge 1$
RBE2	A rigid body connected to an arbitrary number of grid points. The independent degrees-of-freedom are the six components of motion at a single grid point. The dependent degrees-of-freedom at the other grid points all have the same user-selected component numbers.	$m \ge 1$
RBE3	Defines a constraint relation in which the motion at a “reference” grid point is the least square weighted average of the motions at other grid points. The element is useful for “beaming” loads and masses from a “reference” grid point to a set of grid points.	$1 \le m \le 6$
RSPLINE	Defines a constraint relation whose coefficients are derived from the deflections and slopes of a flexible tubular beam connected to the referenced grid points. This element is useful in changing mesh size in finite element models.	$m \ge 1$
RSSCON	Define a multipoint constraint relation which models a clamped connection between shell and solids.	$m \ge 5$
MPC	Rigid constraint that involves user-selected degrees-of-freedom at both grid points and at scalar points. The coefficients in the equation of constraint are computed and input by the user.	$m = 1$

You can use any combination of the above elements in an NX Nastran analysis in any of the structural solution sequences. However, you should use these elements with care in geometric nonlinear

analysis (see the *NX Nastran Handbook for Nonlinear Analysis*). The rigid elements are ignored in the heat transfer solution sequences.

Typical Applications for R-type Elements

Typical applications that use R-type entries are shown in Table 7-2.

Table 7-2. Typical Application for R-type Elements

Application	R-Type Entries
Triangular Bell Crank	RTRPLT
Rigid Engine Blocks	RBE1
Tripod with Hinged Rigid Legs	RROD
Rigid Bulkhead	RBE2
Evaluation of Resultant Loads	RBE2
Connection of a Bar Element to a Shell	RBE2 or RBE3
Hinge Between Two Plates	RBAR
Recording Motion in a Nonglobal Direction	RBAR
Relative Motion	MPC
Incompressible Fluid in an Elastic Container	MPC
“Beaming” Loads and Masses	RBE3
Change in Mesh Size	RSPLINE
Transitions Between Plate and Solid Elements	RSSCON

Understanding R-type Elements and Degrees-of-Freedom

In NX Nastran, each MPC entry generates a single equation of the form

$$A_1 u_1 + A_2 u_2 + \ldots A_n u_n = 0$$

Equation 7-1.

where u_1 , u_2 , ..., u_n are user-designated degrees-of-freedom (grid point plus component) and A_1 , A_2 , ..., A_n are coefficients which are user-supplied. The first named degree-of-freedom is placed in the u_m set (degrees-of-freedom eliminated by multipoint constraints).

In NX Nastran, you can use either MPCs or R-type elements to model rigid bodies and rigid constraints. The MPC entry provides considerable generality but lacks user convenience. Specifically, you must supply the coefficients in the equations of constraint defined through the MPC entry. With R-type elements, NX Nastran automatically generates a constraint equation (an internal MPC equation) of the form Eq. 7-1 for each dependent degree of freedom.

When using an R-type element, you must define which degrees of freedom are dependent and which are independent. The simplest way to describe this is to say that the motion of a dependent degree of freedom is expressed as a linear combination of one or more of the independent degrees of freedom.

- All dependent degrees of freedom are placed in the u_m set. The complete u_m set consists of the first named terms on the MPC entries plus the designated degrees-of-freedom on the rigid element entries. You have complete control over the membership of this set.

- All independent degrees of freedom are temporarily placed in the u_n set, which is the set that is not made dependent by MPCs or R-type elements. This designation may be temporary; members of the u_n set may be removed by additional constraints in your model.

The following section lists the procedural requirements and rules that you must follow when using MPCs and R-type elements in an analysis:

1. A member of the u_m set can't also be a member of any other user defined set. User-defined sets include:

   u_s – degrees-of-freedom eliminated by SPCi entries, AUTOSPC, and PS field on GRID entries

   u_o – degrees-of-freedom specified on OMITi Bulk Data entries

   u_r – degrees-of-freedom specified on SUPORT and SUPORT1 Bulk Data entries

   u_a – members of the analysis set specified on ASETi Bulk Data entries or exterior degrees-of-freedom in superelement analysis

2. A degree-of-freedom can't be designated as a member of the u_m set more than once unless PARAM,AUTOMPC,YES is defined. See AUTOMPC in the QRG for details.

   A fatal error results if, for example, PARAM,AUTOMPC,NO is defined, and the same degree-of-freedom is designated as dependent by two rigid elements or it the first-named degree-of-freedom on any MPC entry is also a designated member of u_m on a rigid element entry.

3. The user-selected independent degrees-of-freedom u_n , for the RBAR, RTRPLT, RBE1, RBE2, RBE3 and RSPLINE elements must be sufficient to define any general rigid body motion of the element.

   These degrees-of-freedom are independent only for the particular element, and they may be declared dependent by other rigid element, MPCs, SPCs, OMITs, or SUPORTs. As far as a particular rigid element is concerned, it is always acceptable to select all six independent degrees-of-freedom at one grid point. This may not, however, be a good choice when the total problem requirements are considered.

   For these elements(RBAR, RTRPLT, RBE1, RBE2, RBE3, RSPLINE), you list the degrees-of-freedom in u_m and u_n . The remaining degrees-of-freedom at the grid points to which the rigid element is jointed aren't involved with the rigid element. This lack of connection represents either a sliding or rotating joint release, or both. The rigid rod element (RROD) is an exception because once a component of translation is placed in u_m , all of the five remaining components of translation will automatically be placed in u_n . The rotational degrees-of-freedom are not involved in the RROD element.

4. You must avoid over-constraining the structure when two or more rigid elements are used. A structure is over-constrained when the degrees-of-freedom, which remain after the members of u_m have been selected, are insufficient to represent a general rigid body motion of the structure as a whole.

   Consider, for example, a number of RBAR elements connected together to form a rigid ring. Let the grid points be numbered from 1 to N and assume that the u_m degrees-of-freedom for each rigid element are placed at the higher numbered grid point so that the only degrees-of-freedom which remain independent as each element is added to the ring are those at grid point 1. The addition of the last rigid element between grid points N and 1 will remove even those independent degrees-of-freedom and thereby over-constrain the structure.

5. For the RSSCON element, the shell degrees-of-freedom are placed in u_m. The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower edge of the solid. The shell's two rotational degrees-of-freedom are also connected to the translational degrees-of-freedom of the upper and lower edge of the solid. The RSSCON only impresses a rigid constraint on the shell's two rotational degrees-of-freedom.

6. Nonlinear forces in dynamic analysis can't be applied to members of the u_m set.

7.2 Rigid Element Processing Options

Rigid elements are represented mathematically as a system of linear constraint equations that can be represented in matrix form as:

$[RG]\{x\} = \{0\}$

A constraint equation exists for each dependent degree of freedom (DOF).

When NX Nastran processes RBAR, RBE1, RBE2, RBE3, RROD and RTRPLT rigid elements, the software uses either the linear elimination method or the Lagrange multiplier method. You can select the rigid element processing method that you want by using the RIGID case control command.

- Use RIGID = LINEAR to select the linear elimination method. This is the default method.
- Use RIGID = LAGRAN to select the Lagrange multiplier method.

When determining which rigid element processing method to select, note that:

- Only the Lagrange multiplier method allows for thermal expansion in rigid elements.
- The Lagrange multiplier method may yield a performance improvement for dynamic solutions even though it adds DOF to the problem. This occurs because the Lagrange multiplier method more efficiently processes the very densely populated mass matrices that rigid elements can produce.

For other cases, the linear elimination method is preferred because the artificial stiffness that is added to the model when using the Lagrange multiplier method can produce either ill-conditioned stiffness matrices or overly stiff models.

Linear elimination method

The linear elimination method partitions the system of linear constraint equations into dependent (M-set) and independent (N-set) DOF groupings.

$$\begin{bmatrix} RG_M & RG_N \end{bmatrix} \begin{Bmatrix} x_M \\ x_N \end{Bmatrix} = \{0\}$$

The dependent DOF are then solved for in terms of the independent DOF as follows:

$$\{x_M\} = -[RG_M]^{-1}[RG_N]\{x_N\} = [GM]\{x_N\}$$

The equation of motion before elimination of dependent DOF is:

$$\begin{bmatrix} M_{MM} & 0 \\ 0 & M_{NN} \end{bmatrix} \begin{Bmatrix} \ddot{x}_M \\ \ddot{x}_N \end{Bmatrix} + \begin{bmatrix} K_{MM} & 0 \\ 0 & K_{NN} \end{bmatrix} \begin{Bmatrix} x_M \\ x_N \end{Bmatrix} = \begin{Bmatrix} p_M \\ p_N \end{Bmatrix}$$

The linear elimination method reduces the equation of motion to just the independent DOF as follows:

$$\left([GM]^T [M_{MM}][GM] + [M_{NN}] \right) \{\ddot{x}_N\} + \left([GM]^T [K_{MM}][GM] + [M_{NN}] \right) \{x_N\} = \{p_N\} + [GM]^T \{p_M\}$$

Note that the mass, stiffness and loads on the dependent DOF (M-set) get distributed onto the independent DOF (N-set). This can result in very densely populated matrices, which is a disadvantage of the elimination method. For example, a single lumped mass distributed onto a large number of grids using an RBE3 element will generate a fully populated mass matrix on the associated DOF. For these cases, the Lagrange multiplier method may improve performance.

Lagrange multiplier method

An alternate approach is the Lagrange multiplier method. Unlike the linear elimination method, NX Nastran does not differentiate between dependent and independent DOF when using the Lagrange multiplier method. Instead, NX Nastran defines a Lagrange multiplier for each constraint equation and then treats the Lagrange multipliers as additional DOF. The resulting equation of motion is:

$$\begin{bmatrix} M_{GG} & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_G \\ \ddot{\lambda}_M \end{Bmatrix} + \begin{bmatrix} K_{GG} & RG^T \\ RG & 0 \end{bmatrix} \begin{Bmatrix} x_G \\ \lambda_M \end{Bmatrix} = \begin{Bmatrix} p_G \\ 0 \end{Bmatrix}$$

Although the Lagrange multiplier method adds DOF to the problem rather than removing them, the mass matrix remains very sparse and the stiffness matrix remains relatively sparse. The problem is that the augmented stiffness matrix is often singular. Because the augmented stiffness matrix is often singular, NX Nastran optionally adds artificial stiffness terms that connect the DOF in each constraint. The artificial stiffness can be thought of as beam elements that span the same DOF as the constraint equations.

With the artificial stiffness added, singularity is eliminated and the equation of motion becomes:

$$\begin{bmatrix} M_{GG} & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_G \\ \ddot{\lambda}_M \end{Bmatrix} + \begin{bmatrix} K_{GG} + \tilde{K}_{GG} & RG^T \\ RG & 0 \end{bmatrix} \begin{Bmatrix} x_G \\ \lambda_M \end{Bmatrix} = \begin{Bmatrix} p_G \\ 0 \end{Bmatrix}$$

If the constraints are for a rigid element like an RBE2, the artificial stiffness terms do not alter the problem because the relationship between the constrained DOF represents rigidity already. However, if the constraints are for an RBE3 element, the artificial stiffness terms can produce:

- A numerically ill-conditioned stiffness matrix if the artificial stiffness is too little.
- An overly stiff stiffness matrix if the artificial stiffness is too large.

To control artificial stiffness when RBE3 elements are present, two scale factors, c_λ and c_K, are available. The equation of motion with the scale factors included is:

$$\begin{bmatrix} M_{GG} & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_G \\ \ddot{\lambda}_M \end{Bmatrix} + \begin{bmatrix} K_{GG} + c_K \tilde{K}_{GG} & c_\lambda RG^T \\ c_\lambda RG & 0 \end{bmatrix} \begin{Bmatrix} x_G \\ \lambda_M \end{Bmatrix} = \begin{Bmatrix} p_G \\ 0 \end{Bmatrix}$$

The c_λ scale factor multiplies the constraint equation matrix, [*RG*], and can be set to help avoid a numerically ill-conditioned stiffness matrix. The c_K scale factor multiplies the artificial stiffness terms and can be set to help avoid over stiffening the stiffness matrix.

The values for the scale factors are problem dependent.

- Set the c_λ scale factor so that the terms in the constraint equations are the same order of magnitude as the terms in the stiffness matrix. The c_λ scale factor is set directly by specifying the LMFACT parameter. The LMFACT parameter is applicable to all rigid elements. The default value is 1.0×10^6, which is an appropriate value for most models.

- Set the c_K scale factor indirectly using the new LMSTAT and LMDYN parameters. The LMSTAT and LMDYN parameters are applicable to RBE3 elements only.

  For a statics solution, specify the LMSTAT parameter, where c_K and the LMSTAT setting are related by:

  $$c_K = \frac{1.0}{10^{LMSTAT}}$$

  The default value for LMSTAT is 6, which results in $c_K = 1.0 \times 10^{-6}$. To completely remove artificial stiffness from the problem, set LMSTAT to -1. When LMSTAT is set to -1, c_K is set to zero.

  For a dynamics solution, specify the LMDYN parameter, where c_K and the LMDYM setting are related by:

  $$c_K = \frac{1.0}{10^{LMDYN}}$$

  The default value for LMDYN is -1. When LMSTAT is set to -1, c_K is set to zero and artificial stiffness is completely removed from the problem.

For modal equations, there is no requirement that the stiffness matrix be non-singular. Thus, c_K can always be set to zero. However for static solutions, some non-zero value is normally required because a DOF in the M-set which has zero stiffness will be restrained by the AUTOSPC operation.

Thermal expansion of rigid elements using Lagrange multiplier method

Thermal expansion on the RBAR, RBE1, RBE2, RBE3, RROD and RTRPLT elements is calculated when:

- ALPHA is defined on the rigid element definition.

- The case control RIGID = LAGRAN is defined.

- The grid point initial and load temperatures are defined using the TEMPERATURE case control command on the grids defining the rigid element connectivity.

When thermal strain is included, the right hand side of the equation of motion includes the change in displacement, u_T, that results from the thermal loading.

$$\begin{bmatrix} M_{GG} & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_G \\ \ddot{\lambda}_M \end{Bmatrix} + \begin{bmatrix} K_{GG} + c_K \tilde{K}_{GG} & c_\lambda R G^T \\ c_\lambda R G & 0 \end{bmatrix} \begin{Bmatrix} x_G \\ \lambda_M \end{Bmatrix} = \begin{Bmatrix} p_G \\ u_T \end{Bmatrix}$$

The change in temperature for the thermal strain is the difference between the grid point temperatures selected with the TEMPERATURE(LOAD) and TEMPERATURE(INIT) case control commands. The load and initial temperature values used on RBAR, RROD, and RTRPLT elements is an average calculated from the grid point values. On RBE1, RBE2 and RBE3 elements, an average load and initial temperature value 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.

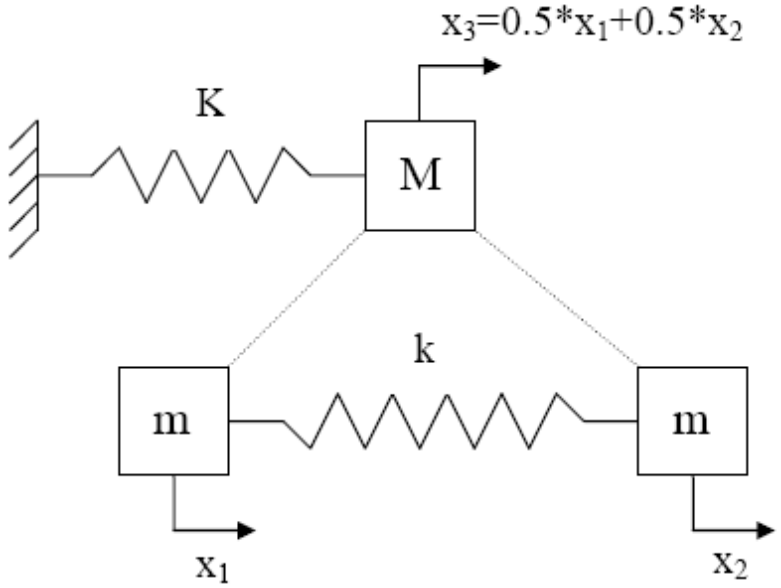
The rigid element thermal strains are calculated from:

$\varepsilon_{thermal} = \alpha(AVGTEMP(LOAD) - AVGTEMP(INIT))$

If load or initial temperatures are undefined, they are assumed to be zero.

Rigid element processing example

In this three DOF example, x_1 and x_2 are independent, and x_3 is dependent.



The equation of motion before applying constraints is:

$$\begin{bmatrix} m & 0 & 0 \\ 0 & m & 0 \\ 0 & 0 & M \end{bmatrix} \begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \\ \ddot{x}_3 \end{Bmatrix} + \begin{bmatrix} k & -k & 0 \\ -k & k & 0 \\ 0 & 0 & K \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x3 \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \\ p_3 \end{Bmatrix}$$

The constraint equation is:

$$\begin{bmatrix} -0.5 & -0.5 & 1 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \end{Bmatrix} = 0$$

Solving the constraint equation for x_3 gives:

$$x_3 = \begin{bmatrix} 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix}$$

Using the elimination method, x_3 is removed from the equation of motion as follows:

$$\begin{bmatrix} m+0.25M & 0.25M \\ 0.25M & m+0.25M \end{bmatrix} \begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \end{Bmatrix} + \begin{bmatrix} k+0.25K & -k+0.25K \\ -k+0.25K & k+0.25K \end{bmatrix} \begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \end{Bmatrix} = \begin{Bmatrix} p_1+0.5p_3 \\ p_2+0.5p_3 \end{Bmatrix}$$

This equation can be easily solved, though it is clear that the mass and stiffness for x_3 has been redistributed onto x_1 and x_2.

Now consider the Lagrange multiplier method. The equation of motion now includes an extra term as follows:

$$\begin{bmatrix} m & 0 & 0 & 0 \\ 0 & m & 0 & 0 \\ 0 & 0 & M & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \\ \ddot{x}_3 \\ \ddot{\lambda} \end{Bmatrix} + \begin{bmatrix} k & -k & 0 & -0.5 \\ -k & k & 0 & -0.5 \\ 0 & 0 & K & 1 \\ -0.5 & -0.5 & 1 & 0 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \\ \lambda \end{Bmatrix} = \begin{Bmatrix} p_1 \\ p_2 \\ p_3 \\ 0 \end{Bmatrix}$$

Although the Lagrange multiplier method has added DOF to the equation of motion, the sparsity of the matrices is retained.

7.3 The RROD Element

For the RROD element, you specify a single component of translation at one of its two end points as a dependent degree of freedom. The equivalent component at the other end is the independent degree of freedom. Consider the example shown in Figure 7-1.

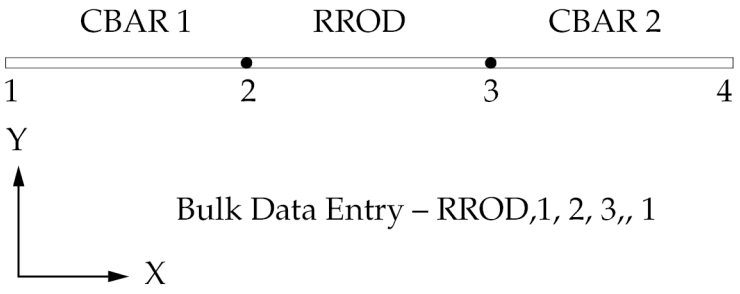


Figure 7-1. An RROD Connection

For this example, a rigid connection is made between the X component of grid point 3 to the X component of grid point 2. When you specify the RROD entry as shown, you are placing component 1 of grid point 3 (in the global system) into the *m*-set. The remaining 5 components at grid point 3 and all the components at grid point 2 are placed in the *n*-set and hence, are independent. CBAR 1 is not connected to components Y, Z, R_x , R_y , and R_z of grid point 3 in any manner; therefore, the ends of the bars are free to move in any of these directions. However, the ends of the two CBAR elements are rigidly attached in the X-direction. Having the connection only in the X-direction is consistent with Table 7-1, which shows that one degree of freedom is placed in the *m*-set for each RROD element.

See Also

- "RROD" in the *NX Nastran Quick Reference Guide*

7.4 The RBAR Element

The RBAR element rigidly connects from one to six dependent degrees of freedom (the *m* -set) to exactly six independent degrees of freedom. The six independent degrees of freedom must be capable of describing the rigid body properties of the element.

The format for the RBAR entry is as follows:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB		

Field	Contents
EID	Element identification number.
GA, GB	Grid point identification number of connection points.
CNA, CNB	Component numbers of independent degrees of freedom in the global coordinate system for the element at grid points GA and GB.
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 Also

- "RBAR" in the *NX Nastran Quick Reference Guide*

The most common approach when using the RBAR element is to define one end of the RBAR with all six independent degrees of freedom with dependent degrees of freedom at the other end. (However, placing all of the independent degrees of freedom at one end is not a requirement.) To determine if the choice you make for the independent degrees of freedom meets the rigid body requirements, ensure that the element passes the following simple test:

If you constrain all of the degrees of freedom defined as independent on the RBAR element, is the element prevented from any possible rigid body motion?

As an example, consider the RBAR configurations shown in Figure 7-2. For configurations (a) and (c), if the six independent degrees of freedom are held fixed, the element cannot move as a rigid body in any direction. However, for (b), if all six of the independent degrees of freedom are held fixed, the element can still rotate about the Y-axis. Configuration (b) doesn't pass the rigid body test and does not work as an RBAR element.

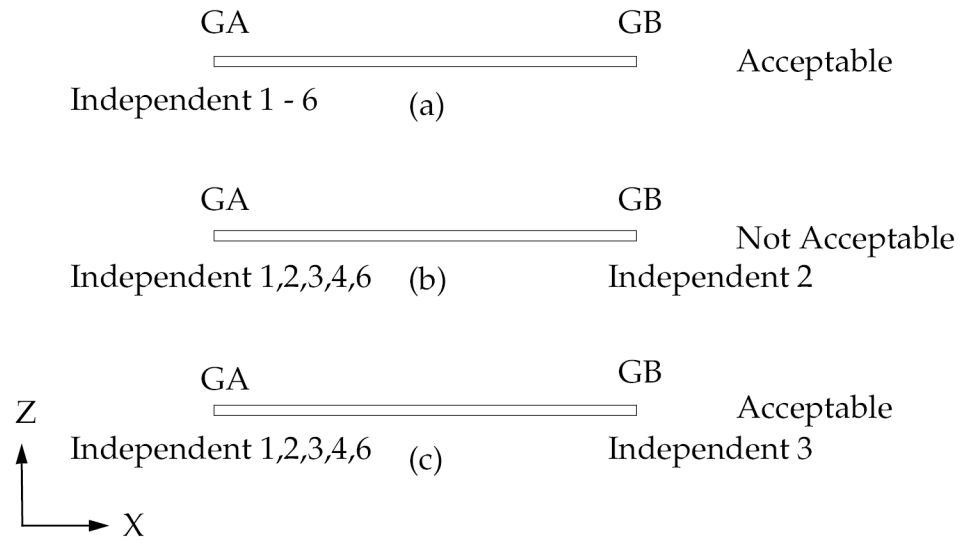


Figure 7-2. Defining Independent DOFs on the RBAR

NX Nastran generates internal MPC equations for the R-type elements. As an example of this, consider the model of a thick plate with bars attached as shown in Figure 7-3. The interface between the bars and the plate is modeled two ways, first using MPC entries and second using RBAR elements.

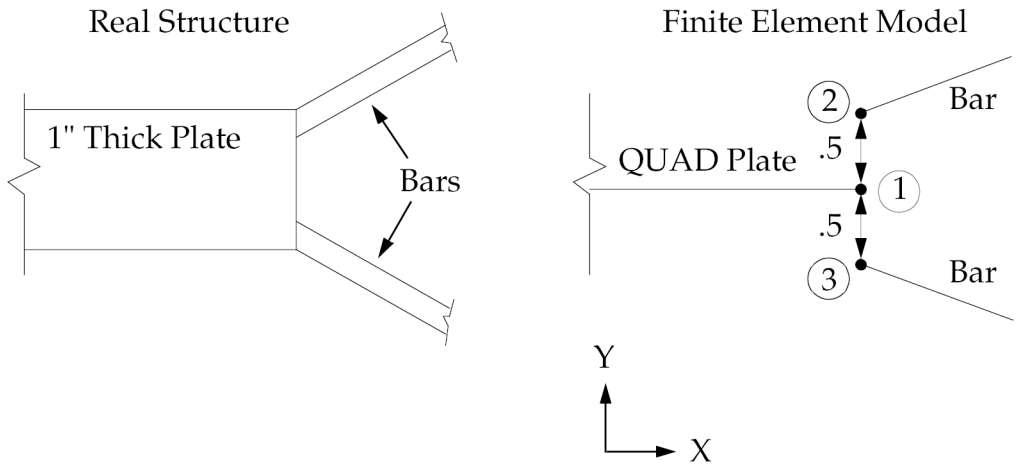


Figure 7-3. Model of a Thick Plate with Bars Attached

Option 1 – Model the Transition with MPC Equations

Plate theory states that plane sections remain planar. If this is the case, then grid points 2 and 3 are slaves to grid point 1. Therefore, you need to write the equations for the in-plane motion of grid points 2 and 3 as a function of grid point 1. Each RBAR element creates up to six constraint equations.

Looking only at the motion in the x-y plane,

$$u_{1_2} = u_{1_1} - .5 \cdot u_{6_1} \qquad (u_{1_2} = \text{displacement in the 1-direction at grid point 2})$$

$$u_{1_3} = u_{1_1} + .5 \cdot u_{6_1}$$

$$u_{6_3} = u_{6_1}$$

$$u_{6_2} = u_{6_1}$$

$$u_{2_2} = u_{2_1}$$

$$u_{2_3} = u_{2_1}$$

The MPC entries for this model are as follows:

1	2	3	4	5	6	7	8	9	10
MPC	1	2	1	1.	1	1	-1.		
		1	6	.5					
MPC	1	3	1	1.	1	1	-1.		
		1	6	-.5					
MPC	1	3	6	1.	1	6	-1.		
MPC	1	2	6	1.	1	6	-1.		
MPC	1	2	2	1.	1	2	-1.		
MPC	1	3	2	1.	1	2	-1.		

Note

MPC = 1 must appear in the Case Control Section to use these entries.

Option 2 - Model the Transition with RBAR Entries

The R-type elements are easier to use than the equivalent MPC entries.

1	2	3	4	5	6	7	8	9	10
$RBAR	EID	GA	GB	CNA	CNB	CMA	CMB		
RBAR	99	1	2	123456					
RBAR	100	1	3	123456					

RBAR 99 generates MPC equations for the motion of grid point 2 as a function of grid point 1. Likewise, RBAR 100 generates MPC equations for the motion of grid point 3 as a function of grid point 1. These RBARs generate the MPC equations for all six DOFs at grid points 2 and 3. If it is desired to have the equations generated only for the in-plane motion, the field labeled as CMB in the RBAR entries has the values 126 entered.

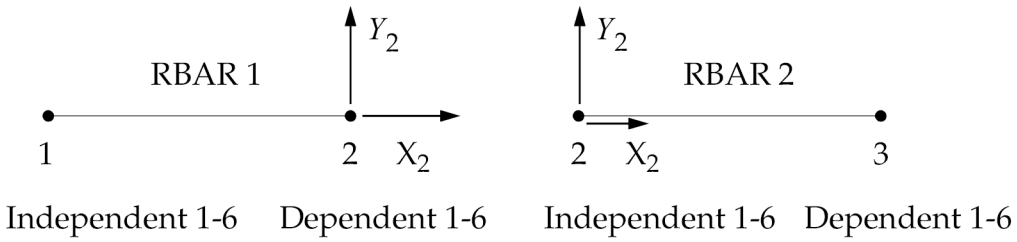
A particular area of confusion for the new user is when you need to connect R-type elements together. The important thing to remember is that you can place a degree of freedom into the *m* -set only once. Consider the two RBAR elements shown in Figure 7-4 that are acting as a single rigid member.

If you choose grid point 1 for RBAR 1 to be independent (1-6), then grid point 2 for RBAR 1 must be dependent (1-6). Since grid point 2 is dependent for RBAR1, it must be made independent for RBAR

2. If you made grid point 2 dependent for RBAR 2 as well as RBAR 1, a fatal error would result. Since grid point 2 is independent (1-6) for RBAR 2, grid point 3 will be dependent (1-6).

If you chose grid point 1 of RBAR 1 to be dependent, then grid point 2 for RBAR 1 would be independent. Grid point 2 of RBAR 2 would be dependent, and grid point 3 of RBAR 2 would be independent.

The RBAR element is often used to rigidly connect two grid points in your model.



Note: RBARs 1 and 2 are connected. They are shown separated for clarity.

Figure 7-4. Connecting Two RBAR Elements

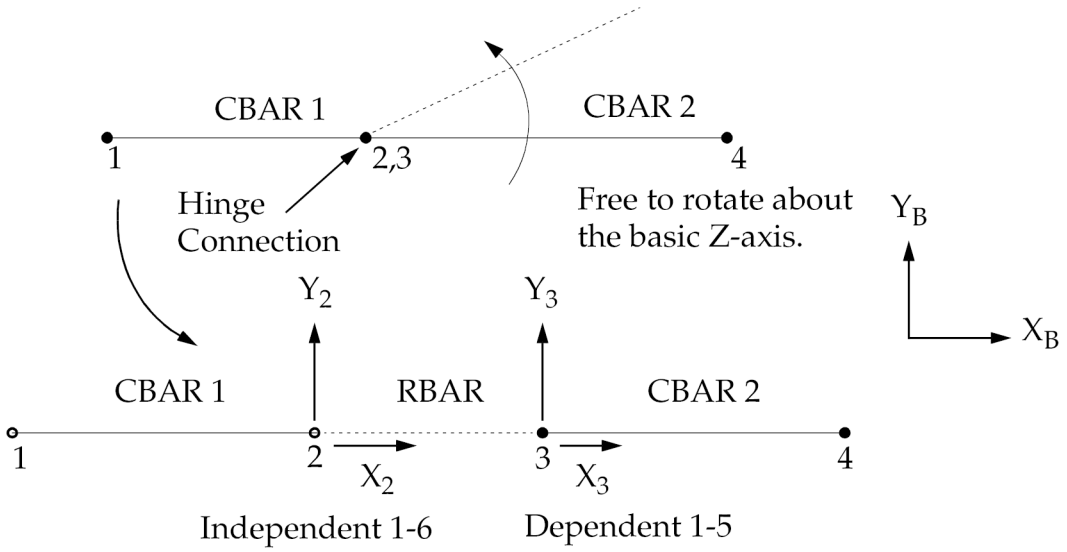
Both options for connecting the RBAR elements are shown in Listing 7-1. For clarity, CBAR3, which is connected to grid point 3, is not shown in Lisitng 7-4.

```
$ FILENAME RBAR1.DAT
ID LINEAR,RBAR1
SOL 101
TIME 2
CEND
TITLE = CONNECTING 2 RBARS
DISPLACEMENT = ALL
SUBCASE 1
  LOAD = 1
SUBCASE 2
  LOAD = 2
BEGIN BULK
$
GRID    1               0.      0.      0.
GRID    2               10.     0.      0.
GRID    3               20.     0.      0.
GRID    4               30.     0.      0.              123456
$
$  OPTION 1
$
RBAR    1       1       2       123456
RBAR    2       2       3       123456
$
$  OPTION 2
$
$RBAR    1       1       2               123456
$RBAR    2       2       3               123456
$
CBAR    3       1       3       4       0.      1.      0.
PBAR    1       1       1.      1.      1.      1.
MAT1    1       20.4            .3
$
```

```
$ POINT LOAD
$
FORCE   1       1               1.      1.      0.      0.
FORCE   2       1               1.      0.      1.      0.
$
ENDDATA
```

Listing 7-1. Connecting RBAR Elements

As a final example of the RBAR element, consider the hinge model shown in Figure 7-5.



Note: Grid points 2 and 3 are coincident. They are shown separated for clarity.

Figure 7-5. Modeling a Hinge Using an RBAR

The simplest way to model the hinge connection with an RBAR is to use coincident grid points at the center of rotation (grid points 2 and 3 in this example) and define an RBAR between the two grid points. This RBAR has zero length, which is acceptable for the RBAR. Make all six of the components associated with one grid point independent. Make only a select number of components of the other grid point dependent, leaving independent the components representing the hinge. A partial listing of the input file for this model is shown in Listing 7-2. Note that the components 1 through 6 of grid point 2 are independent and components 1 through 5 of grid point 3 are dependent. Component 6 of the grid point 3 is left independent, permitting CBAR 2 to rotate about the Z axis with respect to CBAR 1.

```
$ FILENAME - RBAR2.DAT
ID LINEAR,RBAR2
SOL 101
TIME 2
CEND
TITLE = CONNECTING 2 BARS WITH AN RBAR HINGE
DISPLACEMENT = ALL
LOAD = 1
FORCE = ALL
BEGIN BULK
$
GRID    1               0.      0.      0.              123456
GRID    2               10.     0.      0.
GRID    3               10.     0.      0.
GRID    4               20.     0.      0.              123456
$
RBAR    99      2       3       123456                  12345
$
CBAR    1       1       1       2       0.      1.      0.
CBAR    2       1       3       4       0.      1.      0.
PBAR    1       1       .1      .01     .01     .02
MAT1    1       20.+4           .3
$
$ POINT LOAD
$
FORCE   1       2               100.     0.      1.      0.
$
ENDDATA
```

Listing 7-2. Hinge Joint Using an RBAR

A word of caution: if you intentionally use coincident grid points (as in this example), you may run the risk of removing them accidentally if you later use the “equivalence” option available in most preprocessors. Equivalencing grid points causes all duplicate grid points in your model to be deleted; however, this may not be your intention as illustrated in this example.

7.5 The RBE2 Element

Adjacent elements whose stiffness differ by several orders of magnitude or more can produce an ill-conditioned stiffness matrix, which leads to problems when solving the model. For example, if you were to use a CBAR element with extremely large values of I1 and I2 to simulate a rigid connection, an ill-conditioned stiffness matrix would likely arise. The RBE2 (Rigid Body Element, type 2) element defines a rigid body whose independent degrees of freedom are specified at a single point and whose dependent degrees of freedom are specified at an arbitrary number of points. The RBE2 element uses constraint equations to couple the motion of the dependent degrees of freedom to the motion of the independent degrees of freedom. Consequently, RBE2 elements do not contribute directly to the stiffness matrix of the structure and ill-conditioning is avoided.

The RBE2 provides a very convenient tool for rigidly connecting the same components of several grid points together. You should note that multiple RBARs or an RBE1 can be used wherever an RBE2 is used; however, they may not be as convenient.

The format for the RBE2 entry is as follows:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-					

Field	Contents
EID	Element identification number.
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned.
CM	Component numbers of the dependent degrees of freedom in the global coordinate system at grid points GMi.
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned.

See Also

- "RBE2" in the *NX Nastran Quick Reference Guide*

When using an RBE2, you need to specify a single independent grid point (the GN field) in which all six components are assigned as independent. In the CM field, you can specify the dependent degrees of freedom at grid points GMi in the global coordinate system. The GMi grid points are the grid points at which the dependent degrees of freedom are assigned. The dependent components are the same for all the listed grid points (if this is unacceptable, use the RBAR elements, multiple RBE2s elements, or the RBE1 element).

As an example showing the use of an RBE2 element, consider the tube shown in Figure 7-6. The goal is to maintain a circular cross section at the end of the tube while applying a torque about the axis of the tube. Furthermore, you allow the tube to expand in the R direction, but the center of the end of the tube should not move from its original position.

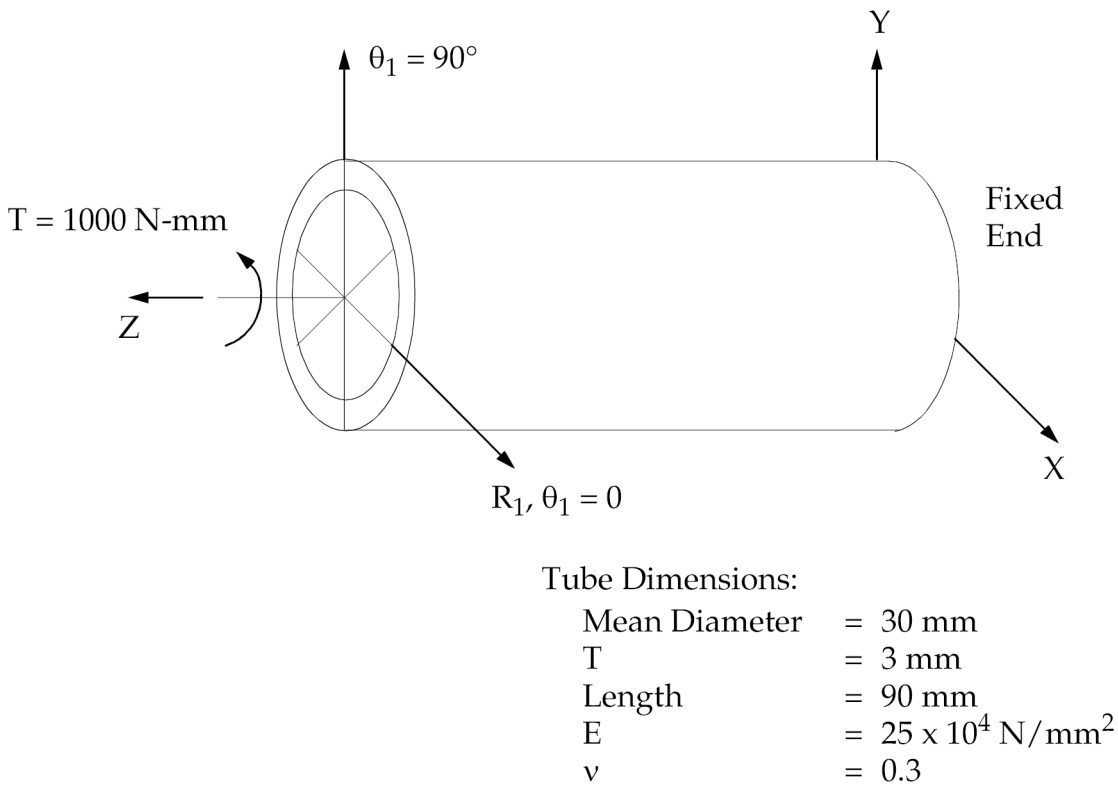


Figure 7-6. Tube with an End Torque Loading

The input file for this model is given in Lisitng 7-3. The goal is to have the end of the tube rotate uniformly while allowing the tube to expand in the R-direction. The simplest way to accomplish this is to define a local cylindrical coordinate system with the origin located at the center of the free end as shown in Figure 7-6. This cylindrical coordinate system is then used as the displacement coordinate system (field 7) for all of the grid points located at the free end. Now when an RBE2 element is connected to these grid points, the dependent degrees of freedom are in the local coordinate system. By leaving the R-direction independent, the tube is free to expand in the radial direction.

Grid point 999 is defined at the center of the free end to serve as the independent point. The θ, Z, R_r , R_θ , and R_z components of the grid points on the end of the tube are dependent degrees of freedom. To ensure that the axis of the tube remains in the same position, an SPC is applied to all components of grid point 999 except in the R_Z direction, which is the component about which the torque is applied. It is interesting to note that the CD field of grid point 999, the independent point, is different than that of the CD field of the dependent point, and this is an acceptable modeling technique. Furthermore, you should always use a rectangular coordinate system in the CD field for any grid point that lies on the polar axis. In this example, grid point 999 lies on the Z-axis (see Figure 7-6); therefore, it should not use coordinate system 1 for its CD field.

```
$ FILENAME - TORQUE.DAT
ID     LINEAR,TORQUE
SOL    101
TIME   5
CEND
TITLE = TUBE WITH END TORQUE
SET 1 =110, 111, 112, 119, 120, 127, 128,132
DISP = 1
SPC = 1
LOAD = 1
BEGIN BULK
PARAM   AUTOSPC YES
$
CORD2C  1       0       0.0     0.0     90.     0.0     0.0     91.0
        1.0     0.0     91.0
$
GRID    101             15.     0.0     0.0
GRID    102             10.6066 10.6066 0.0
GRID    103             7.105-1515.     0.0
GRID    104             15.     0.0     30.
GRID    105             10.6066 10.6066 30.
GRID    106             1.066-1415.     30.
GRID    107             15.     0.0     60.
GRID    108             10.6066 10.6066 60.
GRID    109             1.066-1415.     60.
GRID    110             15.     0.0     90.     1
GRID    111             10.6066 10.6066 90.     1
GRID    112             1.421-1415.     90.     1
GRID    113             -10.606610.6066 0.0
GRID    114             -15.    7.105-150.0
GRID    115             -10.606610.6066 30.
GRID    116             -15.    1.066-1430.
GRID    117             -10.606610.6066 60.
GRID    118             -15.    1.066-1460.
GRID    119             -10.606610.6066 90.     1
GRID    120             -15.    1.421-1490.     1
GRID    121             -10.6066-10.60660.0
GRID    122             0.0     -15.    0.0
GRID    123             -10.6066-10.606630.
GRID    124             -1.78-14-15.    30.
GRID    125             -10.6066-10.606660.
GRID    126             -3.2-14 -15.    60.
GRID    127             -10.6066-10.606690.     1
```

```
GRID    128             -4.97-14-15.    90.     1
GRID    129             10.6066 -10.60660.0
GRID    130             10.6066 -10.606630.
GRID    131             10.6066 -10.606660.
GRID    132             10.6066 -10.606690.     1
GRID    999             0.0     0.0     90.
$
RBE2    200     999     23456   110     111     112     119     120     +
+       127     128     132
$QUAD4S REMOVED, SEE THE FILE ON THE DELIVERY MEDIA
$ THIS SECTION CONTAINS THE LOADS, CONSTRAINTS, AND CONTROL BULK DATA ENTRIES
$
MOMENT  1       999     0       1000.   0.0     0.0     1.
$
SPC1    1       123456  101     102     103     113
SPC1    1       123456  114     121     122     129
SPC1    1       12345   999
$
$ THIS SECTION CONTAINS THE PROPERTY AND MATERIAL BULK DATA ENTRIES
$
PSHELL  1       1       3.      1
$
MAT1    1       250000.         .3
ENDDATA
```

Listing 7-3. Applying a Torque to a Tube Using an RBE2

The displacement vector of the end grid points is shown in Figure 7-7. The q direction T2 is the same for each of the end grid points as desired. The R-direction T1 is small but not exactly zero, indicating that the tube is permitted to expand in the radial direction.

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2
110	G	1.487426E-20	2.526745E-04	.0	.0	.0
111	G	8.858868E-20	2.526745E-04	.0	.0	.0
112	G	-4.178414E-21	2.526745E-04	.0	.0	.0
119	G	-1.985002E-20	2.526745E-04	.0	.0	.0
120	G	-4.204334E-20	2.526745E-04	.0	.0	.0
127	G	4.297897E-20	2.526745E-04	.0	.0	.0
128	G	8.858550E-21	2.526745E-04	.0	.0	.0
132	G	-9.068584E-20	2.526745E-04	.0	.0	.0

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
110	G	1.487426E-20	2.526745E-04	.0	.0	.0	1.684497E-05
111	G	8.858868E-20	2.526745E-04	.0	.0	.0	1.684497E-05
112	G	-4.178414E-21	2.526745E-04	.0	.0	.0	1.684497E-05
119	G	-1.985002E-20	2.526745E-04	.0	.0	.0	1.684497E-05
120	G	-4.204334E-20	2.526745E-04	.0	.0	.0	1.684497E-05
127	G	4.297897E-20	2.526745E-04	.0	.0	.0	1.684497E-05
128	G	8.858550E-21	2.526745E-04	.0	.0	.0	1.684497E-05
132	G	-9.068584E-20	2.526745E-04	.0	.0	.0	1.684497E-05

Figure 7-7. Selected Output for the RBE2 Example

RBE2 Example

A stiffened plate is modeled with two CQUAD4 elements and a CBAR element representing the stiffener, as shown in Figure 7-8. Two RBE2 elements are used to connect the CBAR stiffener to the plate elements.

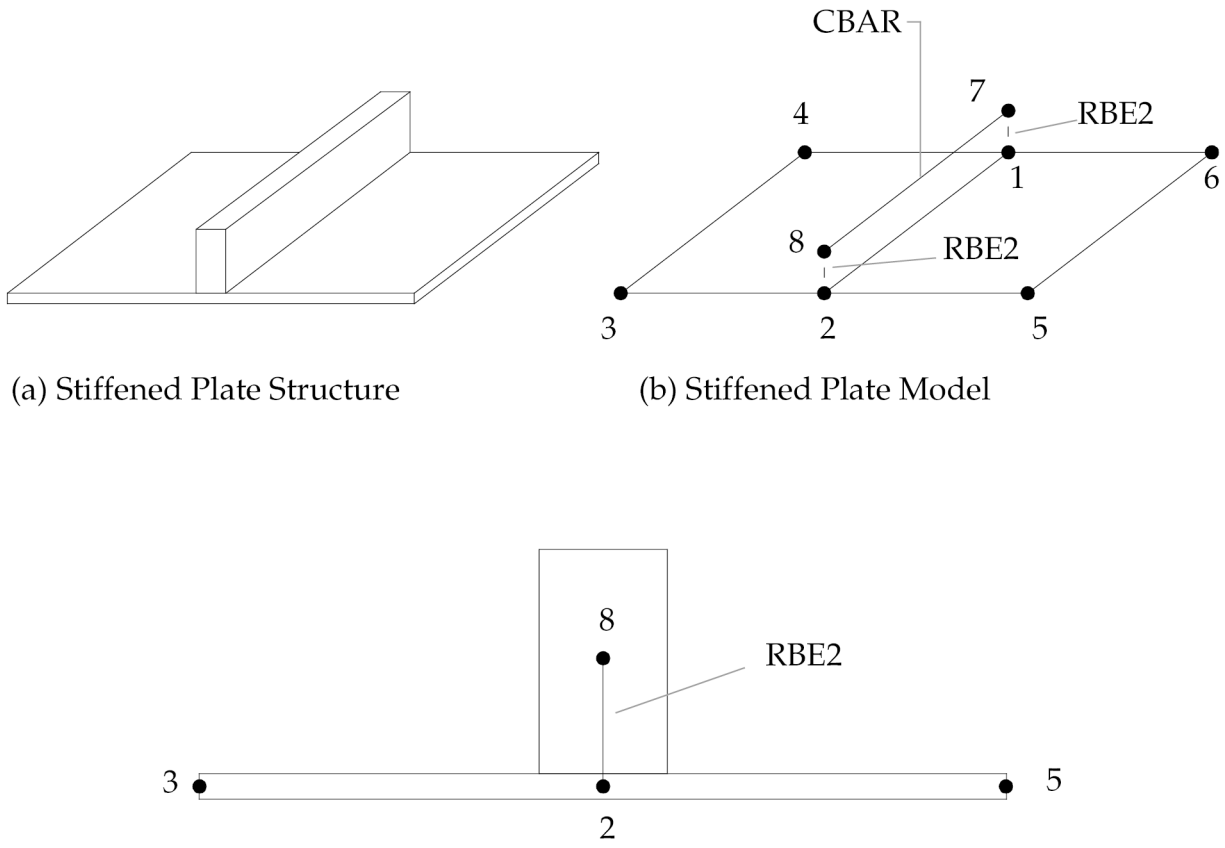


Figure 7-8. RBE2 Example

Grid points 7 and 8 are at the ends of the CBAR element and lie along the stiffener's neutral axis. An RBE2 element connects all six dependent degrees-of-freedom at grid point 7 (on the beam) to all six independent degrees-of-freedom at grid point 1 (on the plate). There is a similar element at the other end of the beam. Remember that an RBE2 element is not a finite element, but a set of equations that define a kinematic relationship between different displacements.

The required RBE2 entries are written as follows:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4		
RBE2	12	1	123456	7					
RBE2	13	2	123456	8					

7.6 The RBE3 Element

The RBE3 element is a powerful tool for distributing applied loads and mass in a model. Unlike the RBAR and RBE2 elements, the RBE3 doesn't add additional stiffness to your structure. Forces and moments applied to reference points are distributed to a set of independent degrees of freedom based on the RBE3 geometry and local weight factors.

The format of the RBE3 entry is as follows:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	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.-			

Field	Contents
EID	Element identification number. Unique with respect to other rigid elements.
REFGRID	Reference grid point identification number.
REFC	Component numbers at the reference grid point.
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j.
Ci	Component numbers with weighting factor WTi at grid points Gi,j.
Gi,j	Grid points whose components Ci have weighting factor WTi in the averaging equations.
"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.
GMi	Identification numbers of grid points with degrees of freedom in the *m* -set.
CMi	Component numbers of GMi to be assigned to the *m* -set.

See Also

- "RBE3" in the *NX Nastran Quick Reference Guide*

The manner in which the forces are distributed is analogous to the classical bolt pattern analysis. Consider the bolt pattern shown in Figure 7-9 with a force and moment M acting at reference point A. The force and moment can be transferred directly to the weighted center of gravity location along with the moment produced by the force offset.

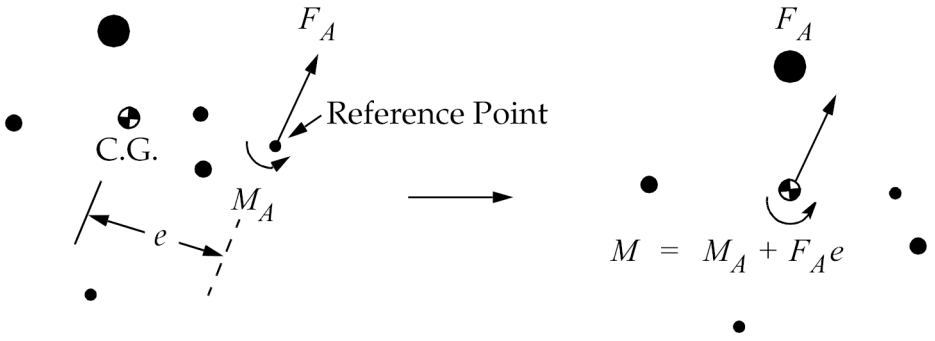
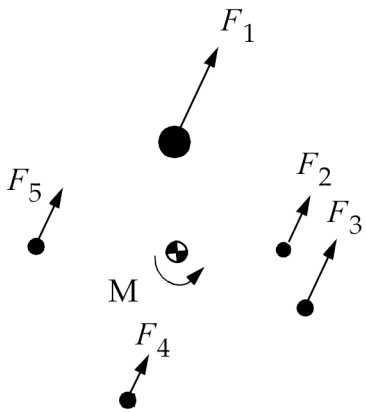
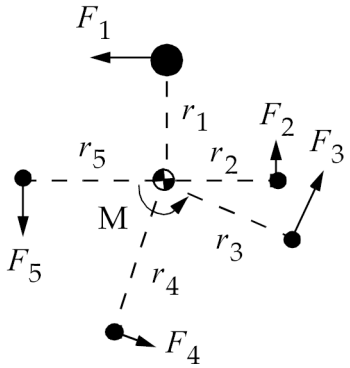


Figure 7-9. RBE3 Equivalent Force and Moment at the Reference Point

The force is distributed to the bolts proportional to the weighting factors. The moment is distributed as forces, which are proportional to their distance from the center of gravity times their weighting factors, as shown in Figure 7-11. The total force acting on the bolts is equal to the sum of the two forces. These results apply to both in-plane and out-of-plane loadings.



$$\text{where } F_i = F_A\left(\frac{\omega_i}{\sum \omega_i}\right)$$



$$\text{where } F_i = \left(\frac{M\omega_i r_i}{\omega_1 r_1^2 + \omega_2 r_2^2 + \omega_3 r_2^2 + \omega_3 r_3^2 + \omega_5 r_5^2}\right)$$

where:

F_i = force at DOF i

ω_i = weighting factor for DOF i

r_i = radius from the weighted center of gravity to point i

Figure 7-10. RBE3 Force Distribution

As an example, consider the cantilever plate modeled with a single CQUAD4 element shown in Figure 7-11. The plate is subjected to nonuniform pressure represented by a resultant force acting at a distance of 10 mm from the center of gravity location. The simplest way to apply the pressure is to use an RBE3 element to distribute the resultant load to each of the four corner points.

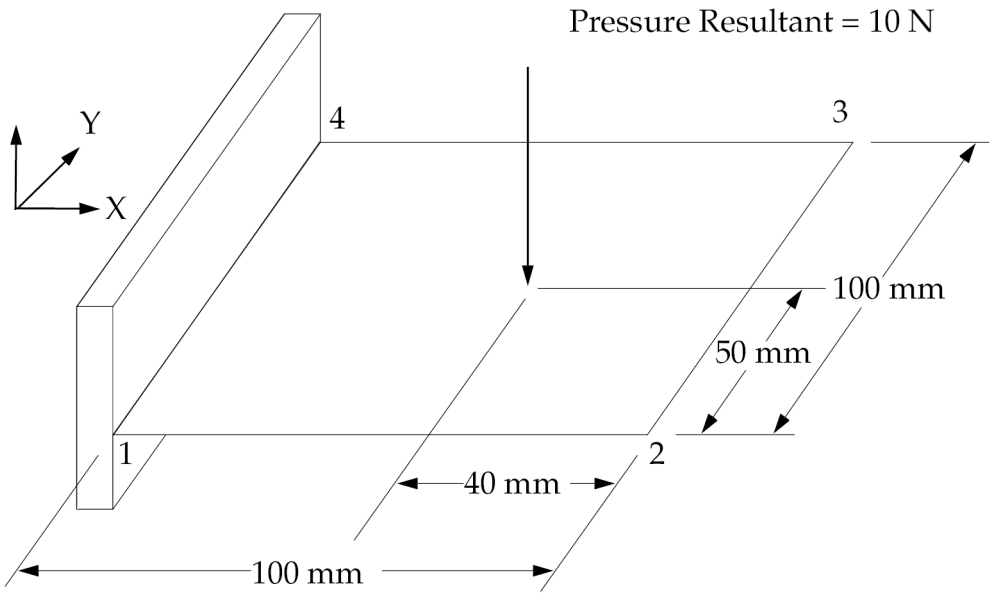


Figure 7-11. Using an RBE3 to Represent a Nonuniform Pressure Load

The input file representing this example is shown in Listing 7-4. Grid point 99 is called the REFGRID and is the location where the force is applied. This point is connected only to those degrees of freedom listed on the REFC field (the T3 component in this example). The default action of this element is to place the REFC degrees of freedom in the *m* -set. The element has provisions to place other DOFs in the *m* -set instead. However, this is an advanced feature and is beyond the scope of this user's guide. The groups of connected grid points begin in field 5. For this example, the connected grid points are the corner points.

```
$ FILENAME - RBE3.DAT
ID     LINEAR,RBE3
SOL    101
TIME   5
CEND
TITLE = SINGLE ELEMENT WITH RBE3
SPC  = 1
LOAD = 1
OLOAD     = ALL
GPFORCE   = ALL
SPCFORCES = ALL
BEGIN BULK
$
RBE3    10              99      3       1.0     123     1       2
        3       4
FORCE   1       99              100.    0.      0.      1.
$
PARAM   POST    0
$
GRID    1               0.      0.      0.
GRID    2               100.    0.      0.
GRID    3               100.    100.    0.
GRID    4               0.      100.    0.
GRID    99              60.     50.     0.
$
PSHELL  1       4       10.     4
$
MAT1    4       4.E6            0.
$
```

```
CQUAD4  1       1       1       2       3       4
$
SPC1    1       123456  1       4
ENDDATA
```

Listing 7-4. Distributing Force with an RBE3

The start of a group is indicated by a real number WTi, which is used as a weighting factor for the grid points in the group. In this example, a simple distribution based only on the geometry of the RBE3 is desired so that a uniform weight is applied to all points. The weighting factors are not required to add up to any specific value. For this example, if the WT1 field is 4.0 instead of 1.0, the results will be the same.

The independent degrees of freedom for the group are listed in the Ci field. Note that all three translational DOFs are listed even though the REFC field does not include the T1- and T2-direction. All three translational DOFs in the Ci field are included because the DOFs listed for all points must be adequate to define the rigid body motion of the RBE3 element even when the element is not intended to carry loads in certain directions. If any translational degrees of freedom are not included in C1 in this example, a fatal message is issued.

The element described by this RBE3 entry does not transmit forces in the T1- or T2-direction. The two reasons for this are that the reference grid point is not connected in this direction and all of the connected points are in the same plane. Note that the rotations are not used for the independent DOFs. In general, it is recommended that only the translational components be used for the independent degrees of freedom.

A selected portion of the output file produced by this example is shown in Figure 7-12.

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	6.000000E-04	1.567011E-20	-9.000000E-06	.0
3	G	.0	.0	6.000000E-04	3.282253E-21	-9.000000E-06	.0
4	G	.0	.0	.0	.0	.0	.0
99	G	.0	.0	3.600000E-04	.0	.0	.0

L O A D V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
99	G	.0	.0	1.000000E+02	.0	.0	.0

F O R C E S O F S I N G L E - P O I N T C O N S T R A I N T

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0
4	G	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0

G R I D P O I N T F O R C E B A L A N C E

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0
1	1	QUAD4	.0	.0	3.000000E+01	-3.681428E-12	-3.000000E+03	.0
1		*TOTALS*	.0	.0	-2.000000E+01	3.594530E-12	4.092726E-12	.0
2	1	QUAD4	.0	.0	-3.000000E+01	8.115214E-12	1.315632E-11	.0
2		*TOTALS*	.0	.0	-3.000000E+01	8.115214E-12	1.315632E-11	.0
3	1	QUAD4	.0	.0	-3.000000E+01	7.376250E-12	-6.994877E-12	.0
3		*TOTALS*	.0	.0	-3.000000E+01	7.376250E-12	-6.994877E-12	.0
4		F-OF-SPC	.0	.0	-5.000000E+01	7.275958E-12	3.000000E+03	.0
4	1	QUAD4	.0	.0	3.000000E+01	-4.079332E-12	-3.000000E+03	.0
4		*TOTALS*	.0	.0	-2.000000E+01	3.196626E-12	4.547474E-13	.0
99		APP-LOAD	.0	.0	1.000000E+02	.0	.0	.0
99		*TOTALS*	.0	.0	1.000000E+02	.0	.0	.0

Figure 7-12. Select Output for the RBE3 Example

The displacement of grid points 1 and 4 is zero due to the SPC applied to these points. The sum of the SPC forces at these two grid points is equal to the load applied to the reference grid point. The load transmitted to the corner points can be seen by inspecting the GPFORCE output. The force applied to the points due to the R-type elements and MPC entries is not listed specifically in the GPFORCE output. These forces show up as unbalanced totals (which should typically be equal to numeric zero). The forces applied to the corner grid points 1 through 4 are -20, -30, -30, and -20 N, respectively.

The most common usage of the RBE3 element is to transfer motion in such a way that all six DOFs of the reference point are connected. In this case, all six components are placed in the REFC field, and only components 123 are placed in the Ci field.

The load distributing capability of the RBE3 element makes it an ideal element to use to apply loads from a coarse model (or hand calculation) onto a detailed model of a component. For example, the shear distribution on a cross section is a function of the properties of that section. This shear loading may be applied to a cross section by performing a calculation of the shear distribution based on unit loading and using an RBE3 element with appropriate weighting factors for each grid point. In this manner, only one shear distribution need be calculated by hand. Since there are usually multiple loading conditions to be considered in an analysis, they may be applied by defining different loads to the dependent point on the RBE3 element.

For example, consider the tube attached to a back plate as shown in Figure 7-13. Suppose that you are not particularly interested in the stress in the tube or the attachment, but you are concerned

about the stresses in the back plate. For this reason, you choose not to include the tube in the model; however, you want the load transferred from the tube into the back plate attachment to be approximately correct. The question is: How should the loads be applied to the attachment to simulate the behavior of the tube?

Engineering principles dictate that the Z forces (forces acting normal to the back plate) acting on the attachment vary linearly as a function of the distance from the neutral axis. The simplest method of distributing the Z forces to the attachment grid is with an RBE2 or an RBE3 element. If an RBE2 is used, the attachment ring is rigid in the Z-direction. If an RBE3 element is used, no additional stiffness is added to the attachment ring. It is an engineering decision regarding which element to use since both are approximations. For this example, use the RBE3 element. Since the weight factors for the grid points in the Z-direction are equal, the forces are distributed to the grid points based on the geometry of the grid pattern only.

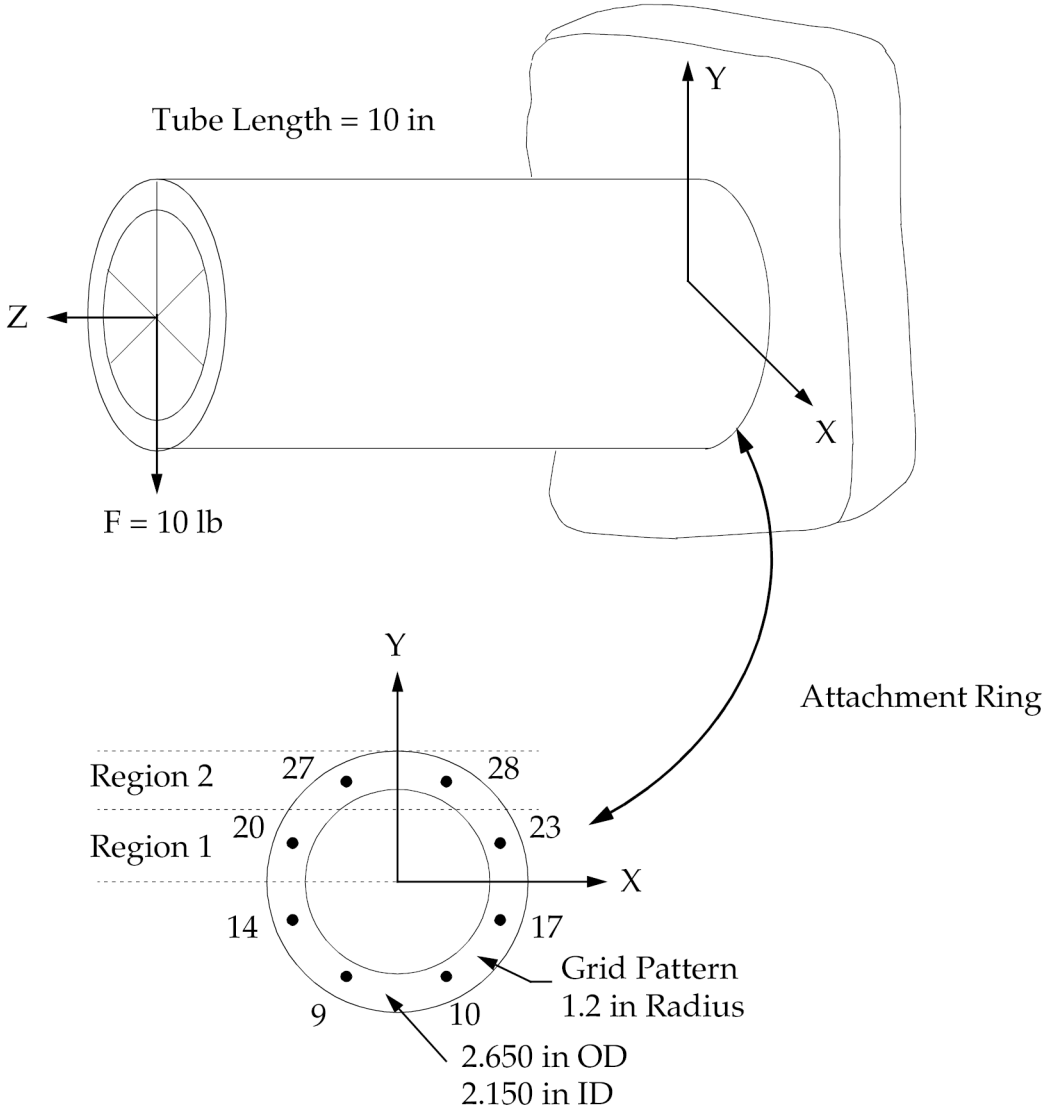


Figure 7-13. Attachment Ring

The shear force acting on the attachment doesn't act linearly; it is maximum at the neutral plane and tapers to zero at the top and bottom fibers (if the tube is solid, the shear distribution is a quadratic function, but in our example, it is a thick walled tube). The first step is to calculate the shear forces acting on the attachment ring as a function of the distance from the neutral plane using the classical strength of materials calculations. The result of this calculation is shown in Figure 7-14. The shear force curve is divided into two regions, each region representing a grid point region as shown in Figure 7-13. The area under the curve for each region represents the portion of the shear force transmitted to the grid points within the region. Using these area values as the coefficients for an RBE3 entry, the RBE3 distributes the shear force in a manner similar to the shear force curve.

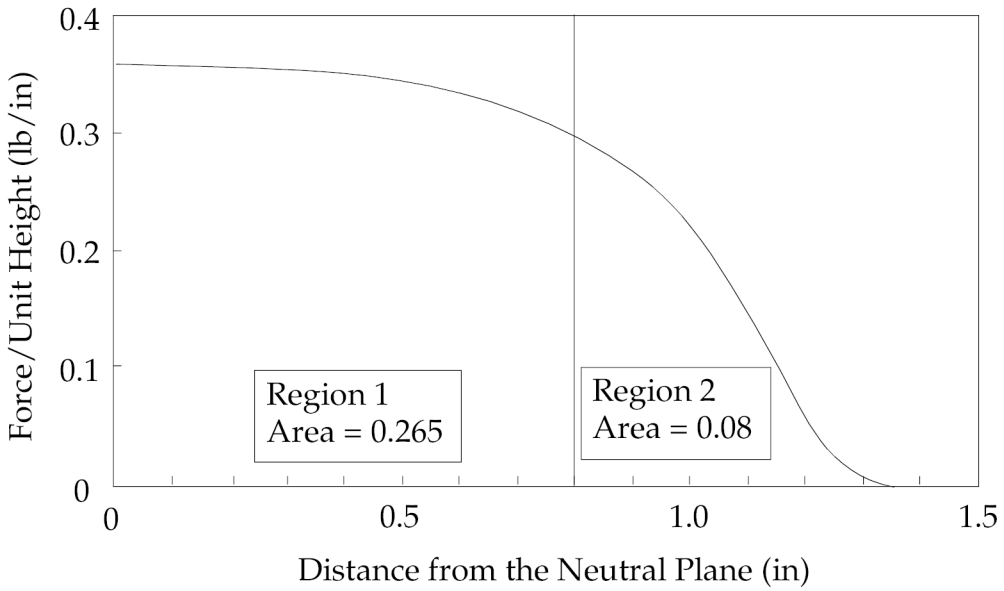


Figure 7-14. Shear Force on the Attachment Ring

The input file for this example is shown in Lisitng 7-5. The reaction force and moment due to the applied load of 10 lb acting at the end of the tube are 10 lb and 100 in-lb, respectively.

```
$FILENAME - SHEAR1.DAT
ID     LINEAR,SHEAR1
SOL    101
TIME   5
CEND
TITLE = SHEAR TEST CASE USING AN RBE3
SET 1 = 9,10,14,17,20,23,27,28
GPFORCE = 1
SPC = 1
LOAD = 1
BEGIN BULK
PARAM   POST    0
PARAM   AUTOSPC YES
$
$ RIGID CONNECTION USING TWO RBE3
$
RBE3    100             99      3456    1.0     123     9       10
        14      17      20      23      27      28
RBE3    101             99      12      0.08    123     9       10
        27      28      0.265   12      14      17      20      23
$
GRID    99              2.0     2.0     0.0
```

```
$
FORCE   1       99              1.      0.0     1.0     0.0
MOMENT  1       99              1.      1.0     0.0     0.0
$
$ ONLY THE END GRIDS ARE SHOWN
$
GRID    9               1.6     .8      0.0
GRID    10              2.4     .8      0.0
GRID    14              .8      1.6     0.0
GRID    17              3.2     1.6     0.0
GRID    20              .8      2.4     0.0
GRID    23              3.2     2.4     0.0
GRID    27              1.6     3.2     0.0
GRID    28              2.4     3.2     0.0
$
$ QUAD4S, PSHELL, MAT1, AND SPC NOT SHOWN
$
ENDDATA
```

Listing 7-5. Distributing the Attachment Forces with RBE3

A full model including the model of the tube was also generated for comparison. The grid point forces for the attachment points for both the simple model described above and the full model are summarized in Table 7-3. As can be seen, the force distribution from the RBE3 is close to that of the full model. This approximation is acceptable for many applications. The disadvantage of using this type of simplified model is that the stiffness of the tube is neglected. The full model is located on the delivery media with the name “shear1a.dat”.

Table 7-3. Comparison of RBE3 Attachment Forces to the Full Model

GridPoint	F_Y		F_Z	
	RBE3 Model	Full Model	RBE3 Model	Full Model
8,9	–0.058	–0.077	0.188	0.180
14,17	–0.192	–0.172	0.063	0.074
20,23	–0.192	–0.172	–0.063	–0.074
27,28	–0.058	–0.077	–0.188	–0.180

The most common user error in RBE3 element specification results from placing 4, 5, or 6 in the Ci (independent DOF) field in addition to the translation components. The rotations of the dependent point are fully defined by the translational motion of the independent points. The ability to input 4, 5, or 6 in the Ci field is only for special applications, such as when all of the connected points are colinear.

Small checkout models are recommended whenever you are specifying elements with nonuniform weight factors, asymmetric geometry or connected degrees of freedom, or irregular geometry. Using small checkout models is especially necessary when the reference point is not near the center of the connected points.

In summary, the intended use of the RBE3 element is to transmit forces and moments from a reference point to several non-colinear points. The rotation components 4, 5, and 6 should be placed in the Ci field only for special cases, such as when the independent points are colinear.

Changes to Handling of RBE3 Elements

The RBE3 entry defines a reference grid point (REFGRID, field 4) and connected grid points and components (Gij, Ci) in subsequent fields. If only the translational degrees-of-freedom are listed for

the connected grid points, the behavior of the element is unchanged from that of MSC.Nastran® Version 70.5 (MSC.Nastran is a registered trademark of MSC.Software Corporation). The element theory has been modified with result changes when rotations of the connected grid points (Ci=4, 5, 6) are included. The reference grid point theory, on the other hand, remains unchanged; there are no new (or even old) restrictions on reference grid point rotations. Changes for results are avoided in MSC.Nastran® Version 70.7 and NX Nastran if you use translations rather than rotations on the connected grid points.

However, several third-party graphical user interfaces (GUIs) connect these rotational degrees-of-freedom by default, which can lead to some cases of substantial differences in answers. We recommend that the default degrees-of-freedom for the connected grid points be set to the translational degrees-of-freedom only (i = 1, 2, 3). Again, no such restriction exists for the reference grid.

In a limited number of contexts, it makes modeling sense to connect these rotational degrees-of-freedom. For example, when all of the connected points are colinear, the element will be unstable for rotations about this axis. A rotation component on one of the connected grid points that has a component parallel to the line of connected points will solve this problem. The results will then be the same on all systems new and old, because this is a statically determinate load path, and there is only one (automatically adjusted) coefficient that will provide equilibrium.

If there are many connected grid points spread over a surface or volume and one compares the results first with no connected rotations, and then with all rotations connected one often finds that the more highly connected model is actually softer. While this is counter-intuitive for flexible finite elements, it is to be expected for spline constraint elements such as the RBE3. If more points are connected and they have lower stiffness than the originally connected points, the overall stiffness of the reference point is dominated by the softer connected rotational stiffness effects. The net effect is that connecting more degrees-of-freedom often reduces the reference point stiffness, rather than increasing it as one would expect.

Chapter 8: Beam Cross Sections

- *Using Supplied Beam and Bar Libraries*
- *Adding Your Own Beam Cross Section Library*

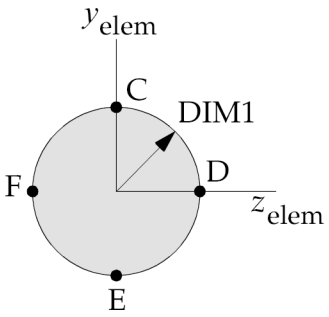
8.1 Using Supplied Beam and Bar Libraries

The Bulk Data entries PBARL and PBEAML provide libraries of cross-sectional shapes. For open sections and the HAT1 section, the equations the software uses to calculate cross-sectional properties are based on thin wall assumptions and may produce inaccurate results if the cross section contains thick walls. If so, you should consider using the mesh-based Pilkey method to calculate the cross-sectional properties. For more information on the mesh-based Pilkey method, see Using PBARL to Define Beam Cross Section Properties and Using PBEAML to Define Beam Cross Section Properties.

The following pages list the equations that the software uses to calculate PBEAM and PBAR cross-sectional property entries from PBEAML or PBARL cross-sectional dimension entries. Symbols absent for a particular cross section are normally set to zero.

A	Cross-sectional area.
y_c	Distance to centroid along Y element axis.
z_c	Distance to centroid along Z element axis.
y_s	Distance to shear center along Y element axis.
z_s	Distance to shear center along Z element axis.
I_1	Moment of inertia about the Z element axis at the centroid. $I_1 = I_{(zz)elem}$
I_2	Moment of inertia about the Y element axis at the centroid. $I_2 = I_{(YY)elem}$
I_{12}	Product moment of inertia at the centroid. $I_{12} = I_{(ZY)elem}$
J	Torsional Stiffness Constant.
C, D, E, F	Location of the stress recovery points in the element coordinate system relative to the shear center. For the PBARL the locations must be changed to be relative to the centroid. This can be done by adding y_{na}, z_{na} to the listed equations.
K_1, K_2	Shear stiffness factor for plane 1 and plane 2.
I_w	Warping coefficient for the cross section relative to the shear center.
y_{na}, z_{na}	Coordinates of the centroid relative to the shear center.

ROD Cross Section



$$r = \text{DIM1}$$

TYPE = "ROD"

Figure 8-1. Geometric Property Formulas for a ROD

$$A = \pi r^2$$

$$\left.\begin{matrix} I_1 \\ I_2 \end{matrix}\right\} = \frac{\pi}{4} r^4$$

$$J = \frac{\pi}{2} r^4$$

$$C = r, 0$$

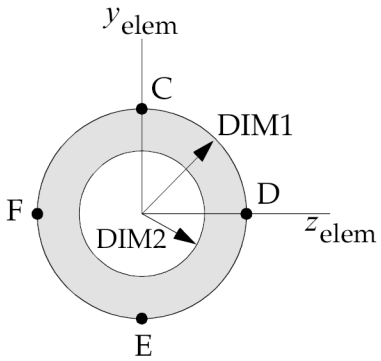
$$D = 0, r$$

$$E = -r, 0$$

$$F = 0, -r$$

$$\left.\begin{matrix} K_1 \\ K_2 \end{matrix}\right\} = 0.9$$

TUBE Cross Section



$$r_0 = \text{DIM1}$$

$$r_i = \text{DIM2}$$

TYPE = "TUBE"

Figure 8-2. Geometric Property Formulas for a TUBE Cross Section

$$A = \pi(r_0^2 - r_i^2)$$

$$\left.\begin{matrix} I_1 \\ I_2 \end{matrix}\right\} = \frac{\pi}{4}(r_0^4 - r_i^4)$$

$$J = \frac{\pi}{2}(r_0^4 - r_i^4)$$

$$C = r_0, 0$$

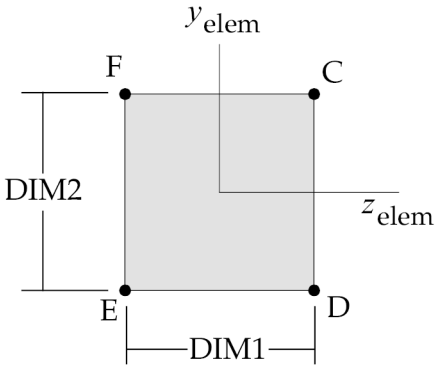
$$D = 0, r_0$$

$$E = -r_0, 0$$

$$F = 0, -r_0$$

$$\left.\begin{matrix} K_1 \\ K_2 \end{matrix}\right\} = 0.5$$

BAR Cross Section



b = DIM1
h = DIM2

TYPE = "BAR"

Figure 8-3. Geometric Property Formulas for a BAR

$$A = bh$$

$$I_1 = \frac{bh^3}{12}$$

$$I_2 = \frac{b^3h}{12}$$

$$I_{12} = 0.$$

$$J = bh^3\left[\frac{1}{3} - .21\frac{h}{b}\left(1 - \frac{h^4}{12b^4}\right)\right]$$

$$C = \frac{h}{2}, \frac{b}{2}$$

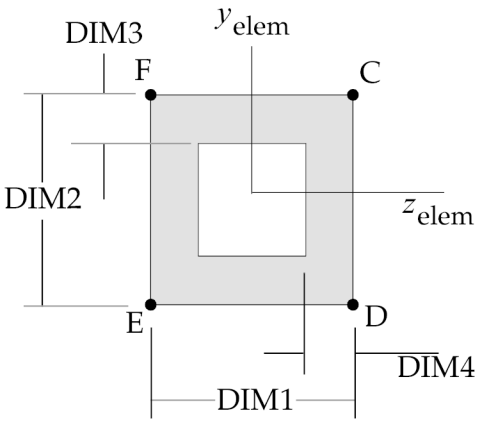
$$D = -\frac{h}{2}, \frac{b}{2}$$

$$E = -\frac{h}{2}, -\frac{b}{2}$$

$$F = \frac{h}{2}, -\frac{b}{2}$$

$$\left.\begin{matrix} K_1 \\ K_2 \end{matrix}\right\} = \frac{5}{6}$$

BOX Cross Section



TYPE = "BOX"

$$b = \text{DIM1}$$
$$h = \text{DIM2}$$
$$t_1 = \text{DIM3}$$
$$t_2 = \text{DIM4}$$
$$b_i = b - 2t_2$$
$$h_i = h - 2t_1$$

Figure 8-4. Geometric Property Formulas for a BOX

$$A = bh - b_i h_i$$

$$I_1 = \frac{bh^3}{12} - \frac{b_i h_i^3}{12}$$

$$I_2 = \frac{hb^3}{12} - \frac{h_i b_i^3}{12}$$

$$I_{12} = 0.$$

$$J = \frac{2t_2 t_1 (b - t_2)^2 (h - t_1)^2}{bt_2 + ht_1 - t_2^2 - t_1^2}$$

$$C = \frac{h}{2}, \frac{b}{2}$$

$$D = -\frac{h}{2}, \frac{b}{2}$$

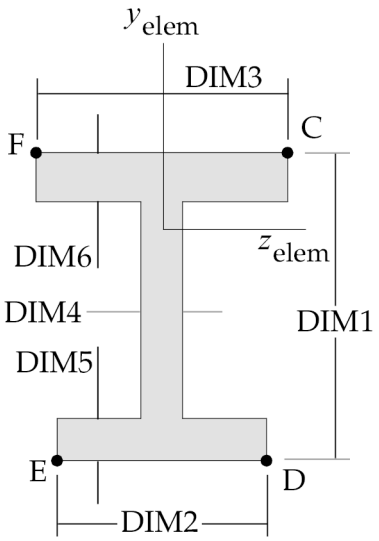
$$E = -\frac{h}{2}, -\frac{b}{2}$$

$$F = \frac{h}{2}, -\frac{b}{2}$$

$$K_1 = \frac{h_i t_2}{A} \cdot 2$$

$$K_2 = \frac{b_i t_1}{A} \cdot 2$$

I Cross Section



$$a = \text{DIM2}$$
$$b = \text{DIM3}$$
$$h = \text{DIM1}$$
$$t_a = \text{DIM5}$$
$$t_b = \text{DIM6}$$
$$t_w = \text{DIM4}$$
$$h_w = h - (t_a + t_b)$$
$$h_f = h - 0.5(t_a + t_b)$$

Figure 8-5. Geometric Property Formulas for an I Section

$$A = t_a a + h_w t_w + b t_b$$

$$y_c = \frac{1}{A}\left(\frac{1}{2} h_w (h_w + t_a) t_w + h_f t_b b\right)$$

$$y_s = \frac{t_b h_f b^3}{(t_b b^3 + t_a a^3)}$$

$$y_{na} = y_c - y_s$$

$$I_1 = \frac{bt_b^3}{12} + \frac{at_a^3}{12} + \frac{t_w h_w^3}{12} + (h_f - y_c)^2 bt_b + y_c^2 at_a + (y_c - 0.5(h_w + t_a))^2 h_w t_w$$

$$I_2 = \frac{b^3 t_b}{12} + \frac{t_a a^3}{12} + \frac{h_w t_w^3}{12}$$

$$I_{12} = 0.$$

$$J = \frac{1}{3}(t_b^3 b + t_a^3 a + t_w^3 h_f)$$

Note that I_c and y_s are relative to the center of flange defined by Dim1 and Dim4.

$$C = h_f - y_s + .5t_b, .5b$$
$$D = -y - .5t_a, .5a$$
$$E = -y - .5t_a, -.5a$$
$$F = h_f - y_s + .5t_b, -.5a$$

(continued)

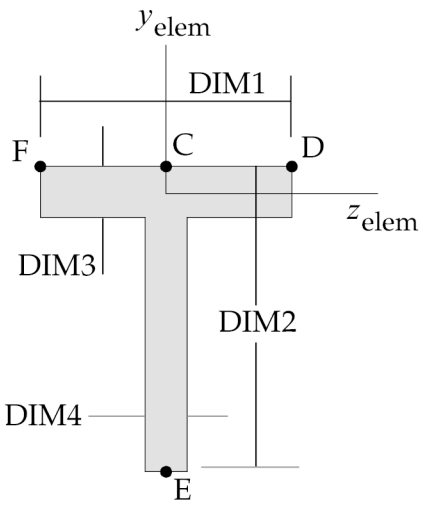
$$K_1 = \frac{t_w h_w}{A}$$

$$K_2 = \frac{5(at_a + bt_b)}{6A}$$

$$I_w = \frac{h^2 t_b t_a b^3 a^3}{12(t_b b^3 + t_a a^3)}$$

Note that I_w and y_s are based on thin walled formulations.

T Cross Section



$$d = \text{DIM1}$$
$$h = \text{DIM2} - 0.5t_f$$
$$t_f = \text{DIM3}$$
$$t_w = \text{DIM4}$$
$$h_w = \text{DIM2} - t_f$$

TYPE = "T"

Figure 8-6. Geometric Property Formulas for a T Section

$$A = dt_f + h_w t_w$$

$$y_{na} = -\frac{h_w t_w (h_w + t_f)}{2A}$$

$$I_1 = \frac{dt_f^3}{12} + \frac{t_w h_w^3}{12} + h_w t_w (y_{na} + .5(h_w + t_f))^2 + dt_f y_{na}^2$$

$$I_2 = \frac{t_f d^3}{12} + \frac{h_w t_w^3}{12}$$

$$I_{12} = 0.$$

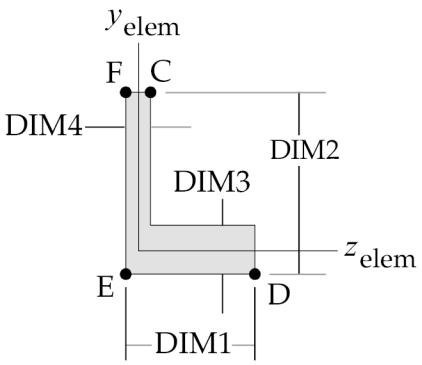
$$J = \frac{1}{3}(t_f^3 d + t_w^3 h)$$

$$C = .5t_f, 0$$
$$D = .5t_f, .5d$$
$$E = -h, 0$$
$$F = .5t_f, -.5d$$

$$K_1 = \frac{h_w t_w}{A}$$

$$K_2 = \frac{t_f d}{A}$$

L Cross Section



$$b = \text{DIM1} - 0.5t_2$$
$$h = \text{DIM2} - 0.5t_1$$
$$t_1 = \text{DIM3}$$
$$t_2 = \text{DIM4}$$
$$h_2 = \text{DIM2} - t_1$$
$$b_1 = \text{DIM1} - t_2$$

Figure 8-7. Geometric Property Formulas for an L Section

$$A = (b + .5t_2)t_1 + h_2t_2$$

$$y_c = \frac{t_2h_2(h_2 + t_1)}{2A}$$

$$z_c = \frac{t_1b_1(b_1 + t_2)}{2A}$$

$$I_1 = \frac{t_1^3(b + .5t_2)}{12} + t_1(b + .5t_2)y_c^2 + \frac{t_2h_2^3}{12} + h_2t_2(0.5(h_2 + t_1) - y_c)^2$$

$$I_2 = \frac{t_2^3h_2}{12} + t_2h_2z_c^2 + \frac{t_1(b + .5t_2)^3}{12} + t_1(b + .5t_2)(.5b_1 - z_2)^2$$

$$I_{12} = z_cy_ct_1t_2 - b_1t_1y_c(.5(b_1 + t_2) - z_c) - h_2t_2z_c(.5(h_2 + t_1) - y_c)$$

$$J = \frac{1}{3}(t_3^3b + t_2^3h)$$

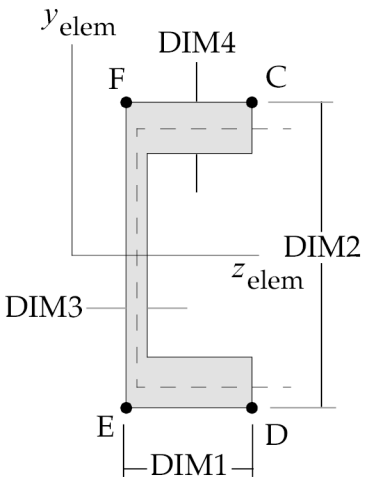
$$K_1 = \frac{h_2t_2}{A}$$

$$K_2 = \frac{b_1t_1}{A}$$

$$y_{na} = y_c$$

$$z_{na} = z_c$$

CHAN Cross Section



$$b = \text{DIM1} - .5t_w$$
$$h = \text{DIM2} - t_f$$
$$t_w = \text{DIM3}$$
$$t_f = \text{DIM4}$$
$$b_f = \text{DIM1} - t_w$$
$$h_w = \text{DIM2} - 2.t_f$$

TYPE = "CHAN"

Figure 8-8. Geometric Properties for a CHAN Cross Section

$$A = 2t_f b_f + (h + t_t)t_w$$

$$z_c = \frac{b_f t_f (b_f + t_w)}{A}$$

$$z_s = \frac{b^2 t_f}{\left(2bt_w + \frac{1}{3}ht_f\right)}$$

$$I_1 = \frac{h^2 t_f b_f}{2} + \frac{b_f t_f^3}{6} + \frac{(h + t_f)^3 t_w}{12}$$

$$I_2 = \frac{(h + t_f)t_w^3}{12} + \frac{b_f^3 t_f}{6} + .5(b_f + t_w)^2 b_f t_f - z_c^2 A$$

$$J = \frac{1}{3}(2bt_f^3 + ht_w^3)$$

$$I_w = \frac{t_f b^3 h^2}{12}\left(\frac{2t_w h + 3t_f b}{t_w h + 6t_f b}\right)$$

$$C = .5(h + .5t_f), z_s + b$$
$$D = -.5(h + .5t_f), z_s + b$$
$$E = -.5(h + .5t_f), z_s - .5t_w$$
$$F = .5(h + .5t_f), z_s - .5t_w$$

$$K_1 = \frac{t_w h_w}{A}$$

$$K_2 = \frac{2 t_f b_f}{A}$$

$$z_{na} = z_c + z_s$$

Note that z_c , z_s are distances measured relative to an origin positioned at the center of the web.

HAT1 Cross Section

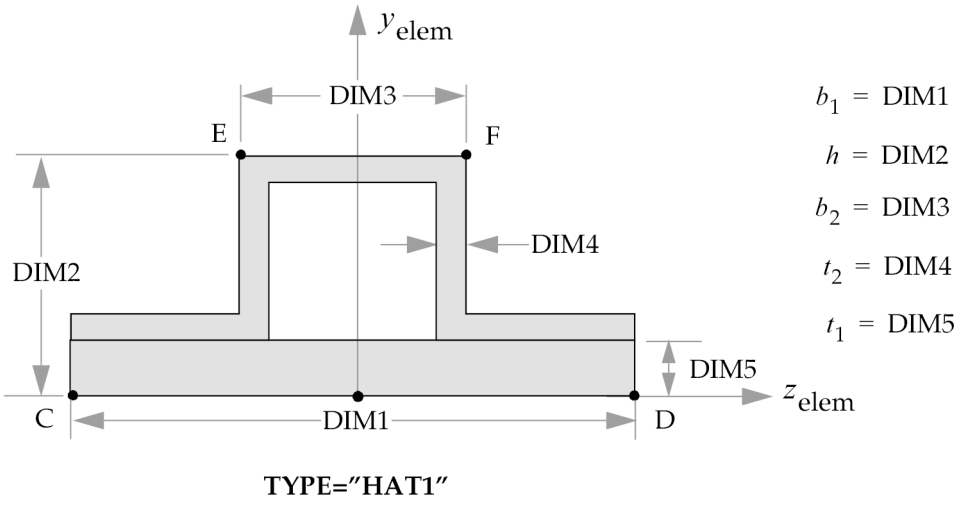


Figure 8-9. Geometric Property Formulas for a Closed-hat Section

$$A = b_1 t_1 + t_2(b_1 - b_2) + 2t_2(h - t_1) + t_2(b_2 - 2t_2)$$

$$y_c = \frac{(b_1 t_1^2/2) + t_2(b_1 - b_2)(t_1 + t_2/2) + t_2(h^2 - t_1^2) + t_2(b_2 - 2t_2)(h - t_2/2)}{A}$$

$$y_s = \frac{\psi}{\psi_1} + \frac{t_2}{2}$$

where:

$$\psi_1 = \frac{t_2(b_1 - b_2 + t_2)^3}{6} + \frac{t_2(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)^2}{16} + \frac{t_2(h - t_1 - t_2)(b_2 - t_2)^2}{2} + \frac{t_2(b_2 - t_2)^3}{12} + \frac{t_1 b_1^3}{12}$$

$$\psi = (b_2 - t_2)(h - t_1/2 - t_2/2)\left[\frac{t_1(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)}{24} + \frac{t_1 b_1^2}{12} + \psi_3\right]$$
$$+ (b_2 - t_2)(h - t_1 - t_2)\left[\frac{t_2(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)}{8} + \frac{t_2(h - t_1 - t_2)(b_2 - t_2)}{4} + \psi_3\right] + \frac{t_1 t_2 b_1^3}{12}$$
$$- \frac{t_1 t_2 (b_1 - b_2 + t_2)(b_1 + b_2 - t_2)(b_2 - t_2)}{8} + \frac{t_2^2 b_1^3}{12} - \frac{t_1 t_2 b_1^2 (b_2 - t_2)}{12}$$
$$+ \frac{t_1 t_2 (b_2 - t_2)(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)}{12} - \frac{t_2^2 (b_2 - t_2)(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)}{8} - \frac{t_2^2 b_1^2 (b_2 - t_2)}{12}$$
$$+ \frac{t_2^2 (b_2 - t_2)(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)}{12}$$

and

$$\psi_3 = \frac{t_1 t_2 \psi_2}{2t_1(h - t_1 - t_2) + (b_2 - t_2)(t_1 + t_2)}$$

where:

$$\psi_2 = \frac{(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)(h - t_1 - t_2)}{4} + \frac{(h - t_1 - t_2)^2 (b_2 - t_2)}{2} + \frac{(b_2 - t_2)^3}{12}$$
$$+ \frac{(b_1 - b_2 + t_2)(b_1 + b_2 - t_2)(b_2 - t_2)}{6} + \frac{(h - t_1 - t_2)(b_2 - t_2)^2}{2} + \frac{b_1^2 (b_2 - t_2)}{12}$$

$$y_{na} = y_s - (h - y_c)$$

$$I_1 = \frac{b_1 t_1^3}{12} + b_1 t_1 (y_c - t_1/2)^2 + \frac{(b_1 - b_2) t_2^3}{12}$$
$$+ (b_1 - b_2) t_2 (y_c - (t_1 + t_2/2))^2$$
$$+ \frac{t_2 (h - t_1)^3}{6} + 2t_2 (h - t_1)\left(y_c - \left(\frac{h - t_1}{2}\right) + t_1\right)^2$$
$$+ \frac{(b_2 - 2t_2) t_2^3}{12} + (b_2 - 2t_2) t_2 (y_c - (h - t_2/2))^2$$

$$I_2 = \frac{t_1 b_1^3}{12} + \frac{t_2 (b_1 - b_2)^3}{48} + \frac{t_2 (b_1 - b_2)(b_1 + b_2)^2}{16}$$
$$+ \frac{(h - t_1) t_2^3}{6} + \frac{(h - t_1) t_2 (b_2 - t_2)^2}{2}$$
$$+ \frac{2t_3 (b_2/2 - t_2)^3}{3}$$

$$I_{12} = 0$$

$$J = \frac{4t_1t_2(b_2 - t_2)^2(h - t_1/2 - t_2/2)^2}{(b_2 - t_2)(t_1 + t_2) + t_1(2h - t_1 - t_2)} + (b_1 - b_2)(t_1 + t_3)^3/3$$

$$C = y_s - h\ ,\ -b_1/2$$

$$D = y_s - h\ ,\ b_1/2$$

$$E = y_s\ ,\ -b_2/2$$

$$F = y_s\ ,\ b_2/2$$

$$K_1 = \frac{1}{n}\sum_{i=1}^{n}\frac{2(h - t_1)t_2}{mA}$$

$$K_2 = \frac{1}{n}\sum_{i=1}^{n}\frac{1 - 2(h - t_1)t_2}{mA}$$

where *n* is the number of sections and *m* = 2 for end A or end B and *m* = 1 for any intermediate station.

8.2 Adding Your Own Beam Cross Section Library

The standard cross sections provided with the software should be adequate in the majority of cases. If these standard sections are not adequate for your purposes, you can add your own library of cross sections to suit your needs. To add your own library, you need to write few simple subroutines in FORTRAN and link them to NX Nastran through inter-process communications.

The *NX Nastran Installation and Operations Guide* describes the current server requirements and provide you with the location of starter subroutines as described below.

This process requires writing and/or modifying up to eight basic subroutines:

1. BSCON – Defines the number of dimensions for each of the section types.
2. BSBRPD – Calculates section properties based on section dimensions.
3. BSGRQ – Defines NSECT, the number of section types, and NDIMAX, the maximum number of dimensions (including nonstructural mass) required by any of the sections.
4. BSBRT – Provides the name, number of dimensions and number of design constraints for each section type.
5. BSBRID – Provides information for the calculation of gradients of section properties with respect to section dimensions.

6. BSBRGD – Calculates any nonlinear gradients of section properties with respect to section dimensions.

7. BSBRCD – Defines constraints in the design of section dimensions.

8. BSMSG – A utility routine; handles errors that occur in the beam library.

BSCON and BSBRPD are always required. BSGRQ, BSBRT, and BSBRID are required if you wish to perform sensitivity and/or optimization tasks using the beam library. BSBRGD is required if you are providing nonlinear analytical sensitivities in the design task, and BSBRCD is an optional routine that can be provided to help the optimizer to stay within physical design constraints. BSMSG handles any error messages you feel are appropriate.

This section describes each of these basic routines. Routines that are called by these basic routines are also described with adequate examples to allow you to construct your own library. All the example routines shown are for the 32-bit machines.

For 64-bit machines, all the routine names that end with “D” should be changed to end with “S,” and all real variables must be designated as single precision instead of double precision. Therefore, the naming convention for routines on 64-bit machines are: BSCON, BSBRPS, BSGRQ, BSBRT, BSPRIS, BSBRGS, BSBRCS, and BSMSG.

BSCON SUBROUTINE

This routine provides the number of fields in the continuation lines to be read from the Bulk Data entries PBARL and PBEAML for each cross section in the library. The value of the ENTYP variable may be 0, 1, or 2. When ENTYP = 0, the value returned is the number of DIMi. When ENTYP = 1, the value returned includes both the DIMi and NSM fields. The value of 1 is used for PBARL only. When ENTYP = 2, the value returned includes the DIMi, NSM, SO, and XIXB fields for 11 different stations. The value of 2 applies to PBEAML only.

The calling sequence and example routine for the standard library is given below.

```
SUBROUTINE BSCON(GRPID,TYPE,ENTYP,NDIMI,ERROR)
C ------------------------------------------------------------------------
C   Purpose
C     To get the number of maximum fields in continuation entries for
C     each section in the library.
C
C   Arguments:
C
C    GRPID   input  integer      Integer id of this group or group name.
C                                Not used, reserved for future use.
C    TYPE    input  character*8  Name of cross section
C    ENTYP   input  integer      O: dimensions only without NSM
                                 1: PBARL, total # of data items for 2:PBEAML
C    NDIMI   output integer      Number of dimi fields for the 'ENTYP'
C                                section
C    ERROR   output integer      Error code
C
C   Called by BCCON
C ------------------------------------------------------------------------
C===  Argument Type Declaration
      INTEGER      GRPID,ENTYP,NDIMI,ERROR
      CHARACTER*8  TYPE
C===  Default to 'nothing wrong'
      ERROR = 0
C===  Dimensions vary with section type
      IF    ( TYPE.EQ.'ROD     ') THEN
```

```
               NDIMI = 1
         ELSEIF( TYPE.EQ.'TUBE    ' .OR. TYPE.EQ.'BAR     ') THEN
               NDIMI = 2
         ELSEIF( TYPE.EQ.'HEXA    ') THEN
               NDIMI = 3
         ELSEIF( TYPE.EQ.'BOX     ' .OR. TYPE.EQ.'T       ' .OR.
        +        TYPE.EQ.'L       '                          .OR.
        +        TYPE.EQ.'CHAN    ' .OR. TYPE.EQ.'CROSS   ' .OR.
        +        TYPE.EQ.'H       '                          .OR.
        +        TYPE.EQ.'I1      ' .OR. TYPE.EQ.'T1      ' .OR.
        +        TYPE.EQ.'CHAN1   '                          .OR.
        +        TYPE.EQ.'Z       ' .OR. TYPE.EQ.'CHAN2   ' .OR.
        +        TYPE.EQ.'T2      ' .OR. TYPE.EQ.'HAT     ') THEN
               NDIMI = 4
         ELSEIF( TYPE.EQ.'I       ' .OR. TYPE.EQ.'BOX1    ') THEN
               NDIMI = 6
         ELSE
C===     Set error code if invalid name for the section
               ERROR = 5150
               RETURN
         ENDIF
C===     Number of data items to be read on PBARL entry is DIMi
C===     plus the NSM field
         IF (ENTYP.EQ.1) NDIMI = NDIMI+1
C===     Number of data items to be read on PBEAML entry is DIMi
C===     plus the NSM, SO and X/XB fields for eleven different
C===     stations.
         IF (ENTYP.EQ.2) NDIMI = (NDIMI+3)*11
C-----------------------------------------------------------------
         RETURN
         END
```

BSBRPD Subroutine

Finite element analysis requires section properties such as area, moment of inertia, etc., instead of section dimension. Therefore, the dimensions specified on PBARL and PBEAML need to be converted to equivalent properties usually specified on PBAR and PBEAM entries. The images of all these entries are stored in EPT datablock as records.

BSBRPD subroutine is the interface of your properties evaluator with NX Nastran. You may use your own naming convention for the subroutines that calculate the cross-section properties from the dimensions. The calling tree used for the standard library is shown in Figure 8-10.

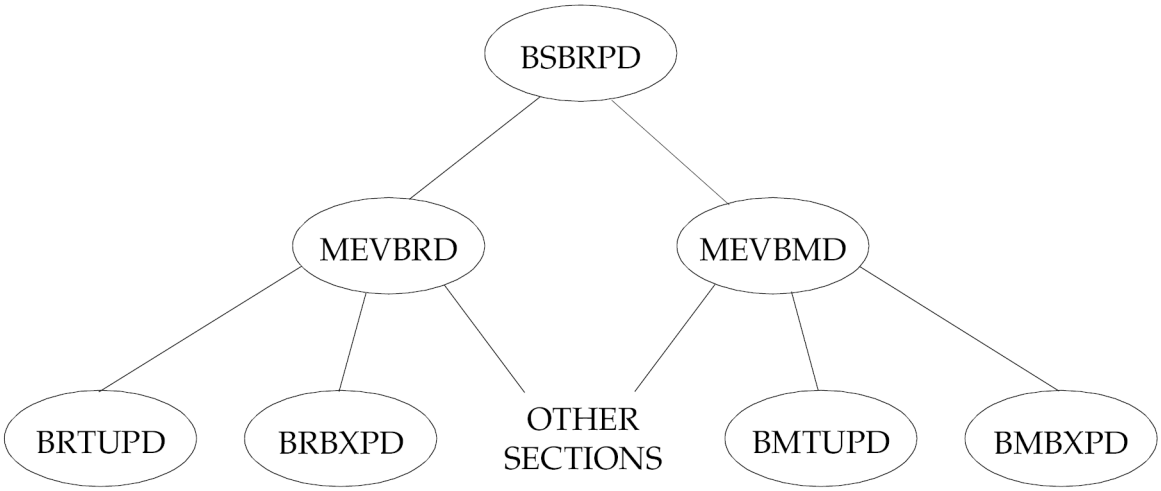


Figure 8-10. Calling Tree Generate Property Data

The BSBRPD calls the bar evaluator routine MEVBRD for the BAR element and the beam evaluator routine MEVBMD for the BEAM element. The evaluators in turn call the routines for each section. The routines are named BRXXPD and BMXXPD, where XX is a two-letter identifier for the section. For example, the routines for the TUBE section are called BRTUPD and BMTUPD. The details for various routines are given in Listing 8-1.

```
SUBROUTINE BSBRPD(GRPID,ENTYP,TYPE,IDI,NID,IDO,NIDO,DIMI,NDIMI,
     +                  DIMO,NDIMO,ERROR)
C=======================================================================
C   PURPOSE:  This is the interface for the properties evaluator with
C             NX Nastran.
C
C   ARGUMENTS
C       GRPID  input   integer      The ID of group name
C       ENTYP  input   integer      1: PBARL, 2: PBEAML
C       TYPE   input   character*8  Arrays for cross section types
C       IDI    input   integer      Array containing the integer words
C                                   in the PBARL or PBEAML EPT record
C       NID    input   integer      Dimension of the IDI array.  It is
C                                   equal to two.
C       IDO    output  integer      Array containing the integer words
C                                   in the PBAR or PBEAM EPT record.
C       NIDO   output  integer      Dimension of the IDO array.  It is
C                                   equal to two for PBAR and four for
C                                   the PEBAM EPT record.
C       DIMI   input   real         Array containing the floating words
C                                   in the PBARL or PBEAML EPT record
C                                   for the 'TYPE' section
C       NDIMI  input   integer      Dimension of the DIMI array.
C       DIMO   output  real         Array conatining the real words for
C                                   the PBAR or PBEAM EPT record.
C       NDIMO  input   integer      Dimension of the DIMO array.  It is
C                                   equal to 17 for the PBAR and 193 for
C                                   the PBEAM EPT record.
C       ERROR  output  integer      Error code
C
C-----------------------------------------------------------------------
C   CALLED BY:
C             BCBRP
C-----------------------------------------------------------------------
C   CALLS:
C         MEVBRD ,MEVBMD
```

```
C-----------------------------------------------------------------------
C   IMPLICIT DECLARATIONS
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION  (A-H,O-Z)
C.....................................................................
C   EXPLICIT DECLARATIONS
      INTEGER ENTYP,ERROR,GRPID
      CHARACTER*8 TYPE
C-----------------------------------------------------------------------
C   DIMENSION STATEMENTS
      INTEGER IDI(NID), IDO(NIDO)
      DOUBLE PRECISION DIMI(NDIMI), DIMO(NDIMO)
C=======================================================================
C=== ENTYP=1, FOR PBAR1; 2, FOR PBEAM1
      IF (ENTYP.EQ.1) THEN
         CALL MEVBRD(GRPID,TYPE,IDI,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO
     +               ,ERROR)
      ELSE IF (ENTYP.EQ.2) THEN
         CALL MEVBMD(GRPID,TYPE,IDI,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO
     +               ,ERROR)
      END IF
C-----------------------------------------------------------------------
      RETURN
      END
```

Listing 8-1. BSBRPD Subroutines

MEVBRD and MEVBMD Subroutines

MEVBRD and MEVBMD are the branched routines for the various sections, and convert the section dimensions to section properties for Bar and Beam elements. You may rename these routines as you like or move the function of these routines to BSBRPD. These routines call the BRXXPD routines where XX is the two-letter keyword for various section types. The MEVBRD routine for the standard library is given in Listing 8-2.

```
SUBROUTINE MEVBRD(GRPID,TYPE,ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,
     +                  NDIMO,ERROR)
C=======================================================================
C   Purpose
C           Call the default type subroutine to convert PBAR1 to PBAR
C
C   Arguments
C
C   GRPID input   int     ID of group
C   TYPE  input   char    Type of cross section
C   ID    input   int     Array of values PID, MID contained in PBAR1
C                         entries
C   NID   input   int     Size of ID array, NID=2 for PBAR1 entry
C   IDO   output  int     Array of integer values contained in PBAR
C                         entries
C   NIDO  output  int     Size of IDO array, NIDO=2 for PBAR entry
C   DIMI  input   flt     Dimension values of cross section
C   NDIMI input   int     Size of DIMI array
C   DIMO  output  flt     Properties of cross section
C   NDIMO output  flt     Size of DIMO array
C   ERROR output  int     Type of error
C
C   Method
C             Call the subroutine with respect to  the section type
C
C   Called by
C             BSBRPD
C
```

```
C   CALLS
C             BRRDPD,BRTUPD,BRBRPD,BRBXPD,BRIIPD,BRTTPD,BRLLPD,BRCHPD
C-----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION  (A-H,O-Z)
C     Calling  sequence arguments
      INTEGER          ERROR,GRPID,ID(NID),IDO(NIDO)
      CHARACTER*8      TYPE
      DOUBLE PRECISION DIMI(NDIMI),DIMO(NDIMO)
C=======================================================================
C     Clear the output array before usage
      CALL    ZEROD ( DIMO, NDIMO )
      CALL    ZEROI (  IDO, NIDO  )
      ERROR = 0
C
      IF    ( TYPE.EQ.'ROD     ') THEN
              CALL BRRDPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'TUBE    ') THEN
              CALL BRTUPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'BAR     ') THEN
              CALL BRBRPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'BOX     ') THEN
              CALL BRBXPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'I       ') THEN
              CALL BRIIPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'T       ') THEN
              CALL BRTTPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'L       ') THEN
              CALL BRLLPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'CHAN    ') THEN
              CALL BRCHPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'CROSS   ') THEN
              CALL BRCRPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'H       ') THEN
              CALL BRHHPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'T1      ') THEN
              CALL BRT1PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'I1      ') THEN
              CALL BRI1PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'CHAN1   ') THEN
              CALL BRC1PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'Z       ') THEN
              CALL BRZZPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'CHAN2   ') THEN
              CALL BRC2PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'T2      ') THEN
              CALL BRT2PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'BOX1    ') THEN
              CALL BRB1PD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'HEXA    ') THEN
              CALL BRHXPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSEIF( TYPE.EQ.'HAT     ') THEN
              CALL BRHTPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
      ELSE
              ERROR = 5150
      END IF
C-----------------------------------------------------------------------
      RETURN
      END
```

Listing 8-2. MEVBRD Subroutine

BRXXPD and BMXXPD Subroutines

The purpose of the BRXXPD and BMXXPD routines is to calculate the properties from the section dimensions. For each cross section, subroutines are required to convert the images of PBARL and PBEAML records to the images of PBAR and PBEAM records in the EPT datablock.

BRTUPD Subroutine

BRTUPD is an example routine that shows how to convert PBARL EPT record to PBAR EPT record for the Tube section. First, the details of the PBARL and the PBAR record are shown, and then the routine itself is given.

PBARL Record

The PBARL record in the EPT datablock is a derived from the PBARL Bulk Data entry and is given below.

Table 8-1. PBARL (9102, 91, 52)

Word	Name	Type	Description
1	PID	I	Property identification number.
2	MID	I	Material identification number.
3	Group	Char	Group Name.
4	Group	Char	Group Name.
5	TYPE	Char4	Cross-section Type.
6	TYPE	Char4	Cross-section Type.
7	Dim1	RS	Dimension1.
8	Dim2	RS	Dimension2.
n+7-1	Dim n	RS	Dimension n (note that the final dimension is the nonstructural mass).
n+7	Flag	I	–1. Flag indicating end of cross-section dimensions.

PBAR Record

The PBAR record in the EPT datablock is derived from the PBAR Bulk Data entry and consists of 19 words. It is a replica of the Bulk Data entry, starting with PID field. The word 8 in the record is set to 0.0 since the field 9 in the first line of the PBAR Bulk Data entry is not used. The details of the PBAR record are given in Table 8-2.

Table 8-2. PBAR (52, 20, 181)

Word	Name	Type	Description
1	PID	I	Property identification number.
2	MID	I	Material identification number.
3	A	RS	Area of cross-section.
4	I1	RS	Area moment of inertia for bending in plane 1.
5	I2	RS	Area moment of inertia for bending in plane 2.
6	J	RS	Torsional constant.

Table 8-2. PBAR (52, 20, 181)

Word	Name	Type	Description
7	NSM	RS	Nonstructural mass per unit length.
8	FE	RS	Not used. Set to 0.0.
9	C1	RS	Stress recovery location.
10	C2	RS	Stress recovery location.
11	D1	RS	Stress recovery location.
12	D2	RS	Stress recovery location.
13	E1	RS	Stress recovery location.
14	E2	RS	Stress recovery location.
15	F1	RS	Stress recovery location.
16	F2	RS	Stress recovery location.
17	K1	RS	Area factor of shear for plane 1.
18	K2	RS	Area factor of shear for plane 2.
19	I12	RS	Area product of inertia.

.

```
BRTUPD Subroutine
      SUBROUTINE BRTUPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
C=====================================================================
C   Purpose:
C      Convert PBAR1(entity type : TUBE) to PBAR
C
C   Arguments:
C
C   DIMI  input  flt Array of dimension values for cross-section
C                    (SEE FIG. 5 IN MEMO SSW-25, REV. 4, DATE 8/16/94)
C   NDIM  input  int Size of DIMI array
C   DIMO  output flt Array of property values for cross-section
C   NDIMO output int Size of DIMO array
C   ERROR output int Type of error
C
C   DISCRIPTION FOR DIMO ARRAY:
C       DIMO (1)  = A
C       DIMO (2)  = I1
C       DIMO (3)  = I2
C       DIMO (4)  = J
C       DIMO (5)  = NSM
C       DIMO (6)  = FE
C       DIMO (7)  = C1
C       DIMO (8)  = C2
C       DIMO (9)  = D1
C       DIMO (10) = D2
C       DIMO (11) = E1
C       DIMO (12) = E2
C       DIMO (13) = F1
C       DIMO (14) = F2
C       DIMO (15) = K1
C       DIMO (16) = K2
C       DIMO (17) = I12
C
C   Method
C
C   Called by:
C             MEVBRD
```

```
C-----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION  (A-H,O-Z)
C     Calling sequence arguments
      INTEGER          ID(NID),IDO(NIDO),ERROR
      DOUBLE PRECISION DIMI(NDIMI),DIMO(NDIMO)
C=======================================================================
C
C=== WRITE THE PART OF INTEGER
      DO 30 II = 1,NID
         IDO(II) = ID(II)
 30   CONTINUE
C
      DIM1 = DIMI(1)
      DIM2 = DIMI(2)
      DIMO(1) = PI*(DIM1*DIM1-DIM2*DIM2)
      DIMO(2) = PI*(DIM1**4-DIM2**4)/4.D0
      DIMO(3) = DIMO(2)
      DIMO(4) = PI*(DIM1**4-DIM2**4)/2.D0
      DIMO(5) = DIMI(3)
      DIMO(6) = 0.D0
      DIMO(7) = DIM1
      DIMO(8) = 0.D0
      DIMO(9) = 0.D0
      DIMO(10) = DIM1
      DIMO(11) = -DIM1
      DIMO(12) = 0.D0
      DIMO(13) = 0.D0
      DIMO(14) = -DIM1
      DIMO(15) = 0.5D0
      DIMO(16) = 0.5D0
      DIMO(17) = 0.D0
      IF ( DIMI(1).LE.DIMI(2) ) ERROR = 5102
C-----------------------------------------------------------------------
      RETURN
      END
```

Listing 8-3. BRTUPD Subroutine

BMTUPD Subroutine

BMTUPD is an example routine that shows how to convert PBEAML EPT record to PBEAM EPT record for the Tube section. First, the details of the PBEAML and the PBEAM records are shown, and then the routine itself is given.

PBEAML Record

The PBEAML record in the EPT datablock is a derived from the PBEAML Bulk Data entry and is given in Table 8-3.

Table 8-3. PBEAML (9202, 92, 53)

Word	Name	Type	Description
1	PID	I	Property ID.
2	MID	I	Material ID.
3	Group	Char	Group Name.
4	Group	Char	Group Name.
5	TYPE	Char4	Cross-section Type.

Table 8-3. PBEAML (9202, 92, 53)

Word	Name	Type	Description
6	TYPE	Char4	Cross-section Type.
7	SO	RS	Stress output request flag, 1=yes,=no.
8	XXB	RS	X/XB - parametric location of the station.
9	Dim1	RS	Dimension 1.
10	Dim2	RS	Dimension 2.
n+9-1	Dim n	RS	Dimension n (note that the final dimension is the nonstructural mass).
n+9	Flag	I	–1. Flag indicating end of cross-section dimensions.
Words 7 through ndim+11 repeat 11 times.			

PBEAM Record

The PBEAM record in EPT datablock consists of 197 words. The first five words and the last 16 words are common to all the 11 stations. Each of the 11 stations have their own 21 unique words. The details of the PBEAM record are given in Table 8-4.

Table 8-4. PBEAM (5402, 54, 262)

Word	Name	Type	Description
1	PID	I	Property identification number.
2	MID	I	Material identification number.
3	N	I	Number of intermediate stations.
4	CCF	I	Constant cross-section flag. 1 = constant, 2 = variable.
5	X	RS	Unused.
6	SO	RS	Stress output request. 1.0 = yes, 0.0 = no.
7	XXB	RS	Parametric location of the station. Varies between 0. and 1.0.
8	A	RS	Area.
9	I1	RS	Moment of inertia for bending in plane 1.
10	I2	RS	Moment of inertia for bending in plane 2.
11	I12	RS	Area product of inertia.
12	J	RS	Torsional constant.
13	NSM	RS	Nonstructural mass.
14	C1	RS	Stress recovery location.
15	C2	RS	Stress recovery location.
16	D1	RS	Stress recovery location.
17	D2	RS	Stress recovery location.
18	E1	RS	Stress recovery location.
19	E2	RS	Stress recovery location.
20	F1	RS	Stress recovery location.
21	F2	RS	Stress recovery location.
Words 6 through 21 repeat 11 times.			
182	K1	RS	Area factor for shear for plane 1.

Table 8-4. PBEAM (5402, 54, 262)

Word	Name	Type	Description
183	K2	RS	Area factor for shear for plane 2.
184	S1	RS	Shear-relief coefficient for plane 1.
185	S2	RS	Shear-relief coefficient for plane 2.
186	NSIA	RS	Nonstructural mass moment of inertia at end A.
187	NSIB	RS	Nonstructural mass moment of inertia at end B.
188	CWA	RS	Warping coefficient for end A.
189	CWB	RS	Warping coefficient for end B.
190	M1A	RS	Y-coordinate of center of gravity for nonstructural mass at end A.
191	M2A	RS	Z-coordinate of center of gravity for nonstructural mass at end A.
192	M1B	RS	Y-coordinate of center of gravity for nonstructural mass at end B.
193	M2B	RS	Z-coordinate of center of gravity for nonstructural mass at end B.
194	N1A	RS	Y-coordinate for neutral axis at end A.
195	N2A	RS	Z-coordinate for neutral axis at end A.
196	N1B	RS	Y-coordinate for neutral axis at end B.
197	N2B	RS	Z-coordinate for neutral axis at end B.

```
SUBROUTINE BMTUPD(ID,NID,IDO,NIDO,DIMI,NDIMI,DIMO,NDIMO,ERROR)
C=======================================================================
C   Purpose
C     Convert PBEAM1(entity type : TUBE) to PBEAM
C
C   Arguments
C
C   ID    input  int  Contain the integer information PID, MID
C   NID   input  int  Size of ID array, NID = 2
C   DIMI  input  flt  Dimension values of cross section
C                     ( See  FIG.5 IN MEMO SSW-25, REV. 4, DATE 8/16/94)
C   NDIMI input  int  Size of DIMI array
C   IDO   output int  Contain the integer information PID,MID,N,CCF
C   NIDO  output int  Size of IDO array, NIDO = 4
C   DIMO  output flt  Properties of cross section
C   NDIMO output int  Size of DIMO array
C   ERROR output int  Type of error
C
C   Description for DIMO array
C       DIMO (1)   = X
C       DIMO (2)   = SO
C       DIMO (3)   = XXB
C       DIMO (4)   = A
C       DIMO (5)   = I1
C       DIMO (6)   = I2
C       DIMO (7)   = I12
C       DIMO (8)   = J
C       DIMO (9)   = NSM
C       DIMO (10)  = C1
C       DIMO (11)  = C2
C       DIMO (12)  = D1
C       DIMO (13)  = D2
C       DIMO (14)  = E1
```

```
C       DIMO (15)  = E2
C       DIMO (16)  = F1
C       DIMO (17)  = F2
C    DIMO(2) thru DIMO(17) repeat 11 times
C       DIMO (178) = K1
C       DIMO (179) = K2
C       DIMO (180) = S1
C       DIMO (181) = S2
C       DIMO (182) = NSIA
C       DIMO (183) = NSIB
C       DIMO (184) = CWA
C       DIMO (185) = CWB
C       DIMO (186) = M1A
C       DIMO (187) = M2A
C       DIMO (188) = M1B
C       DIMO (189) = M2B
C       DIMO (190) = N1A
C       DIMO (191) = N2A
C       DIMO (192) = N1B
C       DIMO (193) = N2B
C
C   Method
C               Simply calculate the properties and locate that data
C               to the image of PBEAM entries
C   Called by
C               MEVBMD
C-----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION  (A-H,O-Z)
C     Calling  sequence arguments
      INTEGER          ERROR,ID(NID),IDO(NIDO)
      DOUBLE PRECISION DIMI(NDIMI),DIMO(NDIMO)
C     Local variables
      INTEGER NAM(2)
C     NASTRAN common blocks
      COMMON /CONDAD/PI
C     Local data
      DATA NAM/4HBMTU,4HPD  /
C=======================================================================
C
C=== WRITE THE PART OF INTEGER
      DO 30 II = 1,NID
         IDO(II) = ID(II)
 30   CONTINUE
C-----------------------------------------------------------------------
C=== DETECT HOW MANY STATION , CONSTANT OR LINEAR BEAM.
C-----------------------------------------------------------------------
      ISTATC = NDIMI/11
      DO 35 II = 0,10
         NW = II*ISTATC
         IF (DIMI(3+NW).EQ.0.D0) THEN
            IDO(3) = II-2
            IDO(4) = 1
            IF (DIMI(3).NE.DIMI(3+NW-ISTATC)) IDO(4)=2
            GO TO 40
         END IF
 35   CONTINUE
 40   DIMO(1) = 0.D0
      DO 100 L1 = 0,10
         LC = 16*L1
         NW = L1*ISTATC
         IF (DIMI(3+NW+ISTATC).EQ.0.D0) LC = 160
         DIM1 = DIMI(3+NW)
         DIM2 = DIMI(4+NW)
         IF ( DIM1.LE.DIM2 ) ERROR = 5102
         DIMO(2+LC) = DIMI(1+NW)
```

```
         DIMO(3+LC) = DIMI(2+NW)
         DIMO(4+LC) = PI*(DIM1*DIM1-DIM2*DIM2)
         DIMO(5+LC) = PI*(DIM1**4-DIM2**4)/4.D0
         DIMO(6+LC) = DIMO(5+LC)
         DIMO(7+LC) = 0.D0
         DIMO(8+LC) = PI*(DIM1**4-DIM2**4)/2.D0
         DIMO(9+LC) = DIMI(5+NW)
         DIMO(10+LC) = DIM1
         DIMO(11+LC) = 0.D0
         DIMO(12+LC) = 0.D0
         DIMO(13+LC) = DIM1
         DIMO(14+LC) = -DIM1
         DIMO(15+LC) = 0.D0
         DIMO(16+LC) = 0.D0
         DIMO(17+LC) = -DIM1
         IF (LC.EQ.160) GO TO 110
 100  CONTINUE
 110  DIMO(178) = 0.5D0
      DIMO(179) = 0.5D0
C-----------------------------------------------------------------------
 300  RETURN
      END
```

Listing 8-4. BMTUPD Subroutine

BSGRQ Subroutine

For optimization of PBARL entries, you need to provide overall information such as number of cross sections in your library and the maximum number of fields in the continuation lines. BSGRQ provides the information and is required only if you wish to perform sensitivity or optimization with the section dimensions. The calling sequence and example routine is given in Listing 8-5.

```
SUBROUTINE BSGRQ(GRPID,NSECT,NDIMAX,ERROR)
C
C =====================================================================
C   PURPOSE:
C     PROVIDE OVERALL CHARACTERISTICS OF A BEAM SECTION LIBRARY
C ---------------------------------------------------------------------
C   ARGUMENTS:
C
C      GRPID  input  integer  Group name.  Not used, reserved for future
C                             use.
C      NSECT  output integer  Number of different section types
C      NDIMAX output integer  Maximum number of dimension for any
C                             section type
C      ERROR  output integer  Indicates if an error has occurred. The
C                             code returned indicates the type of error
C ---------------------------------------------------------------------
C   CALLED BY:
C      BCGRQ routine
C---------------------------------------------------------------------
C
C===  Set number of section in the library
         nsect  = 19
C===  Set the maximum number of DIMi fields (including 1 for nonstructured mass) required by any one
         ndimax = 7
C---------------------------------------------------------------------
      RETURN
      END
```

Listing 8-5. BSGRQ Routine

Note that the arguments GRPID and ERROR are not used. GRPID is reserved for future use. You may use the ERROR argument to send an error code which could later be used to print an error message.

BSBRT Subroutine

The BSBRT routine provides the name, number of fields in the continuation line in the PBARL Bulk Data entry, and the number of constraints for each section in the library. As an example, the name of the tube section, shown in Figure 8-11, in the standard library is “TUBE”, the number of dimensions for the tube section is three (OUTER RADIUS, INNER RADIUS, and NSM), and there is one physical constraint.

The physical constraint is that the inner radius (DIM2) can not be greater than outer radius (DIM1). It is necessary to specify the constraints so that the optimization of the section dimension in SOL200 does not result into an inconsistent shape.

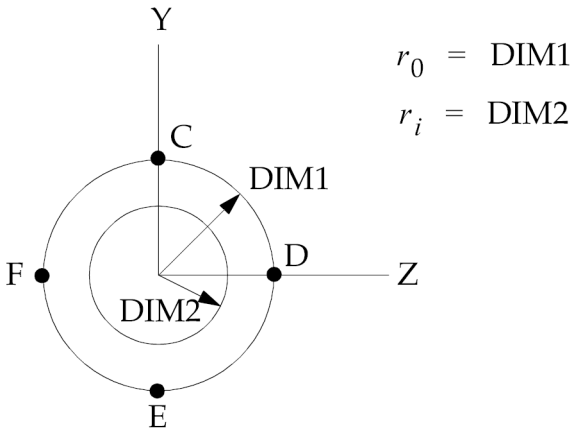


Figure 8-11. Tube Section

This routine is called during optimization process. The calling sequence and example routine is given in Listing 8-6.

```
BSBRT Subroutine
      SUBROUTINE BSBRT(GRPID,ENTYP,TYPE,NDIM,NCONST,NSECT,ERROR)
C ======================================================================
C   Purpose
C          Get the name, number of  fields iin the continuation line and
C          number of constraints
C          to be used by optimization routines
C
C   Arguments
C
C       GRPID  input  integer      The ID of group name
C       ENTYP  input  integer      1: PBARL, 2: PBEAML
C       TYPE   output character*8  Arrays for cross section types
C       NDIM   output integer      Number of dimensions for isect
C                                  section type, including 1 for
                                   nonstructural mass
C       NCONST output integer      Number of dimensional constraints
C                                  imposed by isect section type
```

```
C       NSECT  input  integer      Number of sections
C       ERROR  output integer      Error code
C
C   Called by
C             BCBRT routine
C -------------------------------------------------------------------------
C===  Argument Type Declaration
      INTEGER      GRPID,ENTYP,NDIM(NSECT),NCONST(NSECT)
      CHARACTER*8  TYPE(NSECT)
C===  Local variables
      INTEGER NAM(2)
C =========================================================================
C===  Currently, only PBARL is supported for optimization. Based on
C===  ENTYP the library may have different number of DIMi fields and
C===  number of constraints. Currently, GRPID and ENTYP are not being
C===  used. So just set them to default values, even though they are
C===  input type.
     GRPID = 1
      ENTYP = 1
C -------------------------------------------------------------------------
C===  Set the name, number of fields in the continuation line for the PBARL entry,
C===  and number of constraints in the TYPE, DIMI and NCONST arrays,
      respectively .
C===   Note that the value in DIMI is one more (for NSM field) than DIMi fields
C===   Make sure names are all capitals, even if they are lower case in the
       input data file.
      TYPE(1) = 'ROD'
      NDIM(1) = 2
      NCONST(1) = 0
C
      TYPE(2) = 'TUBE'
      NDIM(2) = 3
      NCONST(2) = 1
C
      TYPE(3) = 'BAR'
      NDIM(3) = 3
      NCONST(3) = 0
C
      TYPE(4) = 'BOX'
      NDIM(4) = 5
      NCONST(4) = 2
C
      TYPE(5) = 'I'
      NDIM(5) = 7
      NCONST(5) = 3
C
      TYPE(6) = 'T'
      NDIM(6) = 5
      NCONST(6) = 2
C
      TYPE(7) = 'L'
      NDIM(7) = 5
      NCONST(7) = 2
C
      TYPE(8) = 'CHAN'
      NDIM(8) = 5
      NCONST(8) = 2
C
      TYPE(9) = 'CROSS'
      NDIM(9) = 5
      NCONST(9) = 1
C
      TYPE(10) = 'H'
      NDIM(10) = 5
      NCONST(10) = 1
C
```

```
      TYPE(11) = 'T1'
      NDIM(11) = 5
      NCONST(11) = 1
C
      TYPE(12) = 'I1'
      NDIM(12) = 5
      NCONST(12) = 1
C
      TYPE(13) = 'CHAN1'
      NDIM(13) = 5
      NCONST(13) = 1
C
      TYPE(14) = 'Z'
      NDIM(14) = 5
      NCONST(14) = 1
C
      TYPE(15) = 'CHAN2'
      NDIM(15) = 5
      NCONST(15) = 2
C
      TYPE(16) = 'T2'
      NDIM(16) = 5
      NCONST(16) = 2
C
      TYPE(17) = 'BOX1'
      NDIM(17) = 7
      NCONST(17) = 2
C
      TYPE(18) = 'HEXA'
      NDIM(18) = 4
      NCONST(18) = 1
C
      TYPE(19) = 'HAT'
      NDIM(19) = 5
      NCONST(19) = 2
C
C =====================================================================
      RETURN
      END
```

Listing 8-6. BSBRT Subroutine

BSBRID Subroutine

The BSBRID subroutine is required if optimization is to be performed. Its function is to provide information required in the calculation of the sensitivities (gradients) of the bar properties with respect to the bar dimensions.

Two basic types of information are provided. The first is the SENTYP array, which indicates how each section property varies as a function of each dimension. Values in the SENTYP array can be either: 0 for no variation; 1 for a linear variation; 2 for a nonlinear variation; or 3 for an unknown variation. The SENTYP = 3 option is to be used when you know that the property is a function of the design dimension, but analytical gradient information is not being provided using the BSBRGD subroutine. In this case, NX Nastran will calculate the gradients for you using central differencing techniques.

The second piece of information is the ALIN array. This array provides any linear sensitivity data. For example, the C1 stress recovery location for the TUBE section is · DIM1 so that this sensitivity of this stress recovery point with respect to the first dimension is 1.0.

You may use your own naming convention for the subroutines that specify the section sensitivity data. The calling tree used for the standard library is shown in Figure 8-12.

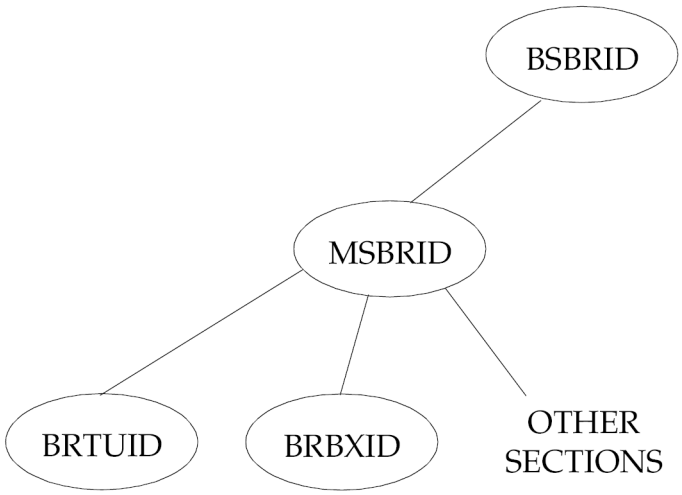


Figure 8-12. Calling Tree to Generate Sensitivity Data

BSBRID calls the bar evaluator routine MSBRID/S for the BAR element while the corresponding beam evaluator routine is absent since the PBEAM1 dimensions cannot be designed. The evaluator in turn calls the routines for each section. The routines are named BRXXID, where XX is a two-letter identifier for the section. For example, the routine for the TUBE section is called BRTUID. The details for various routines are given in Listing 8-7.

```
SUBROUTINE BSBRID(GRPID,ENTYP,SECTON,SENTYP,ALIN,NDIM,NPROP,
     1                  ERROR)
C
C ======================================================================
C   Purpose
C     set up section dependent information for a particular cross
C     section type
C
C   Arguments
C
C       GRPID  input  integer      ID of the group
C       ENTYP  input  integer      1: PBAR1, 2: PBEAM1
C       SECTON input  character*8  Section type
C       SENTYP output integer      Type of sensitivity, 0: invariant,
C                                  1: linear, 2: nonlinear, 3: calculated by
C                                     finite difference
C       ALIN   output double       Matrix providing the linear
C                                  factors for sensitive relationships
C       NDIM   input  integer      Number of dimensions
C       NPROP  input  integer      Number of properties
C       ERROR  output integer      Type of error
C
C   Method
C               Simply transfer control based on entry type
C
C   Called by
C               BCBRID
C
C   Calls
C               MSBRID, MSBMID
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
```

```
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      INTEGER          ENTYP,ERROR,GRPID,SENTYP(NPROP,NDIM)
      CHARACTER*8      SECTON
      DOUBLE PRECISION ALIN(NPROP,NDIM)
C ======================================================================
      GRPID = 1
      IF(ENTYP.EQ.1) THEN
        CALL MSBRID(SECTON,SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE
         ERROR = 5400
      END IF
C
      RETURN
      END
```

Listing 8-7. BSBRID Subroutine

MSBRID Subroutine

MSBRID is a branched routine for providing information on the calculation of sensitivities for each of the bar types. You may rename this routine as you like or move its function to BSBRID. MSBRID calls the BRXXID routines, where XX is the two-letter keyword for various section types. The routine MSBRID for the standard library is given in Listing 8-8.

```
MSBRID Subroutine
      SUBROUTINE MSBRID(SECTON,SENTYP,ALIN,NDIM,NPROP,ERROR)
C
C ======================================================================
C   Purpose
C     To set up section dependent information for PBAR1 cross section
C
C   Arguments
C
C       SECTON input  character*8 Name of section type
C       SENTYP output integer     Type of sensitivity, 0: invariant,
C                                 1: linear, 2: nonlinear
C       ALIN   output double      Matrix providing the linear factors
C                                 for sensitive relationships
C       NDIM   input  integer     No. of dimensions
C       NPROP  input  integer     No. of properties in EPT datablock
C       ERROR  output integer     Type of error
C
C    Method
C               Simply transfer the section dependent information of
C               the 19 kinds
C    Called by
C               BSBRID
C
C    Calls
C               BRRDID, BRBRID, BRTUID, BRBXID, BRIIID, BRLLID, BRTTID,
C               BRCHID, BRCRID, BRHHID, BRT1ID, BRI1ID, BRC1ID, BRZZID,
C               BRC2ID, BRT2ID, BRB1ID, BRHXID, BRHTID
C               ZEROI, ZEROD (Nastran utility)
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      CHARACTER*8      SECTON
      INTEGER          ERROR,SENTYP(NPROP,NDIM)
```

```
      DOUBLE PRECISION ALIN(NPROP,NDIM)
C ======================================================================
C
      CALL ZEROI( SENTYP(1,1), NPROP*NDIM )
      CALL ZEROD( ALIN(1,1), NPROP*NDIM )
      ERROR = 0
C
      IF(SECTON.EQ.'ROD') THEN
         CALL BRRDID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'BAR') THEN
         CALL BRBRID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'TUBE') THEN
         CALL BRTUID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'BOX') THEN
         CALL BRBXID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'I') THEN
         CALL BRIIID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'L') THEN
         CALL BRLLID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'T') THEN
         CALL BRTTID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'CHAN') THEN
         CALL BRCHID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'CROSS') THEN
         CALL BRCRID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'H') THEN
         CALL BRHHID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'T1') THEN
         CALL BRT1ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'I1') THEN
         CALL BRI1ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'CHAN1') THEN
         CALL BRC1ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'Z') THEN
         CALL BRZZID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'CHAN2') THEN
         CALL BRC2ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'T2') THEN
         CALL BRT2ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'BOX1') THEN
         CALL BRB1ID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'HEXA') THEN
         CALL BRHXID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE IF(SECTON.EQ.'HAT') THEN
         CALL BRHTID(SENTYP,ALIN,NDIM,NPROP,ERROR)
      ELSE
         ERROR = 5410
      END IF
C
      RETURN
      END
```

Listing 8-8. MSBRID Subroutine

BRTUID Subroutine

BRTUID is an example routine that shows how to define the sensitivity type of each of the bar properties for the tube and the subset of the sensitivities that are linear.

```
SUBROUTINE BRTUID(SENTYP,ALIN,NDIM,NPROP,ERROR)
C
C ======================================================================
C   Purpose
C     To set up section dependent information for rod cross section
C
C   Arguments
C
C       SENTYP output integer  Type of sensitivity, 0: invariant,
C                              1: linear, 2: nonlinear
C       ALIN   output double   Matrix providing the linear factors for
C                              sensitive relationships
C       NDIM   input  integer  No. of dimensions
C       NPROP  input  integer  No. of properties in EPT datablock
C       ERROR  output integer  Type of error
C
C   Method
C               Simply provides the information
C
C   Called by
C               MSBRID
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      INTEGER          ERROR,SENTYP(NPROP,NDIM)
      DOUBLE PRECISION ALIN(NPROP,NDIM)
C     Local variables
      INTEGER NAM(2)
C     Local data
      DATA NAM/4HBRTU,4HID  /
C ======================================================================
C
      ALIN( 7,1) =  1.0D0
      ALIN(10,1) =  1.0D0
      ALIN(11,1) = -1.0D0
      ALIN(14,1) = -1.0D0
      ALIN( 5,3) =  1.0D0
C
      SENTYP( 1,1) = 2
      SENTYP( 1,2) = 2
      SENTYP( 2,1) = 2
      SENTYP( 2,2) = 2
      SENTYP( 3,1) = 2
      SENTYP( 3,2) = 2
      SENTYP( 4,1) = 2
      SENTYP( 4,2) = 2
      SENTYP( 7,1) = 1
      SENTYP(10,1) = 1
      SENTYP(11,1) = 1
      SENTYP(14,1) = 1
      SENTYP( 5,3) = 1
C
      RETURN
      END
```

Listing 8-9. BRTUID Subroutine

BSBRGD Subroutine

The BSBRGD subroutine is required if optimization is to be performed and analytical sensitivities are needed (SENTYP = 2 in subroutine BSBRID). Its function is to provide the nonlinear gradients of the bar properties with respect to the bar dimensions. You may use your own naming convention for

the subroutines that calculate the section gradients from the dimensions. The calling tree used for the standard library is shown in Figure 8-13.

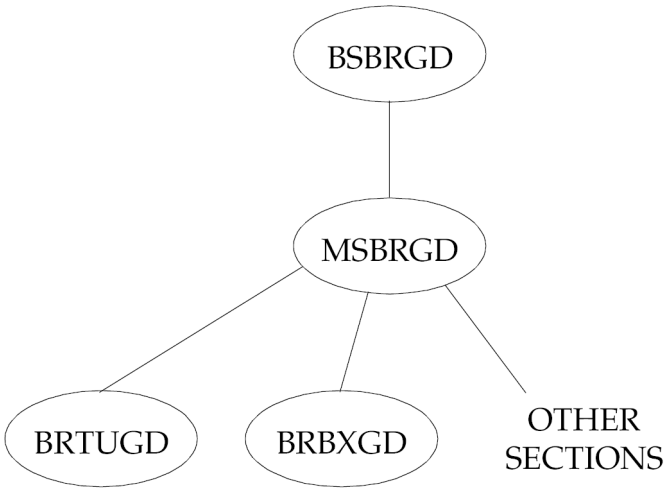


Figure 8-13. Calling Tree to Generate Nonlinear Sensitivity Data

BSBRGD calls the bar evaluator routine MSBRGD/S for the BAR element. The evaluator, in turn, calls the routines for each section. The routines are named BRXXGD, where XX is a two-letter identifier for the section. For example, the routine for the TUBE section is called BRTUGD. The details for various routines are given in Listing 8-10.

```
UBROUTINE BSBRGD(GRPID,ENTYP,TYPE,DIMI,NDIMI,ANONL,NPROP,ERROR)
C
C ======================================================================
C   Purpose
C     To get the nonlinear factors of sensitivities for default sections
C
C   Arguments
C
C       GRPID input  integer      ID of group name
C       ENTYP input  integer      1: PBAR1, 2: PBEAM1
C       TYPE  input  character*8  Type name of cross-section
C       DIMI  input  double       Array from EPT record
C       NDIMI input  integer      Number of dimensions (Plus NSM)
C       ANONL output double       Array providing the nonlinear factors
C                                 for sensitivity relationships
C       NPROP input  integer      Number of properties in PBAR entries
C       ERROR output integer      Type of error
C
C   Method
C             Simply transfer control based on entry types
C   Called by
C             BCBRGD
C
C   Calls
C             MSBRGD
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
```

```
      INTEGER          GRPID,ENTYP,NDIMI,NPROP,ERROR
      CHARACTER*8      TYPE
      DOUBLE PRECISION ANONL(NPROP,NDIMI), DIMI(NDIMI)
C     Local variables
      INTEGER NAM(2)
C     Local data
      DATA NAM/4HBSBR,4HGD  /
C =====================================================================
      GRPID = 1
      IF(ENTYP .EQ. 1) THEN
         CALL MSBRGD(TYPE,DIMI,NDIMI,ANONL,NPROP,ERROR)
     ELSE
         ERROR = 5300
      END IF
C ---------------------------------------------------------------------
      RETURN
      END
```

Listing 8-10. BSBRGD Subroutine

MSBRGD Subroutine

MSBRGD is a branched routine for providing information on the calculation of nonlinear gradients for each of the bar types. You may rename this routine as you like or move its function to BSBRGD. MSBRGD calls the BRXXGD routines, where XX is the two-letter keyword for various section types. The routine MSBRGD for the standard library is given in Listing 8-11.

```
SUBROUTINE MSBRGD(TYPE,DIMI,NDIMI,ANONL,NPROP,ERROR)
C
C =====================================================================
C   Purpose
C     To get the nonlinear factors of sensitivities for PBAR1 entries
C
C   Arguments
C
C       TYPE  input  character*8  Type name of cross-section
C       DIMI  input  double       Array from EPT record
C       NDIMI input  integer      Number of dimensions (Plus NSM)
C       ANONL output double       Array providing the nonlinear factors
C                                 for sensitivity relationships
C       NPROP input  integer      Number of properties in PBAR entries
C       ERROR output integer      Type of error
C
C   Method
C             Simply transfer information based on cross-section type
C   Called by
C             BSBRGD
C
C   Calls
C             BRRDGD,BRBRGD,BRBXGD,BRTUGD,BRIIGD,BRTTGD,BRLLGD,BRCHGD
C             BRCRGD,BRHHGD,BRT1GD,BRT2GD,BRI1GD,BRC1GD,BRC2GD,BRZZGD
C             BRHXGD,BRB1GD,BRHTGD,ZEROD(Nastran utility)
C ---------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      INTEGER          NDIMI,NPROP,ERROR
      CHARACTER*8      TYPE
      DOUBLE PRECISION ANONL(NPROP,NDIMI), DIMI(NDIMI)
C =====================================================================
      CALL ZEROD( ANONL(1,1), NPROP*NDIMI )
      ERROR = 0
```

```
      IF(TYPE.EQ.'ROD')  THEN
            CALL  BRRDGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'BAR')  THEN
             CALL BRBRGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'BOX')  THEN
             CALL BRBXGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'TUBE')  THEN
             CALL BRTUGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'I')  THEN
             CALL BRIIGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'L')  THEN
             CALL BRLLGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'T')  THEN
             CALL BRTTGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'CHAN')  THEN
             CALL BRCHGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'CROSS') THEN
             CALL BRCRGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'H') THEN
             CALL BRHHGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'T1') THEN
             CALL BRT1GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'I1') THEN
             CALL BRI1GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'CHAN1') THEN
             CALL BRC1GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'Z') THEN
             CALL BRZZGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'CHAN2') THEN
             CALL BRC2GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'T2') THEN
             CALL BRT2GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'BOX1') THEN
             CALL BRB1GD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'HEXA') THEN
             CALL BRHXGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE IF(TYPE.EQ.'HAT') THEN
             CALL BRHTGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
      ELSE
         ERROR = 5310
      ENDIF
C
      RETURN
      END
```

Listing 8-11. MSBRGD Subroutine

BRTUGD Subroutine

BRTUGD is an example routine that shows how to define the nonlinear gradients of the TUBE section as a function of the outer and inner radius of the tube.

```
SUBROUTINE BRTUGD(DIMI,NDIMI,ANONL,NPROP,ERROR)
C
C ======================================================================
C   Purpose
C     To get the nonlinear factors of sensitivities for TUBE section
C
C   Arguments
C
C       DIMI  input  double       Array of EPT records (Dimi+NSM)
C       NDIMI input  integer      Number of dimensions (Plus NSM)
C       ANONL output double       Array providing the nonlinear factors
C                                 for sensitivity relationships
C       NPROP input  integer      Number of properties in PBAR entries
C       ERROR output integer      Type of error
C
C   Method
C             Simply calculates the nonlinear factors
C   Called by
C             MSBRGD
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      INTEGER          NDIMI,NPROP,ERROR
      DOUBLE PRECISION ANONL(NPROP,NDIMI), DIMI(NDIMI)
C     Nastran common blocks
      COMMON /CONDAD/ PI
C ======================================================================
      DIM1       = DIMI(1)
      DIM2       = DIMI(2)
      PDIM13     = PI*DIM1**3
      PDIM23     = PI*DIM2**3
C
      ANONL(1,1) = 2*PI*DIM1
      ANONL(1,2) = -2*PI*DIM2
      ANONL(2,1) = PDIM13
      ANONL(2,2) = -PDIM23
      ANONL(3,1) = PDIM13
      ANONL(3,2) = -PDIM23
      ANONL(4,1) = 2*PDIM13
      ANONL(4,2) = -2*PDIM23
C
      RETURN
      END
```

Listing 8-12. BRTUGD Subroutine

BSBRCD Subroutine

The BSBRCD subroutine allows you to place constraints on values the beam dimensions can take during a design task. It is not needed unless optimization is used and, even then, is available only to impose conditions on the dimensions to keep the optimization process from selecting physically meaningless dimensions.

For example, the optimizer might select a TUBE design with the inner radius greater than the outer radius because this allows for a negative area and therefore a negative weight (something a weight minimization algorithm loves!) These constraints are not the same as the PMIN and PMAX property limits that are imposed on the DVPREL1 entry. Instead, these are constraints that occur between or among section dimensions. A DRESP2 entry could be used to develop the same design constraints, but the subroutine reduces the burden on the user interface, the primary goal of the beam library project. The calling tree used for the standard library, which is shown in Figure 8-14.

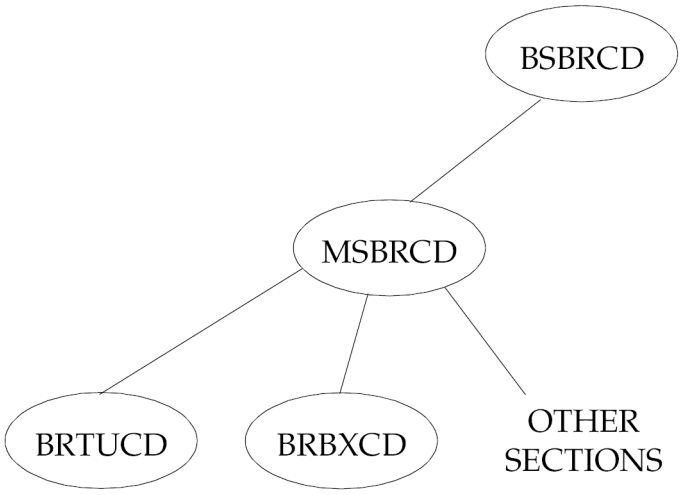


Figure 8-14. Calling Tree to Generate Beam Dimension Constraints

BSBRCD calls the bar evaluator routine MSBRCD. The evaluator in turn calls the routines for each section that requires constraints. The routines are named BRXXCD, where XX is a two-letter identifier for the section. For example the routine for the TUBE section is called BRTUCD. The details for various routines are given in Listing 8-13.

```
SUBROUTINE BSBRCD(GRPID,ENTYP,TYPE,AFACT,NCONST,NDIMI,ERROR)
C
C =====================================================================
C   Purpose
C           To get constraint information for default types
C
C   Arguments
C
C       GRPID  input integer     ID of the group name
C       ENTYP  input integer     1: PBAR1, 2: PBEAM1
C       TYPE   input character*8 Section type
C       NCONST input integer     Number of constraints
C                                for the section type
C       NDIMI  input integer     Number of dimensions
C                                for the section type
C       AFACT  output double     The factor for the NDIMI dimension in
C                                the constraint relation. Dimensions are
C                                NCONST by NDIMI.
C       ERROR  output integer    type of error
C
C   Method
C              Simply transfers control based on PBAR1 or PBEAM1 entries
C
C   Called by
C              BCBRCD
C
C   Calls
C              MSBRCD
C ---------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      CHARACTER*8          TYPE
      INTEGER              GRPID,ENTYP,NCONST,NDIMI,ERROR
      DOUBLE PRECISION     AFACT(NCONST,NDIMI)
C =====================================================================
```

```
C
      GRPID = 1
      IF(ENTYP.EQ.1) THEN
         CALL MSBRCD(TYPE,AFACT,NCONST,NDIMI,ERROR)
      END IF
C -----------------------------------------------------------------------
      RETURN
      END
```

Listing 8-13. BSBRCD Subroutine

MSBRCD Subroutine

MSBRCD is a branched routine for providing information on the calculation of gradients for each of the bar types. You may rename this routine as you like or move its function to BSBRCD. MSBRCD calls the BRXXCD routines, where XX is the two-letter keyword for various section types. The routine MSBRCD for the standard library is given in Listing 8-14.

```
SUBROUTINE BSBRCD(GRPID,ENTYP,TYPE,AFACT,NCONST,NDIMI,ERROR)
C
C =======================================================================
C   Purpose
C            To get constraint information for default types
C
C   Arguments
C
C       GRPID  input integer     ID of the group name
C       ENTYP  input integer     1: PBAR1, 2: PBEAM1
C       TYPE   input character*8 Section type
C       NCONST input integer     Number of constraints
C                                for the section type
C       NDIMI  input integer     Number of dimensions
C                                for the section type
C       AFACT  output double     The factor for the NDIMI dimension in
C                                the constraint relation. Dimensions are
C                                NCONST by NDIMI.
C       ERROR  output integer    type of error
C
C   Method
C              Simply transfers control based on PBAR1 or PBEAM1 entries
C
C   Called by
C              BCBRCD
C
C   Calls
C              MSBRCD
C -----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      CHARACTER*8          TYPE
      INTEGER              GRPID,ENTYP,NCONST,NDIMI,ERROR
      DOUBLE PRECISION     AFACT(NCONST,NDIMI)
C =======================================================================
C
      GRPID = 1
      IF(ENTYP.EQ.1) THEN
         CALL MSBRCD(TYPE,AFACT,NCONST,NDIMI,ERROR)
      END IF
C -----------------------------------------------------------------------
```

```
      RETURN
      END
```

Listing 8-14. MSBRCD Subroutine

BRTUCD Subroutine

BRTUCD is an example routine that shows how to define the constraints for a bar section. This example routine is for the TUBE section and imposes a single constraint that −DIM1 + DIM2 <0.0, where DIM1 is the outer radius and DIM2 is the inner radius of the tube. The constraints should always be specified so that the specified linear combination of dimensions is less or equal to zero when the constraint is satisfied.

```
SUBROUTINE BRTUCD(AFACT,NCONST,NDIMI,ERROR)
C
C ======================================================================
C   Purpose
C           To get constraint information for TUBE type
C
C   Arguments
C
C     AFACT   output double   The factor for the NDIMI dimension in the
C                             constraint relation. Dimensions are NCONST
C                             by NDIMI.
C     NCONST  input  integer  Number of constraints for the section type
C     NDIMI   input  integer  Number of dimensions for the section type
C     ERROR   output integer  type of error
C
C   Method
C              Simply transfers constraint information
C
C   Called by
C             MSBRCD
C ----------------------------------------------------------------------
      IMPLICIT INTEGER (I-N)
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C     Calling sequence arguments
      INTEGER              NCONST,NDIMI,ERROR
      DOUBLE PRECISION     AFACT(NCONST,NDIMI)
C ======================================================================
C
      IF(NCONST.NE.1) THEN
         ERROR = 5502
         RETURN
      END IF
C
      AFACT(1,1) = -1.0D0
      AFACT(1,2) =  1.0D0
C
      RETURN
      END
```

Listing 8-15. BRTUCD Subroutine

BSMSG Subroutine

The Error handling is performed by a subroutine called bsmsg.f. This routine has the following parameters:

```
SUBROUTINE BSMSG(GRPID,ERRCOD,MXLEN,Z,ERROR)
C ======================================================================
C   Purpose
C     To handle the error messages for User Defined Group.
C ----------------------------------------------------------------------
C   Arguments
C
C   GRPID  input  int    ID of group or group name - not used
C   ERRCOD input  int    Error message number if any found
C   MXLEN  input int    Maximum length of the message that can be
C   passed
C   Z      output char   Array to contain the message return
C   ERROR  output int    The code returned indicates the type of error
C
C   Called by
C             BCMSG routine
C--------------------------------
```

The purpose of this subroutine is to return an error String associated with an error code. The error codes are to be returned by the other 7 “BS...” routines and as such the “BSMSG” routine is used as the repository of all of the error messages for the Beam Library applications.

For example, suppose that “BSBRPD” application returns an error code of 5103 when a certain error condition occurs. Then, the Beam Library Client routines will expect that there will be a String returned from the “BSMSG” routine, which corresponds to this error code. These error messages will be printed in the “*.f06” file to guide the user as to what the error could have been and how to fix it. The string may be as long as 160 characters for this release of NX Nastran.

The following is an example of BSMSG code construct:

```
IF(ERRCOD .EQ. 5103) THEN
 Z(1:MXLEN) =  'This is a User Specified Error Message ....'//

&?>              'Messages could be 160 characters long ...'
 ERROR = ERRCOD
 .....
```

Again, it is highly recommended that you use the example Beam Server files as a template to generate your BSMSG routines.

Linking Your Library to NX Nastran

Once you have created the eight “BS...” routines, these routines may be linked with NX Nastran Beam Server Library to build a beam server executable. It is highly recommended that you study, build, and use the example “Beam Server” before you build your own version of the beam server.

The NX Nastran special library contains a main routine as well as the communications routines that allow NX Nastran to communicate with the user-defined beam server.

You may connect up to 10 beam servers in a single job execution. This connection is made using the concept of evaluator groups described in the remainder of this section. For each group, the user specifies on the PBEAML/PBARL entry referring to an external beam server, NX Nastran will start and communicate with the beam server.

You may define as many beam evaluators as required using the “CONNECT” FMS commands. Only 10 of these evaluators, however, may be referenced in the groups on the PBARL or PBEAML Bulk Data entries.

The PBARL/PBEAML entries specifies a “Group” name on the fourth field of the first entry. This group name is associated with an “Evaluator” class using a “Connect” command in the FMS section.

Finally, the “Evaluator” class is associated with an executable using the NX Nastran configuration file specified via the “gmconn” key word on the nastran command line.

The following example shows the mappings mentioned:

1. Group is referenced on the PBARL/PBEAML entry (or entries).

   PBARL,39,6,LOCSERV,I_SECTION (Specify the “Group” name.)

2. The “Group” is associated with an “Evaluator” class.

   CONNECT,BEAMEVAL,LOCSERV,EXTBML (Associate the “Group” name with an “Evaluator” class.)

3. The external evaluator connection file associates the “Evaluator” class with a server executable. The following statement must be specified in the connection file:

   EXTBML,-,*beam_server_pathname*

4. Refer to the external evaluator connection file on the command line using the “gmconn” keyword

   nastran myjob ... gmconn=*external_evaluator_pathname.*

5. In the example Beam Library section that standard output (FORTRAN unit 5) or standard output (FORTRAN unit 6) are not used as these I/O channels are reserved by the Inter-Process Communications (IPC) subsystem.

Example of Building and Linking a Beam Server

As an example of building and linking a beam server executable, the sample beam server will be modified. Complete instructions on building and using a beam server are provided in the *NX Nastran Installation and Operations Guide* for your system.

- Make a copy of the beam server sample source.
- Edit the source for the BRTUPD subroutine; this routine describes the equations that convert the PBARL dimensions into the standard PBAR dimensions for a tube cross section.
- Add an extra multiplication of 3.0 to the DIMO(2) equation to increase the calculated moments of inertia.

Since the formulation of this bar section has been changed, the sensitivities for optimization will also change. Rather than calculate what the new sensitivities should be, the NX Nastran can calculate them using central differencing techniques. To permit this, edit the source file for the BRTUID subroutine and change all occurrences of SENTYP = 2 to SENTYP = 3.

Build your new beam server using the instructions detailed in the *Installation and Operations Guide* . Once you have built the beam server executable, you must create an external evaluator connection to point to your executable. Typically, this file would be kept in the user's home directory, but for this example it will remain in the current directory. Edit the new file bmconfig.fil. Put the following line in the file:

```
LOCBMLS,-,pathname
```

where LOCBMLS is the evaluator referenced in the SAMPLE data file included with the beam server. Remember, this file can contain references to any number of beam servers.

To run the sample job, type in the following command:

```
nastran sample scr=yes bat=no gmconn=bmconfig.fil
```

Common problems which may occur when attempting to run an external beam library job are generally indicated in the F06 by USER FATAL MESSAGE 6498. If this message includes the text “No such group defined,” the PBARL/PBEAML selected a group not defined on a CONNECT entry. If UFM 6498 includes the text “No such evaluator class,” either the “gmconn” keyword was not specified or the CONNECT entry selected an evaluator not defined in the configuration file.

If the job was successful, you can look at the Design Variable History and see that the results for the variable mytubeor are different than the results for tubeor. These variables refer to the outer radius of tube sections from equivalent models. One model used the provided tube section while the other used the tube section in your modified beam server.

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.