

# NX Nastran 12.0.1 Release Guide



# Contents

<b>Proprietary &amp; Restricted Rights Notice</b> . . . . .	<b>5</b>
<b>Topology Optimization</b> . . . . .	<b>1-1</b>
Topology Optimization Enhancement . . . . .	1-1
<b>Multibody dynamics</b> . . . . .	<b>2-1</b>
Multibody dynamics . . . . .	2-1
<b>Windows DMP</b> . . . . .	<b>3-1</b>
Windows DMP single host . . . . .	3-1
<b>Caveats</b> . . . . .	<b>4-1</b>
Caveats . . . . .	4-1
<b>Problem Report (PR) fixes</b> . . . . .	<b>5-1</b>
Problem Report (PR) fixes . . . . .	5-1



# Proprietary & Restricted Rights Notice

© 2018 Siemens Product Lifecycle Management Software Inc. All Rights Reserved.

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

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

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

All other trademarks are the property of their respective owners.

## TAUCS Copyright and License

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

TAUCS License:

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

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

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

## Availability (TAUCS)

As of version 2.1, we distribute the code in 4 formats: zip and tarred-gzipped (tgz), with or without binaries for external libraries. The bundled external libraries should allow you to build the test programs on Linux, Windows, and MacOS X without installing additional software. We recommend that you download the full distributions, and then perhaps replace the bundled libraries by higher performance ones (e.g., with a BLAS library that is specifically optimized for your machine). If you want to conserve bandwidth and you want to install the required libraries yourself, download the lean distributions. The zip and tgz files are identical, except that on Linux, Unix, and MacOS, unpacking the tgz file ensures that the configure script is marked as executable (unpack with `tar xzvpf`), otherwise you will have to change its permissions manually.



# Chapter 1: Topology Optimization

## Topology Optimization Enhancement

Beginning with NX Nastran 12.0, you can use the NX Nastran optimization solution sequence (SOL 200) to run a topology optimization. The SOL 200 topology optimization removes material in a way that is consistent with your objective function and constraints. You can request a single topology optimization solution across multiple analysis types including statics and dynamics.

An alternative enhanced DESOBJ command is available beginning with the NX Nastran maintenance package 12.0.0 MP1. The enhancement simplifies the specification of jobs involving design objectives (DESOBJ) on the minimization of a function of compliance values for multiple static subcases, with specific emphasis on the maximum of these compliance values. While this can already be done in NX Nastran 12.0 with the DRSPAN or BETA methods described in the NX Nastran 12.0 Release Guide, the enhanced DESOBJ command will make it much easier to prepare input data and to run jobs through Simcenter Pre/Post 12.0.0 MP1 directly.

The Simcenter Pre/Post 12.0.0 MP1 release inputs will not change. However, when you use Simcenter Pre/Post 12.0.0 MP1 and you assign a compliance (CMPLNCE) response (DRESP1) to the objective function, Simcenter Pre/Post will automatically write the following to the NX Nastran input file:

```
DESOBJ (MIN, SCSET=ALL, SCFUNC=MAX) = ID_of_compliance_DRESP1
```

In the above, SCSET=ALL refers to all static subcases, and SCFUNC=MAX specifies that the objective is to minimize the maximum of the compliance values from the multiple static subcases.

You can also modify the values assigned to SCSET and SCFUNC using either the Simcenter Pre/Post text editor window or an external text editor to allow any of the options described below.

The updated DESOBJ format is as follows.

$$\text{DESOBJ} \left[ \left( \begin{array}{c} \boxed{\text{MIN}} \\ \boxed{\text{MAX}} \end{array} \right), \text{SCSET} = \left\{ \begin{array}{c} m \\ \text{ALL} \end{array} \right\}, \text{SCFUNC} = \left\{ \begin{array}{c} \text{SUM} \\ \text{AVG} \\ \text{SSQ} \\ \text{RSS} \\ \text{MAX} \\ \text{MIN} \end{array} \right\} \right] = n$$

The MIN or MAX selection still requests that the objective is to be minimized or maximized.

n still refers to a response ID, although, when using the new format (that is, SCSET and SCFUNC are defined), n can only refer to a compliance response defined with the DRESP1 bulk entry.

SCSET=ALL refers to all static subcases.

SCSET=m refers to specific static subcases, where m is the ID of a SET case control command that lists the relevant subcases.

SCFUNC=*function* has the following function choices:

Function	Description
SUM	Compute the sum of the compliance values from the multiple static subcases.
AVG	Compute the average of the compliance values from the multiple static subcases.
SSQ	Compute the sum of the squares of the compliance values from the multiple static subcases.
RSS	Compute the square root of the sum of the squares of the compliance values from the multiple static subcases.
MAX	Use the maximum of the compliance values from the multiple static subcases.
MIN	Use the minimum of the compliance values from the multiple static subcases.



## Chapter 2: Multibody dynamics

### Multibody dynamics

NX Nastran is used by Simcenter Motion to optionally reduce a multi-body dynamics component to a flexible body.

Beginning with NX Nastran 12.0.1, when you run a flexible body reduction from Simcenter Motion, residual vectors are automatically turned off. This occurs even if you request residual vectors with the RESVEC=YES case control command.

Specifically, when NX Nastran recognizes the **MBDEXPORT SCMOTION** case control command in an input file, it will automatically turn off residual vectors.

The updated remarks on the MBDEXPORT case control command are included below.

## Updated MBDEXPORT

### Multibody Dynamics Export

Creates interface file for multibody dynamics and control system software.

THERE ARE  
NO UPDATES  
TO THE  
FORMAT:

$$\begin{aligned}
 & \text{MBDEXPORT} \left[ \begin{array}{l} \text{SCMOTION} \\ \text{RECURDYN} \\ \text{ADAMS} \\ \text{SIMPACT} \\ \text{OP4} = \text{unit} \\ \text{MATLAB} \end{array} \right], \left[ \begin{array}{l} \text{STANDARD} \\ \text{STATESPACE} \end{array} \right], \left[ \text{FLEXBODY} = \left\{ \begin{array}{l} \text{NO} \\ \text{YES} \end{array} \right\} \right], \\
 & \left[ \text{FLEXONLY} = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\} \right], \left[ \text{MINVAR} = \left\{ \begin{array}{l} \text{PARTIAL} \\ \text{CONSTANT} \\ \text{FULL} \\ \text{NONE} \end{array} \right\} \right], \left[ \text{PSETID} = \left\{ \begin{array}{l} \text{NONE} \\ \text{setid} \\ \text{ALL} \\ \text{sktunit} \end{array} \right\} \right], \\
 & \left[ \text{OUTGSTRS} = \left\{ \begin{array}{l} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{OUTGSTRN} = \left\{ \begin{array}{l} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{RECVROP2} = \left\{ \begin{array}{l} \text{NO} \\ \text{YES} \end{array} \right\} \right], \\
 & \left[ \text{CHECK} = \left\{ \begin{array}{l} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{NONCUP} = \left\{ \begin{array}{l} -1 \\ -2 \end{array} \right\} \right]
 \end{aligned}$$

UPDATED  
SIMCENTER  
MOTION  
REMARKS:

1. The generation of standard matrices and the writing of them to a Simcenter Motion OP2 file via OUTPUT2, which is only applicable in a non-restart, residual-only SOL 103 analysis, is initiated by MBDEXPORT SCMOTION FLEXBODY=YES. A specific ASSIGN statement is not required; the data will be written to the standard OP2 file via PARAM,POST,-2 or PARAM,POST,-1. When PARAM,POST,-2 is defined, PARAM,OUGCORD,GLOBAL must also be defined.
2. The parameter LMODES can be used to control which modes are used to derive the final modal basis.
3. The mode shape output will be reduced to the DOF defined in the DISPLACEMENT case control output request.
4. Simcenter Motion requires the definition of the component modal DOF, and the fixed and free boundary DOF. The SPOINT and QSETi bulk entries define the

component modal DOF. The BSETi and CSETi bulk entries define the fixed and free boundary DOF, respectively.

5. Residual vectors are automatically turned off for a solution which includes MBDEXPORT SCMOTION.



## Chapter 3: Windows DMP

### Windows DMP single host

Previously, to run Windows DMP on a single host which includes multiple cores, system administrator tasks were first required. These tasks are described in the *NX Nastran 12 Parallel Processing Guide* under the heading **Windows Single Host Instructions**.

Beginning with NX Nastran 12.0, you are no longer required to complete these system administrator tasks. You can now run Windows DMP on a single host which includes multiple cores by simply including the keyword **DMP**.



## Chapter 4: Caveats

### Caveats

The NX Nastran 12.0.1 maintenance release includes the following caveats.

- (PR 8361246) A fatal error occurs when using CGAP elements as contact elements in SOL 101 and one or both of the grid points on the CGAP entry have an 8 digit ID. CGAP elements are treated as contact elements in SOL 101 when the BCSET case control is defined.

Workaround: Ensure that the grid IDs used for any gap elements are less than 8 digits.

Problem originated: NX Nastran 11.0

This problem will be fixed in NX Nastran 12.0.2.

- (PR 9005850) Requesting MONPNT3 output in a cylindrical or spherical coordinate system results in incorrect output.

Workaround: Only reference Cartesian coordinate systems for MONPNT3 output.

Problem originated: NX Nastran 12.0

This problem will be fixed in NX Nastran 12.0.2.

- (PR 9044460) A problem in SOL 401 occurs when a static subcase (subcase 1) includes a time-unassigned follower load such as a pressure load, and a consecutive static subcase (subcase 2) that is sequentially dependent (SEQDEP=YES) on subcase 1. The problem is that the follower load correction factors are incorrect in subcase 2. Note that time-unassigned loads are selected in the case control with the LOAD=n command.

Workaround: We strongly recommend that you apply follower loads as time-assigned. That is, select your follower loads in the case control with the DLOAD=n command. The DLOAD=n command should be defined globally (above the subcase level) such that it applies to all static subcases. Then you can control the load application using specific time step definitions in each subcase. That is, you would define a TSTEP=n case control command in each subcase which references a specific TSTEP1 bulk entry.

If you are unable to use the workaround above and your follower load must be time-unassigned, an alternate workaround is to define the parameter LVAR to "STEP" on the NLCNTL bulk entry for subcase 2.

Problem originated: NX Nastran 11.0

This problem will be fixed in NX Nastran 13.0.

- (PR 7310163) For a model running SOL 101 which included a bolt preload on a beam element, contact on shell elements, and the contact conditions included friction and SHLTHK=0 defined on the BCTPARM entry, the applied moment loads and the SPC Force resultant output did not balance.

Workaround: None

Problem originated: NX Nastran 5.0

- (PR 8351716) Grid points used on CELAS2 elements that do not also belong to structural elements have no results when you solve a SOL 402 normal modes subcase.

Workaround: None

Problem originated: NX Nastran 12.0

This problem will be fixed in NX Nastran 12.0.2.

- (PR 8351720) When mass normalization is requested on the EIGRL bulk entry for a SOL 402 cyclic modes subcase, the eigenvector normalization is incorrect.

Workaround: Use a different normalization method.

Problem originated: NX Nastran 12.0

This problem will be fixed in NX Nastran 12.0.2.

- (PR 8356744) On Windows, SOL 402 fails with an error when you run a model which includes a MODAL, BUCKLING, FOURIER or CYCMODES subcase, and the network path to run the NX Nastran executable includes a space.

Problem originated: NX Nastran 12.0

This problem will be fixed in NX Nastran 12.0.2.

- (PR 8362037) In a SOL 200 Topology Optimization problem, the manufacturing constraints (including default checkerboard) cannot be used together in the model with the following element types: CBAR, CBEAM, CBUSH, CBUSH1D, CDAMP\*, CELAS\*, CFAST, CGAP, CMASS\*, CONM\*, CONROD, CPLSTN\*, CPLSTS\*, CQUADX\*, CROD, CSHEAR, CTRAX\*, CTUBE, CVISC, CWELD. Elements that can be used together with the manufacturing constraints are CTRIA\*, CTRIAR, CQUAD\*, CQUADR, CHEXA, CPENTA, CPYRAM, CTETRA (supported elements for topology optimization design area) and RBAR, RBE\*, RROD (not in design area).

Workaround: With the following element types: CDAMP\*, CELAS\*, CFAST, CGAP, CMASS\*, CONM\*, CVISC and CWELD, the issue can be avoided by assigning their IDs larger than any of the elements supported by topology optimization regardless if the supported elements are active or inactive.

Problem originated: NX Nastran 12.0

This problem will be fixed in NX Nastran 12.0.2.



## Chapter 5: Problem Report (PR) fixes

### Problem Report (PR) fixes

The NX Nastran 12.0.1 maintenance release includes the following fixes.

PR#	Problem Originated	Problem Description
9038258	8.0	<p>Applying an acceleration enforced motion with the SPCD method produced the wrong result in SOL 112 when the IC(TZERO) capability was also used. The generalized forces for enforced motions were not accounted for in the DMAP code.</p> <p>There is no workaround in previous releases.</p>
9030914	12.0	<p>Issue with coupling and FEMAO.</p> <p>The model for this PR included two fluids separated by a plate. A pressure excitation was defined on one side of the plate (on the fluid). When solving with standard FEM (without FEMAO), the pressure was correctly transmitted to the other side of the plate. When FEMAO was activate, no pressure was transmitted.</p> <p>The issue was that the constraints of the structural plate were not properly dealt with during the coupling stage. The software stopped all transmission by applying the constraints along the wrong axis.</p> <p>There is no workaround in previous releases.</p>
9029595	12.0	<p>In a SOL 108 acoustics solution, during the post processing stage of the analysis, an invalid caseID of 0 was internally generated, resulting in incorrect and semi-random results for that particular postprocessing step. This has been corrected in NX Nastran 12.0.1.</p> <p>There is no workaround in previous releases.</p>
9022227	12.0	<p>In a SOL 108 acoustics solution which included a porous material, the .f06 file included some debug information which should not be included in the output. This output included the following:</p> <pre>nMatPor= 1 1 3</pre> <p>This information is now suppressed in NX Nastran 12.0.1.</p> <p>There is no workaround in previous releases.</p>

9012653	1.0	<p>When solving a vibro-acoustic panel model with SOL 108 which included a static subcase to pre-stress the structure, the following error was generated in the .f06 file:</p> <pre>*** SYSTEM FATAL MESSAGE 6144 (MERGE1) THE SIZES OF THE INPUT MATRICES AND PARTITIONING VECTORS ARE INCOMPATIBLE.</pre> <p>This DMAP bug has been fixed in the NX Nastran 12.0.1 release.</p> <p>There is no workaround in previous releases.</p>
9010232	12.0	<p>The results from a rotor dynamics model were different when comparing the results from NX Nastran 11.0 and NX Nastran 12.0.</p> <p>This problem was related to the parameter ROTCOUP which is used when connecting a rotor to an unsymmetrical support structure and responses at discrete rotor angles are determined. This has been corrected for NX Nastran 12.0.1.</p> <p>There is no workaround in previous releases.</p>
9008399, 9005039	12.0	<p>In SOL 401, a model which included a glue set and a bolt defined with ETYPE=2 produced the following fatal error:</p> <pre>*** USER FATAL MESSAGE 4690 (CNPAIR) GLUE SET ID 40 IS REFERENCED BY BOTH BGSET AND BGADD CARDS.</pre> <p>In a SOL 401 solution, glue set ids are created internally for the ETYPE=2 bolts, and they are assembled as part of a BGADD entry. This bug has been fixed such that the internally generated IDs will no longer conflict with the IDs which already exist in the input file.</p> <p>Workaround for the previous release: Relabel the IDs of the glue sets in your input file such that they no longer conflict with the internally generated IDs shown in the error message.</p>
9008359	12.0	<p>When creating an FRF component using the external superelement approach (FRFOUT), residual vectors were not automatically being computed. This led to incorrect results as compared to the SOL 108 results. This has been corrected for NX Nastran 12.0.1.</p> <p>There is no workaround in the previous release.</p>

9008341	10.0	<p>The recovery of composite stress results from a frequency response random analysis was taking a long time. The cause of the slow recovery is that frequency response analysis results are complex, and the software was calculating the failure indices and strength ratios at discrete phase angles over a full 360 degree range. By default, this calculation is performed in one degree increments. That means for each ply of a composite shell element, failure indices and strength ratios are calculated 360 times.</p> <p>Since failure indices and strength ratios are not output for a random analysis in general, this calculation can be skipped. As a result, we have now removed some unnecessary calculations.</p> <p>There is no workaround in previous releases.</p>
9005947	12.0	<p>In NX Nastran 12.0, the capability to specify imperfections was introduced for SOL 401. When an imperfection is specified, the original model geometry is modified by the imperfections. The resulting geometry after imperfections is then treated as the reference geometry on which the analysis is carried out.</p> <p>Imperfections defined with a Cartesian coordinate system were applied correctly by the software. Although, the software did not correctly apply imperfections defined with cylindrical or spherical coordinate systems.</p> <p>The software now identifies the coordinate system type in which the imperfections are specified and applies appropriate transformations to obtain correct imperfections. With the updates in 12.0.1, imperfections specified in cylindrical and spherical coordinate systems are now processed correctly.</p> <p>There is no workaround in the previous release.</p>
8992158	6.0	<p>A model which included a large number of CWELD/CFAST elements defined with the PARTPAT format took a long time to complete in the MODGM2 module.</p> <p>This performance problem has been fixed for the NX Nastran 12.0.1 release. The elapsed time for this module has been reduced by an approximately factor of three. The use of RAID configured disks and/or SSD disks will also help reduce the elapsed time of a NX Nastran solution.</p> <p>There is no workaround in previous releases.</p>
8352459	8.0	<p>Random analysis subcase labels are incorrect in the output.</p> <p>The random analysis subcase labeling problem is fixed in the NX Nastran 12.0.1 release.</p> <p>There is no workaround in previous releases.</p>

9002397	11.0	<p>In an acoustic analysis, a FATAL error occurred when converting CQUAD4 elements to CQUADR elements when the original CQUAD4 elements were referenced by a PMIC entry. The PMIC entry defines an acoustic microphone element property. The system cell definition 370=5 requests this conversion of CQUAD4/CTRIA3 to CQUADR/CTRIAR.</p> <p>The reported fatal error was as follows:</p> <pre> *** USER FATAL MESSAGE 5419 (MODGM2) STRUCTURAL ELEMENT QUADR ID = 8002 IS ATTACHED TO A FLUID GRID POINT ID = 6010 FATAL ERROR </pre> <p>Now when SYSTEM(370)=5 is defined, CTRIA3/CQUAD4 will not be converted to CTRIA3/CQUADR when they are associated with the PMIC entry.</p> <p>There is no workaround in previous releases.</p>
9001494	10.0	<p>When using the elemental iterative solver with SOL 101, no contact iteration information was displayed in the Simcenter solution monitor.</p> <p>Beginning with NX Nastran 12.0.1, the contact iteration status will now output to the solution monitor.</p> <p>There is no workaround for previous releases.</p>
8351947	1.0	<p>In a SOL108 random analysis, a fatal error occurred but did not include any details about what caused the error.</p> <p>The fatal error was a result of an empty load and response. The file for the load of frequency response existed, but it only included zero values.</p> <p>A load check step has been added beginning in NX Nastran 12.0.1 before calling the FRRD module which will now generate a detailed fatal message if this occurs.</p> <p>There is no workaround in previous releases.</p>
8994479	5.0	<p>For a shock spectrum analysis using different summation methods in different shock spectrum subcases, an incorrect warning message was being written to the f06 file.</p> <p>The problem was due to an invalid data check. The data check was preventing the PARAM,OPTION from being allowed at the subcase level. The invalid check has been removed in NX Nastran 12.0.1 and the problem now solves correctly.</p> <p>There is no workaround in previous releases.</p>

8992079	12.0	<p>A SOL 108 acoustics analysis running NX Nastran 12.0 hangs when attempting to compute ATVs in a DMP parallel run on Windows OS.</p> <p>This issue was traced to an incorrect implementation in the DMP case, causing all processes to attempt to open the OP2 file. The fix was to restrict opening of the OP2 file to the single (master) process.</p> <p>There is no workaround in previous releases.</p>
8991679	12.0	<p>The fatal error below occurred when the 2D or 3D PLOTEL elements introduced in NX Nastran 12.0 were included in a SOL 103 superelement solution.</p> <pre data-bbox="683 680 1448 793"> *** SYSTEM FATAL MESSAGE 3200 (BDYINF) LOGIC ERROR 8 ENCOUNTERED IN SUBROUTINE BDYINF *** SYSTEM WARNING MESSAGE 2072 (BDRYINFO) CARD TYPE 5201 NOT FOUND ON DATA BLOCK. </pre> <p>Now in NX Nastran 12.0.1, all the new 2D and 3D PLOTEL elements are correctly processed.</p> <p>There is no workaround in previous releases.</p>
8989579	12.0	<p>A SOL 108 acoustics solution which includes an ATV running with DMP crashes when more DMP processors were requested than frequencies.</p> <p>This issues has been fixed. This model no longer crashes in NX Nastran 12.0.1.</p> <p>Workaround for the previous release: Either reduce the number of DMP processors to match the number of frequencies or increase the number of frequencies to match the number of DMP processes.</p>
8349211	11.0	<p>A cyclic normal mode subcase requesting MAX normalization in SOL 401 produced unexpected normalized results.</p> <p>Specifying MAX normalization lead to results where the largest component displacement magnitude was less than 1.0 for the cyclic normal modes. The wrong value was being used to normalize the vectors. The fix includes identifying the largest single displacement component (for the physical Eigen vector, not the Sine and Cosine components in the Cyclic/Fourier modal solutions), then using this value to normalize the Eigen vector. The previous implementation was identifying the largest total nodal displacement magnitude and using that to normalize the Eigen vector.</p> <p>There is no workaround in previous releases.</p>
8348400	12.0	<p>In a SOL 108 acoustics analysis running with DMP, analysis.dmp.intel.exe hangs when trying to compute ATVs.</p> <p>This issue has been fixed in NX Nastran 12.0.1.</p>

9039421	9.0	<p>Loads defined with the RFORCE1 entry are not being applied when using the element iterative solver in SOL 101.</p> <p>A bug has been fixed in NX Nastran 12.0.1 in SOL 101 with the element iterative solver where the mass matrix was not being produced. The zero mass matrix was producing a zero load vector for the RFORCE1 loading.</p> <p>There is no workaround in previous releases.</p>
8984348	10.0	<p>A model with plane strain elements produces different SPCFORCE output when comparing output from SOL106 and SOL601,106.</p> <p>The SOL106 SPCFORCE results on the plane strain element were incorrect. The problem was due to the SYSTEM(587) setting incorrectly changing the plane strain element output to a per 2*PI bases. This has been corrected in NX Nastran 12.0.1.</p> <p>There is no workaround in previous releases.</p>
8355009	12.0	<p>A SOL402 model which included a PSOLID entry with IN=0 and ISOP=1 or FULL produced the following warning message:</p> <pre> *** USER WARNING MESSAGE 5460 (IFS5P) ON PSOLID ENTRY PID=1 FULL OR 1 IS SPECIFIED IN THE "ISOP" FIELD AND BLANK, 0, OR BUBBLE IS SPECIFIED IN THE "IN" FIELD. USER INFORMATION: 1. THE "ISOP" FIELD WILL BE RESET TO 0. 2. IF THIS ENTRY IS REFERENCED BY CHEXA OR CPENTA ELEMENTS WITH MIDSIDE NODES, THEN THE "IN" FIELD WILL BE RESET TO 3. </pre> <p>The software reset ISOP to 0 and the internal modes of the solid elements were not deactivated as expected.</p> <p>A workaround in 12.0 is to create the file <b>input_file_prefix.add</b> with the following:</p> <pre> .AEL EAS 0 RETURN </pre> <p>For example, if your input file is named <b>psolid_test.dat</b>, your *.add file should be named <b>psolid_test.add</b>.</p> <p>Beginning in NX Nastran 12.0.1, the warning message will not be displayed when you run SOL402 with a similar PSOLID definition.</p>

8353663	12.0	<p>In SOL402, when a property entry references a material coordinate system, that coordinate system ID was not included in the stress and strain output result datablock CID field. Since these results are output in material coordinates, Simcenter post-processing was not able to transform these results into post processing coordinate systems.</p> <p>Workaround for previous release: You can avoid specifying a material coordinate system on the property entry or look at the results in "native" format in Simcenter post processing.</p>
8352952	12.0	<p>The SOL 402 contact sliding distance output was incorrectly computed. Specifically, the total contact sliding distance for 3D solid elements was not computed and remained zero, and the total contact sliding distance for 2D plane strain elements incremented even when the contact was lost.</p> <p>There is no workaround in the previous release.</p>
8351722	12.0	<p>In SOL 402, a MODES subcase included the RSUB=n parameter to reference a STATICS subcase for its initial computation state. The MODES subcase ignored the static preload computed in the STATICS subcase. This has been fixed for NX Nastran 12.0.1.</p> <p>Workaround for the previous release: You can reorder the subcases so that the MODAL subcase directly follows the STATICS subcase, and use the default of SEQDEP=YES in the MODAL subcase (that is, do not define SEQDEP=NO in the MODAL subcase).</p>
8351721	12.0	<p>SOL 402 incorrectly output strain tensors. Specifically on solid elements, non-diagonal components of strain tensors other than total strain were written using the wrong convention (epsilon instead of gamma) <math>E_{IJ} = G_{IJ} / 2</math>. The Von Mises was output correctly. The displayed strain results from the previous release (12.0) should be multiplied by 2 when visualizing non-diagonal components.</p>
8351719	12.0	<p>The content of TRMBD/TRMBU datablocks output from SOL 402, which enables to rotate the results on deformed elements, was incorrect and not always written (depending on the output requests). The consequence is that stress and strain results may not be correct in Simcenter post processing when looking at individual components of tensors if the initial element axes are not aligned with the global coordinate system. This is not an issue with Von Mises results.</p> <p>Workaround for previous release: The results can be displayed in "native" format in Simcenter.</p>

8956181	12.0	<p>If a model with shell elements was solved using SOL 402, the mean normal was not computed for the nodes along the edges, even though the angle was smaller than what was defined with the parameter SNORM.</p> <p>This has been fixed in NX Nastran 12.0.1.</p> <p>There is no workaround in the previous release.</p>
8356738	12.0	<p>Two problems were discovered when running DMP computations in SOL 402:</p> <ul style="list-style-type: none"> <li>- Excessive memory consumption. The multiple solution DMP processes each allocated the maximum memory. This allocation way exceeded the user-defined memory causing a fatal error when the machine did not have this resource. The workaround for this was to divide the memory zone by 2 times the number of processors.</li> <li>- MPI crash because of a mix of slash and backslashes in the executable path when running DMP computation over the network. If possible, the installation locations should be copied locally to avoid the issue.</li> </ul>
8351724	12.0	<p>Stress results from SOL 402 might be missing in the .op2 file from a MODAL subcase if the model contains composite elements.</p> <p>Workaround for previous release: You can specify the environment variable NXN_FILEKEEP=1, rerun the solution, then load the results file NAME.DES file into Simcenter Post. This file type needs to be enabled in the Simcenter customer defaults to allow Simcenter to read the *.des file type.</p>
8351723	12.0	<p>The ply thermal strain data block is written twice in the .op2 file from SOL 402 for multilayer shell elements. However, the results in both data blocks were correct and could be visualized in Simcenter.</p>
8351714	12.0	<p>There was a problem with the damage result from SOL 402 when associating MATDMG and PSOLID. In the ODAMGPFID datablock identification, the type of law was set to 0 instead of 1 or 2. Therefore, a warning message is written at the end of the .f06 file and Simcenter is not able to display the damage results properly.</p> <p>Workaround for previous release: The MATDMG material should be used with a PCOMPS property instead of PSOLID.</p>
9035074	12.0	<p>When running a SOL 109 analysis using Virtual Fluid Mass, using a selected element output group with the SET case control results in fluid pressures not being written to the f06 file. The problem was found to be associated with the set processing.</p> <p>Workaround for previous release: Using MPRES(PRINT) = All will result in fluid pressures being written to the f06 file.</p>





## 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](http://www.siemens.com/plm).

© 2018 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.