NX Nastran 8 Release Guide

Proprietary & Restricted Rights Notice

TAUCS Copyright and License

Availability (TAUCS)

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

Contents

Chapter

1 Dynamics

Residual vectors

Normal modes computed for use in a modal response solution (SOL 111 or SOL 112) are typically a reduced representation of a structure. The computed modes may represent most of the dynamic behavior of the structure, although higher frequency modes which are not computed can contribute to errors in the resulting response. The higher modes typically contribute statically to the response. Residual vectors can optionally be requested to improve the response by adding the missing static flexibility associated with these higher modes.

Previously, your only option to request the calculation of residual vectors was to use parameters. Now you can use the new RESVEC case control command to request residual vectors. The input format for the new command is as follows.

$$\mathrm{RESVEC}\left[\left(\begin{bmatrix}\mathrm{APPLOD}\\ \mathrm{NOAPPL}\end{bmatrix},\begin{bmatrix}\mathrm{RVDOF}\\ \mathrm{NORVDO}\end{bmatrix},\begin{bmatrix}\mathrm{INRLOD}\\ \mathrm{NOINRL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\mathrm{SYSTEM}\\ \mathrm{NOSYSTEM}\\ \mathrm{COMPONENT}\\ \mathrm{NOCOMPONENT}\\ \mathrm{BOTH\ or\ YES}\\ \mathrm{NO}\end{Bmatrix}$$

The highlighted describers are applied by default if you include “RESVEC” in your case control.

- The residual vectors created from a PARAM,RESVEC,YES entry can now be obtained by specifying the RESVEC case control command with the APPLOD or RVDOF describers.
    - o If you specify the APPLOD describer (default), residual vectors are requested for applied loads. There are two input scenarios to determine the applied loads for computing residual vectors.
        - ■ The load set IDs selected with the EXCITEID field on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries in the bulk data are processed. No DLOAD case control is required in

this case. If a LOADSET case control command exists, these are all ignored.

- ■ If a LOADSET case control command exists, and it selects an LSEQ bulk entry, the load set ID selected in the LID field in the LSEQ bulk entry is processed.

- o If you specify the RVDOF describer (default), residual vectors are requested for DOF included in RVDOF and RVDOF1 bulk entries. RVDOF and RVDOF1 are new bulk entries which you can use to select DOF in both the residual structure (a-set) and superelements (o-set).

  In comparison, when requesting residual vectors using the PARAM,RESVEC,YES entry, the USET,U6 bulk entry can be used to select residual structure DOF (a-set), and the SEUSET,U6 bulk entry can be used to select superelement DOF (o-set). The new RVDOF and RVDOF1 bulk entries have the advantage that they can select DOF in both the residual structure (a-set) and in superelements (o-set).

  In addition, the unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

- The residual vectors created from a PARAM,RESVINER,YES entry can now be requested by specifying the RESVEC case control command with the INRLOD describer. The INRLOD describer requests residual vectors for inertia relief loads (unit acceleration of mass).

- The right hand side of the new RESVEC command includes the SYSTEM / NOSYSTEM, COMPONENT / NOCOMPONENT, BOTH or YES, and NO describers.

  - o You can use the SYSTEM describer to request residual vectors for residual structure DOF (a-set) only.

  - o You can use the COMPONENT describer to request residual vectors for superelement DOF (o-set) only.

  - o You can use the BOTH or YES describers to request residual vectors for both residual structure DOF (a-set) and for superelement DOF (o-set). This is the default option.

- You can use the NO describer to request that no residual vectors be computed.

- A unique RESVEC case control definition can be defined in different superelement subcases. You can use the NOSYSTEM and NOCOMPONENT describers in a superelement subcase to turn off a setting which may have been defined globally.

The following table summarizes the various residual vector requests you can specify with the describers on the RESVEC case control command or parameters. When using only parameter inputs, PARAM,RESVEC,YES must be included with all other residual vector parameters. If the RESVEC case control command is present, the parameters RESVEC and RESVINER are ignored. The other residual vector parameters can be used in conjunction with the RESVEC case control command.

Describer	**Parameter Input**	**Result**
APPLOD	PARAM,RESVEC	Both the APPLOD describer and PARAM,RESVEC request residual vectors for applied loads. PARAM,RESVEC also requests residual vectors for all DOF included in USET,U6 and SEUSET,U6 bulk entries. Although, the software ignores USET,U6 and SEUSET,U6 bulk entries when the RESVEC case control command is defined.
RVDOF		Residual vectors are requested for all DOF included in RVDOF and RVDOF1 bulk entries.
INRLOD	PARAM,RESVINER	Residual vectors are requested for inertia relief load (unit acceleration of mass).
SYSTEM		Residual vectors are requested for residual structure DOF (a-set) only.
COMPONENT		Residual vectors are requested for superelement DOF (o-set) only.
BOTH or YES		Residual vectors are requested for both residual structure DOF (a-set) and for superelement DOF (o-set).
	PARAM,RESVALT	Includes inclusion of dynamic effects of residual vector modes.

Describer	Parameter Input	Result
	PARAM,RESVSE	The default 'NO' will skip the printing of the strain energy of the static shapes (labeled "EXTERNAL WORK"). You can use PARAM,RESVSE,YES to print the strain energy.
	PARAM,RESVSLI	The default 'YES' will test for linearly independent shapes. You can use PARAM,RESVSLI,NO to skip the tests.
	PARAM,RESVSO	The default 'YES' will clean up the static vectors to insure orthogonality to modes. If the stiffness matrix is well conditioned this is unnecessary. You can use PARAM,RESVSO,NO to skip the reorthogonalization.
	PARAM,RESMETH	Method used for eigensolution in reduced basis. PARAM,RESMETH,LAN is an alternate method, somewhat more robust but slower for this class of reduced basis problem. Default = AHOU.
	PARAM,RESVPGF	This parameter is used to filter out small terms in the residual vectors. The default is 1.E-6, meaning that terms in the residual vectors whose absolute values are smaller than 1.E-6 will get filtered out before they are used. To eliminate filtering of residual vectors, you can set RESVPGF to 0.0.
	PARAM,RESVRAT	Filter value for discarding trial vectors with little independent content. You can decrease to discard more vectors. Default = 1.e8

For additional information, see Residual Flexibility in the SubDMAP RESVEC in the *NX Nastran User's Guide*, the new RESVEC case control command and the new RVDOF and RVDOF1 bulk entries.

RESVEC Residual Vector Request

Used to control the computation of residual vectors.

Format:

$$\text{RESVEC}\left[\left(\begin{bmatrix}\text{APPLOD}\\ \text{NOAPPL}\end{bmatrix},\begin{bmatrix}\text{RVDOF}\\ \text{NORVDO}\end{bmatrix},\begin{bmatrix}\text{INRLOD}\\ \text{NOINRL}\end{bmatrix}\right)\right]=\left\{\begin{matrix}\text{SYSTEM}\\ \text{NOSYSTEM}\\ \text{COMPONENT}\\ \text{NOCOMPONENT}\\ \text{BOTH or YES}\\ \text{NO}\end{matrix}\right\}$$

Examples:

```
RESVEC = SYSTEM
RESVEC(NOINRLOD) = COMPONENT
RESVEC = NO
```

Describers:

Describer	Meaning
APPLOD	Compute residual vectors for applied loads. (Default)
NOAPPL	Do not compute residual vectors for applied loads.
RVDOF	Compute residual vectors for degrees-of-freedom included in RVDOF and RVDOF1 bulk entries. (Default)
NORVDO	Do not compute residual vectors for degrees-of-freedom included in RVDOF and RVDOF1 bulk entries.
INRLOD	Compute residual vectors for inertia relief load.
NOINRL	Do not compute residual vectors for inertia relief load. (Default)
SYSTEM	Request residual vectors for residual structure degree-of-freedom only (a-set).
NOSYSTEM	Do not compute residual vectors for residual structure degree-of-freedom (a-set).

Describer	Meaning
COMPONENT	Request residual vectors for superelement degree-of-freedom only (o-set).
NOCOMPONENT	Do not compute residual vectors for superelement degree-of-freedom (o-set).
BOTH (or YES)	Request residual vectors for both residual structure DOF (a-set) and for superelement DOF (o-set).
NO	Turns off calculation of residual vectors.

Remarks:

1. If the RESVEC case control command is present, the parameters RESVEC and RESVINER are ignored.

2. A unique RESVEC case control definition can be defined in different superelement subcases. You can use the NOSYSTEM and NOCOMPONENT describers in a superelement subcase to turn off a setting which may have been defined globally.

3. If a RESVEC case control command is present in a cold start analysis, then only the RESVEC case control command can be used in a restart. Similarly, if parameter RESVEC/RESVINER is used in a cold start, then only the same can be used in a restart.

4. INRLOD designates that inertia load residual vectors are to be calculated. INRLOD is functionally equivalent to PARAM,RESVINER,YES. Inertia loads are computed for each of the 6 basic coordinate system directions and residual vectors are computed for each load.

5. APPLOD designates that the applied load residual vectors are to be calculated. There are two input scenarios to determine the applied loads for computing residual vectors.

   a. The load set IDs selected with the EXCITEID field on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, and SELOAD entries in the bulk data are processed. No DLOAD case control is required in this case. If a LOADSET case control command exists, these are all ignored.

   b. If a LOADSET case control command exists, and it selects an LSEQ bulk entry, the load set ID selected in the LID field in the LSEQ bulk entry is processed.

6. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.

   The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVDOF Degree-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

RVDOF	900	1	200	3					

Fields:

Field	Contents
IDi	Grid point or scalar point identification number. (Integer > 0)
Ci	Component number. ($0 \leq$ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

Remarks:

1. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.

2. The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6 or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVDOF1 Degree-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	“THRU”	ID2					

Example:

RVDOF1	2	101	201	356					

Alternate Example:

RVDOF1	123456	7	THRU	109					

Fields:

Field	Contents
C	Component number. (0 ≤ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
IDi	Grid point or scalar point identification number. (Integer > 0)

Remarks:

1. The operation of the residual vector calculation with RVDOF is functionally equivalent to PARAM,RESVEC,YES when USET, U6 DOF are present. The RVDOF/RVDOF1 bulk entries can select both a-set and o-set DOF. This is different than the USET, U6 capability which requires USET, U6 to select a-set DOF, but requires SEUSET, U6 to select o-set DOF.

2. The unit loads applied to the interior points of a superelement due to RVDOFi bulk entries are passed downstream to the residual for the purpose of residual vector processing by all superelements in its downstream path. This produces more accurate results as compared to the results produced when USETi,U6

or SEUSETi,U6 bulk entries are used for residual vector processing. When USETi,U6 or SEUSETi,U6 bulk entries are used, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

Modal and panel contributions

Beginning with NX Nastran 8, the modal contributions capability has been enhanced as follows:

- A new describer, PANELMC, has been added to the MODCON case control command to allow you to select panels for modal contribution output. You can now examine how specific structural or fluid modes associated with a specific region on the structure side of the fluid-structure interface contribute to the dynamic response.

- In NX Nastran 7.1, the PANCON case control command was created to allow you to examine panel contributions to the fluid dynamic response. Now, you can also examine panel contributions to the structural dynamic response, because PANCON automatically supports reciprocal panel contributions.

For additional information, see the updated MODCON case control command and the PANCON case control command.

For a mathematical description of reciprocal panel contributions, see Reciprocal modal contributions mathematical basis.

Reciprocal modal contributions mathematical basis

The mathematical basis for the displacement panel reciprocal modal contributions generated by the PANCON case control command is as follows. The panel reciprocal modal contributions, $[P_R]$, are given by:

$$[P_R] = -[\Phi_s][Z][\Phi_s]^T[A_{panel}][\Phi_f][\xi_f]$$

where $[\Phi_s]$ are the uncoupled, undamped structural modes, $[A_{panel}]$ is a reduced form of the acoustic coupling matrix for specific panels, $[\Phi_f]$ are the uncoupled, undamped, rigid-wall acoustic modes, and $[x_f]$ are the fluid modal amplitudes.

The matrix, $[Z]$, is given by:

$$[Z] = \left[-\omega^2[m_s] + i\omega[b_s] + [k_s]\right]^{-1}$$

where [m_s] is the structural modal mass, [b_s] is the structural modal damping, [k_s] is the structural modal stiffness, and w is the excitation frequency in rad/unit time.

For velocity contributions, $[P_R]$ is multiplied by iw. For acceleration contributions, $[P_R]$ is multiplied by $-w^2$.

MODCON Modal Contribution Request

Requests modal contribution results for residual.

Format:

$$\text{MODCON}\left[\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{REALorIMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix}, \text{PUNCH}\right],$$

$$\begin{bmatrix}\text{ABS}\\ \text{NORM}\\ \text{BOTH}\end{bmatrix}, \text{TOPS} = \text{ps}, \text{TOPF} = \text{pf}, \text{SOLUTION}=\begin{Bmatrix}\text{ALL}\\ \text{setout}\end{Bmatrix},$$

$$\left.\text{PANELMC} = \begin{Bmatrix}\text{NONE}\\ \text{setp}\\ \text{ALL}\end{Bmatrix}\right] = \begin{Bmatrix}\text{n}\\ \text{ALL}\\ \text{NONE}\end{Bmatrix}$$

Examples:

```
MODCON=123
MODCON(SORT1,PHASE,PRINT,PUNCH,BOTH,TOPS=5)=ALL
```

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of modal dof for each frequency or time. (Default)
SORT2	Output will be presented as a tabular listing of frequency or time for each modal dof. This option is not available for SOL 110.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PRINT	The print file (.f06) will be the output medium. (Default)
PUNCH	The standard punch file (.pch) will be the output medium.

Describer	Meaning
NOPRINT	Generates, but does not print, modal contribution results.
ABS	Output modal contributions in absolute terms. (Default)
NORM	Output modal contributions in normalized terms.
BOTH	Output modal contributions in both absolute and normalized terms.
TOPS (or TOP)	The number of structural modes to list in the output that have the greatest contribution to the response at each frequency or time. The output is sorted in descending order from the structural mode having the greatest contribution; *ps* > 0. If *ps* = 0, no structural mode contributions will be output, only totals. (Default is *ps* = 5)
TOPF	The number of fluid modes to list in the output that have the greatest contribution to the response at each frequency or time. The output is sorted in descending order from the fluid mode having the greatest contribution; *pf* > 0. If *pf* = 0, no fluid mode contributions will be output, only totals. (Default is *pf* = 5)
SOLUTION	SOLUTION = ALL (default) requests that modal contribution calculations be performed at all frequencies or times defined by either the FREQUENCY or TSTEP case control commands, respectively. For SOLUTION = *setout*, modal contribution calculations are performed at the frequencies or times specified by a SET case control command having the identification number of *setout*.
PANELMC	Request modal contributions by panels; only applies to acoustic responses and the contributions from structural modes. PANELMC = ALL requests modal contributions from all panels defined by PANEL bulk entries. PANELMC = *setp* requests modal contributions from all panels listed in the SET case control command having the identification number of *setp*. PANELMC = NONE (default) requests that no modal contributions by panels be calculated.
n	Calculate modal contributions for the list defined by the SETMC case control command having set identification number *n*.
ALL	Calculate modal contributions for the lists defined by all SETMC case control commands specified in and above the current subcase.

Describer	Meaning
NONE	Do not calculate modal contributions. This is useful to turn off modal contribution output for a specific subcase.

Remarks:

1. Both PRINT and PUNCH may be requested.

2. MODCON = NONE overrides an overall output request.

3. SOL 110, 111, 112, and 146 are supported. For SOL 110, modal contributions for superelements are not supported. The TOPF and PANELMC keywords are only supported for SOL 111. The SOLUTION keyword is only supported for SOL 111, 112, and 146.

4. Results for SPC forces do not include the effect of any enforced motion applied at the DOF.

5. The parameters LFREQ, LFREQFL, HFREQ, HFREQFL, LMODES, and LMODESFL are supported.

6. The SOLUTION and PANELMC keywords can be abbreviated to SOLU and PANE, respectively.

7. The SET case control command referenced by SOLUTION = *setout* must contain real values for frequencies or times. Using integer values may lead to erroneous results.

PANCON Acoustic Panel Contribution Request

Requests acoustic panel contribution results for residual.

Format:

$$\mathrm{PANCON}\left[\begin{bmatrix}\mathrm{SORT1}\\ \mathrm{SORT2}\end{bmatrix},\begin{bmatrix}\mathrm{REALorIMAG}\\ \mathrm{PHASE}\end{bmatrix},\begin{bmatrix}\mathrm{PRINT}\\ \mathrm{NOPRINT}\end{bmatrix},\mathrm{PUNCH}\right],$$

$$\begin{bmatrix}\mathrm{ABS}\\ \mathrm{NORM}\\ \mathrm{BOTH}\end{bmatrix},\ \mathrm{TOPP}=\mathrm{pp},\ \mathrm{TOPG}=\mathrm{pg},\ \mathrm{SOLUTION}=\begin{Bmatrix}\mathrm{ALL}\\ \mathrm{setf}\end{Bmatrix},$$

$$\left.\mathrm{PANEL}=\begin{Bmatrix}\mathrm{ALL}\\ \mathrm{setp}\\ \mathrm{NONE}\end{Bmatrix},\ \mathrm{GRID}=\begin{Bmatrix}\mathrm{NONE}\\ \mathrm{setg}\\ \mathrm{ALL}\end{Bmatrix}\right]=\begin{Bmatrix}\mathrm{n}\\ \mathrm{ALL}\\ \mathrm{NONE}\end{Bmatrix}$$

Examples:

```
PANCON=123
PANCON(SORT1,PHASE,PRINT,PUNCH,BOTH,TOPP=5)=ALL
```

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of panels or grids for each frequency. (Default)
SORT2	Output will be presented as a tabular listing of frequency for each panel or grid.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PRINT	The print file (.f06) will be the output medium. (Default)
PUNCH	The standard punch file (.pch) will be the output medium.

Describer	Meaning
NOPRINT	Generates, but does not print, contribution results.
ABS	Output contributions in absolute terms. (Default)
NORM	Output contributions in normalized terms.
BOTH	Output contributions in both absolute and normalized terms.
TOPP	The number of structural panels to list in the output that have the greatest contribution to the response at each frequency. The output is sorted in descending order from the structural panel having the greatest contribution; $pp > 0$. If $pp = 0$, no structural panel contributions will be output, only totals. (Default is $pp = 5$)
TOPG	The number of structural grids to list in the output that have the greatest contribution to the response at each frequency. The output is sorted in descending order from the structural grid having the greatest contribution; $pg > 0$. If $pg = 0$, no structural grid contributions will be output, only totals. (Default is $pg = 20$)
SOLUTION	SOLUTION = ALL (default) requests that contribution calculations be performed at all frequencies defined by the FREQUENCY case control commands. For SOLUTION = *setf*, contribution calculations are performed at the frequencies specified by a SET case control command having the identification number of *setf*.
PANEL	Specifies the set of panels to output panel contributions. PANEL = ALL (default) requests that contributions from all panels defined by PANEL bulk entries be output. PANEL = *setp* requests contributions from panels included in the SET case control command having the identification number of *setp*. PANEL = NONE requests that no contributions from panels be output.
GRID	Specifies the set of grids to output contributions. GRID = ALL requests contributions from all structural grids that are part of the acoustic coupling matrix. GRID = *setg* requests contributions from structural grids included in the SET case control command having the identification number of *setg*. Any grids included in *setg* that are not part of the acoustic coupling matrix will be ignored. GRID = NONE (default) requests that no structural grid contributions be output.

Describer	Meaning
n	Calculate panel and/or grid contributions for the list defined by the SETMC case control command having set identification number *n*. Any response listed in SETMC = *n* that is acoustic will generate panel and/or grid contributions as requested. Any response listed in SETMC = *n* that is structural will generate panel reciprocal contributions if panel contributions are requested.
ALL	Calculate panel and/or grid contributions for the lists defined by all SETMC case control commands specified in and above the current subcase. Any response listed in SETMC case control commands that is acoustic will generate panel and/or grid contributions as requested. Any response listed in SETMC case control commands that is structural will generate panel reciprocal contributions if panel contributions are requested.
NONE	Do not calculate panel or grid contributions. This is useful to turn off contribution output for a specific subcase.

Remarks:

1. Both PRINT and PUNCH may be requested.

2. PANCON = NONE overrides an overall output request.

3. SOL 108 and 111 are supported.

4. The parameters LFREQ, LFREQFL, HFREQ, HFREQFL, LMODES, and LMODESFL are supported.

5. The SOLUTION and PANEL keywords can be abbreviated to SOLU and PANE, respectively.

6. The SET case control command referenced by SOLUTION = *setf* must contain real values for frequencies. Using integer values may lead to unintended results.

7. The SET case control command referenced by PANEL = *setp* must contain the alphanumeric name of existing panels defined by PANEL bulk entries.

Selecting modes for SOL 111 and SOL 112

The MODSEL case control command allows you to select modes from a modal solution to use in the response calculations of a modal frequency response analysis

(SOL 111) or a modal transient response analysis (SOL 112). For NX Nastran 8, the MODSEL case control command has been enhanced and now allows you to:

- Exclude a list of modes from the response calculation. To do so, place a negative sign in front of the identification number of the referenced SET case control command. For example, suppose you specify:

  ```
  SET 20 = 8,9,12
  MODSEL(FLUID) = -20
  ```

  All of the modes will be included in the response calculation except fluid mode numbers 8, 9, and 12.

- Include different MODSEL case control commands or combinations of MODSEL case control commands in subcases. This capability allows you to efficiently examine the response due to various combinations of fluid and structural modes in a single job.

- Use MODSEL case control commands in a restart run. This capability allows you to specify the modes from a normal modes analysis (SOL 103) to use in a consecutive response solution.

For additional information, see the updated version of the MODSEL case control command.

MODSEL Mode Selection

Used to select mode numbers to include in a modal dynamic response solution.

Format:

$$\text{MODSEL}\begin{pmatrix}\text{STRUCTURAL}\\ \text{FLUID}\end{pmatrix}=\begin{Bmatrix}\text{ALL}\\ \text{n}\end{Bmatrix}$$

Examples:

```
MODSEL = 3
MODSEL(FLUID) = 4
MODSEL(STRUCTURAL) = ALL
MODSEL = -10
```

Describers:

Describer	Meaning
STRUCTURAL	Specifies the structural modes to include in the response solution. (Default)
FLUID	Specifies the fluid modes to include in the response solution.
ALL	Designates that all the structural or fluid modes be used in the response solution. (Default)
n	Identification number of the SET case control command containing a list of either structural or fluid modes to be used in the response solution. The mode numbers not included in the SET case control command are removed from the modal space. By preceding the identification number of the SET case control command with a negative sign, the mode numbers listed in the SET case control command are omitted from the response solution. Mode numbers listed in the SET case control command that are larger than the number of computed modes are ignored. (Integer≠0)

Remarks:

1. All structural and fluid modes are used in the response solution if a MODSEL case control command is not included in the input file.

2. The use of MODSEL is supported at the subcase level and for restarts.

3. Multiple MODSEL case control commands can be included in any subcase.

4. MOSEL is supported for restarts and at the subcase level for SOLs 111, 112, 145, and 146. MOSEL is supported for restarts, but not at the subcase level for SOLs 103, 110, and 187. For SOLs 103, 110, and 187, MODSEL must be above the subcase level.

Initial conditions for transient analysis

You can calculate the response of a structure to an arbitrary loading using direct transient analysis (SOL 109) or modal transient analysis (SOL 112). You can use the IC case control command to specify initial conditions and whether or not to account for differential stiffness in both SOL 109 and SOL 112.

Prior to NX Nastran 7.1,

- For direct transient analysis (SOL 109):
  - TIC bulk entries defined in physical space were used as the initial condition when you specified the PHYSICAL (default) describer.
  - The results of a static subcase were used as the initial condition when you specified the STATSUB describer.
  - The results of a static subcase were used as the initial condition and the effects of differential stiffness were accounted for during the solve when you specified the DIFFK describer in combination with the STATSUB describer.
- For modal transient analysis (SOL 112):
  - TIC bulk entries defined in physical space were used as the initial condition when you specified the PHYSICAL (default) describer. During the solve, the initial conditions in physical space were converted to modal space.
  - TIC bulk entries defined in modal space were used as the initial condition when you specified the MODAL describer.
  - The results of a static subcase were converted to modal space during the solve and used as the initial condition when you specified the STATSUB describer.
  - The results of a static subcase were converted to modal space during the solve and used as the initial condition, and the effects of differential stiffness were accounted for during the solve when you specified the DIFFK describer in combination with the STATSUB describer.

As a beta capability in NX Nastran 7.1, the IC case control command was enhanced by the addition of the TZERO describer. When you specified the TZERO describer, the static deflection in modal space resulting from the loading at time = 0 was used as the initial condition for a modal transient analysis (SOL 112).

Beginning with NX Nastran 8, the IC case control command beta capability introduced in NX Nastran 7.1 is now fully supported and has been further enhanced. You can now specify:

- IC(TZERO,DIFFK) to account for the effects of differential stiffness in a direct transient analysis (SOL 109) or a modal transient analysis (SOL 112).

- IC(TZERO) to use the static deflection in physical space resulting from the loading at time = 0 as the initial condition for a direct transient analysis (SOL 109).

An example of a complete input file containing the IC case control command with the TZERO describer specified is *ictzero1.dat*. It can be found in *install_dir/nxn8/nast/tpl*.

For additional information, see the updated IC case control command.

IC Transient Initial Condition Set Selection

Selects the initial conditions for transient analyses (SOLs 109, 112, 129, 159, 601, and 701).

Format:

$$\mathrm{IC}\begin{bmatrix}\underline{\mathrm{PHYSICAL}}\\ \mathrm{MODAL}\\ \mathrm{STATSUB[,DIFFK]}\\ \mathrm{TZERO[,DIFFK]}\end{bmatrix} = n$$

Examples:

```
IC = 17
IC(PHYSICAL) = 10
IC(MODAL) = 20
IC(STATSUB) = 30
IC(STATSUB,DIFFK) = 1030
IC(TZERO)
IC(TZERO,DIFFK)
```

Describers:

Describer	Meaning
PHYSICAL	Use the TIC bulk entries selected by set n as the initial conditions for coordinates involving grid and scalar points (default). See Remark 7.
MODAL	Use the TIC bulk entries selected by set n as the initial conditions for modal coordinates. See Remarks 3 and 7.
STATSUB	Use the solution of the static analysis subcase n as the initial conditions. See Remark 4.
TZERO	Use the static deflection resulting from the loading at time = 0 as the initial condition for a direct transient analysis (SOL 109) or a modal transient analysis (SOL 112). For this option, n is not needed. If n is specified, it will be ignored.
DIFFK	Include the effects of differential stiffness in the solution. See Remarks 4 and 5.

Describer	Meaning
n	For the PHYSICAL option, *n* is the set identification number of TIC bulk entries for structural analysis (SOLs 109, 112, 129, 601, and 701) or TEMP and TEMPD bulk entries for heat transfer analysis (SOL 159). For the MODAL option, *n* is the set identification number of TIC bulk entries for modal transient analysis (SOL 112). For the STATSUB option, *n* is the ID of a static analysis subcase (SOL 109 and 112). For the TZERO option, *n* is not needed and will be ignored if specified. (Integer>0)

Remarks:

1. For structural analysis, TIC bulk entries will not be used (therefore, no initial conditions) unless selected in the case control section.

2. Only the PHYSICAL option (the default) may be specified in direct transient analysis (SOL 109), nonlinear or linear transient analysis (SOL 129), heat transfer analysis (SOL 159), advanced implicit nonlinear analysis (SOL 601,N), and advanced explicit nonlinear analysis (SOL 701).

3. IC(MODAL) may be specified only in modal transient analysis (SOL 112).

4. IC(STATSUB) and IC(STATSUB,DIFFK) cannot both be specified in the same execution. They are only applicable to direct transient analysis (SOL 109) and modal transient analysis (SOL 112). They are not applicable in a DMP solution.

5. The DIFFK keyword is meaningful only when used in conjunction with the STATSUB or TZERO keywords.

6. IC(TZERO) and IC(TZERO,DIFFK) cannot both be specified in the same execution. They are only applicable to direct transient analysis (SOL 109) and modal transient analysis (SOL 112). They are not applicable in a DMP solution.

7. Initial condition definitions on extra points are not supported and will be ignored.

8. The IC case control command must be defined in the global subcase. However, when the TZERO keyword is specified with the IC case control command, the initial condition for each subcase will be defined by the loads included in that subcase.

Static condensation

For a SOL 103, 111, or 112 analysis, when component mode synthesis is the superelement reduction method, BSETi (or, equivalently, BNDFIXi) and CSETi (or, equivalently, BNDFREEi) bulk entries are used to define fixed and free DOFs of the analysis set, respectively. Prior to NX Nastran 8, when static condensation was the superelement reduction method, BSETi (or, equivalently, BNDFIXi) and CSETi (or, equivalently, BNDFREEi) bulk entries were ignored because the fixed and free designations are meaningless when using static condensation. To define the analysis set for static condensation, you were required to use ASETi bulk entries.

Beginning with NX Nastran 8, you can define the analysis set for static condensation in a SOL 101 analysis using the BSETi (or, equivalently, BNDFIXi) and CSETi (or, equivalently, BNDFREEi) bulk entries. Now, when static condensation is the superelement reduction method in a SOL 101 analysis, the software automatically assigns the DOFs listed on BSETi (or, equivalently, BNDFIXi) and CSETi (or, equivalently, BNDFREEi) bulk entries to the analysis set.

Multi-body dynamics and control system software interfaces

The multi-body dynamics and control system software interface has the following enhancements.

- The capability of NX Nastran to interface with control system software has been enhanced. Now, when you use the MBDEXPORT case control command to request the generation of a state-space representation for a flexible component, the generated state-space representation includes accelerations by default.

  For an updated mathematical description of state-space representations of a flexible component, see “Mathematical Description”.

- In addition to solution 103, you can now export any of the supported third-party multi-body dynamics and control system files in a solution 111 or 112 run. The export occurs during the modal portion of these solutions.

For more information, see the updated MBDEXPORT case control command and “Multi-body Dynamics and Control System Software Interfaces” in the *Advanced Dynamic Analysis User’s Guide*.

Mathematical Description

NX Nastran supports both the normal mode reduction method and component mode synthesis (a general form for Craig-Bampton reduction). Both reduction methods can produce a standard representation of the equations of motion and a state-space representation of the equations of motion.

Standard Representation using Normal Modes Reduction

The equation of motion for a system can be written as

$$M_{ff}\ddot{u}_f + D_{ff}\dot{u}_f + K_{ff}u_f = P_f$$

Equation 1-1.

where M_{ff}, D_{ff}, K_{ff} are the mass, viscous damping, and stiffness matrices, respectively. P_f is the load vector and u_f is the displacement vector. The subscript f denotes the NX Nastran free set of DOF (f-set), which is the set of DOF that are not restrained or dependent on other DOF. Thus, the subscript denotes size and form of the associated variable and should not be interpreted as an index. A double subscript designates a matrix with the first subscript indicating the number of rows and the second subscript indicating the number of columns. For a vector, the single subscript indicates the number of rows.

The modes of the undamped system can be found by solving the following eigenvalue problem:

$$K_{ff}\Phi_{fh} = M_{ff}\Phi_{fh}\Omega_{hh}$$

Equation 1-2.

where Φ_{fh} is the matrix of mass normalized mode shapes with h representing the number of modes. Ω_{hh} is a diagonal matrix of the eigenvalues where

$$\Omega_{hh} = \begin{bmatrix} \ddots & & 0 \\ & \omega^2 & \\ 0 & & \ddots \end{bmatrix}$$

where w are the natural frequencies.

The response in modal space g$_h$ is found from

$$u_f = \Phi_{fh}\gamma_h$$

Equation 1-3.

Substituting Equation 1–3 into Equation 1–1 and premultiplying by $\Phi^T{}_{fh}$, we obtain

$$m_{hh}\ddot{\gamma}_h + d_{hh}\dot{\gamma}_h + k_{hh}\gamma_h = \Phi^T_{fh}P_f$$

Equation 1-4.

where

$$m_{hh} = \Phi^T_{fh}M_{ff}\Phi_{fh}$$

$$d_{hh} = \Phi^T_{fh}D_{ff}\Phi_{fh}$$

$$k_{hh} = \Phi^T_{fh}K_{ff}\Phi_{fh}$$

are the reduced mass, damping, and stiffness matrices, respectively. Similarly, premultiplying Equation 1–2 by $\Phi^T{}_{fh}$ and rearranging yields

$$\Omega_{hh} = m^{-1}_{hh}k_{hh}$$

State-Space Representation using Normal Modes Reduction

For the state-space representation used in a control system simulation, the output DOF are a subset of the f-set DOF. The output DOF are designated as the j-set for the following calculations. It is possible to use a partition matrix S_{jf} to reduce the response from the full f-set to the smaller j-set as follows:

$$y_j = S_{jf}u_f$$

$$\dot{y}_j = S_{jf}\dot{u}_f$$

$$\ddot{y}_j = S_{jf}\ddot{u}_f$$

Equation 1-5.

In its simplest form, the matrix S_{jf} is a partitioning matrix of 0s and 1s. If some of the output DOF are dependent (that is, in the NX Nastran m-set), the form is slightly more complicated. Similarly, there is a subset of the f-set DOF that is the force input DOF. These DOF are designated here as the i-set. A partition matrix R_{if} relates the f-set P_f force vector to the i-set input P_i force vector as follows:

$$P_f = R^T_{if}P_i$$

Equation 1-6.

To obtain a state-space representation, begin by rearranging Equation 1–4 and inserting Equation 1–6 to obtain:

$$\ddot{\gamma}_h = m_{hh}^{-1}\Phi_{fh}^T R_{if}^T P_i - m_{hh}^{-1} d_{hh}\dot{\gamma}_h - m_{hh}^{-1} k_{hh}\gamma_h$$

Equation 1-7.

For simplicity, assume that the modal damping matrix d_{hh} is given by

$$d_{hh} = 2m_{hh}\omega_{hh}Z_{hh}$$

where Z_{hh} is a diagonal matrix of all the modal damping ratios and w_{hh} is given by

$$\omega_{hh} = \begin{bmatrix} \ddots & & 0 \\ & \omega & \\ 0 & & \ddots \end{bmatrix}$$

where w are the natural frequencies. Because m_{hh} is also a diagonal matrix, d_{hh} is a diagonal matrix.

Now convert the above system of h second order differential equations to a system of $2h$ first order differential equations by creating a state vector x_{2h} which is related to g_h as follows:

$$\xi_{2h} = \begin{Bmatrix} \dot{\gamma}_h \\ \gamma_h \end{Bmatrix} \qquad \dot{\xi}_{2h} = \begin{Bmatrix} \ddot{\gamma}_h \\ \dot{\gamma}_h \end{Bmatrix}$$

Equation 1-8.

Using Equation 1–8, Equation 1–7 can be rewritten as

$$\dot{\xi}_{2h} = a_{2h\times 2h}\xi_{2h} + b_{2h\times i}P_i$$

where

$$a_{2h\times 2h} = \begin{bmatrix} -m_{hh}^{-1}d_{hh} & -m_{hh}^{-1}k_{hh} \\ I_{hh} & 0_{hh} \end{bmatrix} \qquad b_{2h\times i} = \begin{bmatrix} m_{hh}^{-1}\Phi_{fh}^T R_{if}^T \\ 0_{hi} \end{bmatrix}$$

Here the submatrix I_{hh} is the identity matrix and the submatrices 0_{hh} and 0_{hi} are null matrices. The state-space output vector Y_{2j} is computed by substituting Equation 1–3 into Equation 1–5. Comparing the resulting expression to Equation 1–8, one can write

$$Y_{3j} = \begin{Bmatrix} \ddot{y}_j \\ \dot{y}_j \\ y_j \end{Bmatrix} = c_{3j\times 2h}\xi_{2h} + e_{3j\times i}P_i$$

where

$$c_{3j\times 2h} = \begin{bmatrix} -S_{jf}\Phi_{fh}m_{hh}^{-1}d_{hh} & -S_{jf}\Phi_{fh}m_{hh}^{-1}k_{hh} \\ S_{jf}\Phi_{fh} & 0_{jh} \\ 0_{jh} & S_{jf}\Phi_{fh} \end{bmatrix} \qquad e_{3j\times i} = \begin{bmatrix} S_{jf}\Phi_{fh}m_{hh}^{-1}\Phi_{fh}^{T}R_{if}^{T} \\ 0_{jh} \\ 0_{jh} \end{bmatrix}$$

and the submatrix 0_{jh} is a null matrix.

Standard Representation using Component Mode Synthesis (a general form for Craig-Bampton reduction)

Component mode synthesis reduction is the preferred method to use when multiple connected flexible bodies are modeled. In such a case, it is important to account for static stiffness at the connections in order to obtain the correct local stiffness.

To begin, partition Equation 1–1 into an a-set and an o-set. The a-set contains the connection DOF and the o-set contains all other DOF. After partitioning, Equation 1–1 takes the following form:

$$\begin{bmatrix} M_{oo} & M_{oa} \\ M_{ao} & M_{aa} \end{bmatrix}\begin{Bmatrix} \ddot{u}_o \\ \ddot{u}_a \end{Bmatrix} + \begin{bmatrix} D_{oo} & D_{oa} \\ D_{ao} & D_{aa} \end{bmatrix}\begin{Bmatrix} \dot{u}_o \\ \dot{u}_a \end{Bmatrix} + \begin{bmatrix} K_{oo} & K_{oa} \\ K_{ao} & K_{aa} \end{bmatrix}\begin{Bmatrix} u_o \\ u_a \end{Bmatrix} = P_f$$

Equation 1-9.

The constrained normal modes Ψ_{oq} are found by solving

$$K_{oo}\Psi_{oq} = M_{oo}\Psi_{oq}\Omega_{qq}$$

where the subscript q is the number of constrained normal modes.

The constraint modes are found from

$$G_{oa} = -K_{oo}^{-1}K_{oa}$$

The constrained normal modes and the constraint modes are used together in component mode synthesis reduction to relate the response u_f to a reduced response u_z from a combination of physical response u_a and generalized response h_q as follows:

$$u_f = \begin{Bmatrix} u_o \\ u_a \end{Bmatrix} = \begin{bmatrix} \Psi_{oq} & G_{oa} \\ 0_{aq} & I_{aa} \end{bmatrix} \begin{Bmatrix} \eta_q \\ u_a \end{Bmatrix} = T_{fz} \begin{Bmatrix} \eta_q \\ u_a \end{Bmatrix} = T_{fz} u_z$$

Equation 1-10.

where the z-set is the sum of the a-set and the q-set, and T_{fz} is the Craig-Bampton transformation matrix.

Substituting Equation 1–10 into Equation 1–9 and premultiplying by $T^T{}_{fz}$ we get the Craig-Bampton reduced equation of motion

$$M_{zz}\ddot{u}_z + D_{zz}\dot{u}_z + K_{zz}u_z = T_{fz}^T P_f$$

Equation 1-11.

where

$$M_{zz} = T_{fz}^T M_{ff} T_{fz}$$
$$D_{zz} = T_{fz}^T D_{ff} T_{fz}$$
$$K_{zz} = T_{fz}^T K_{ff} T_{fz}$$

The modes of the system are found by solving the following eigenvalue problem:

$$K_{zz}\Phi_{zh} = M_{zz}\Phi_{zh}\Omega_{hh}$$

where Φ_{zh} is the matrix of eigenvectors for the reduced system, Ω_{hh} is the diagonal matrix of natural frequencies squared for the reduced system, and h is the number of modes. When solving this eigenvalue problem, it is important that modal truncation does not occur, which means that the dimension of h and z should be the same. If modal truncation occurs, static residual stiffness effects from the component mode synthesis reduction will be lost.

Now the physical response u_z is transformed to modal response γ_h using

$$u_z = \Phi_{zh}\gamma_h$$

Equation 1-12.

Substituting Equation 1–12 into Equation 1–11 and premultiplying by $\Phi^T{}_{zh}$ gives

$$m_{hh}\ddot{\gamma}_h + d_{hh}\dot{\gamma}_h + k_{hh}\gamma_h = \Phi_{zh}^T T_{fz}^T P_f$$

Equation 1-13.

where

$$m_{hh} = \Phi_{zh}^T M_{zz} \Phi_{zh}$$
$$d_{hh} = \Phi_{zh}^T D_{zz} \Phi_{zh}$$
$$k_{hh} = \Phi_{zh}^T K_{zz} \Phi_{zh}$$

are the reduced mass, damping, and stiffness matrices, respectively. The matrix of eigenvalues is given by

$$\Omega_{hh} = m_{hh}^{-1} k_{hh}$$

State-Space Representation using Component Mode Synthesis (a general form for Craig-Bampton reduction)

To obtain a state-space representation, rearrange Equation 1–13 and insert Equation 1–6 to yield

$$\ddot{\gamma}_h = m_{hh}^{-1}\Phi_{zh}^T T_{fz}^T R_{if}^T P_i - m_{hh}^{-1} d_{hh}\dot{\gamma}_h - m_{hh}^{-1} k_{hh}\gamma_h$$

Equation 1-14.

Equation 1–14 can be put into state-space format using Equation 1–8. As before, the resulting state vector is given by

$$\dot{\xi}_{2h} = a_{2h\times 2h}\xi_{2h} + b_{2h\times i}P_i$$

where

$$a_{2h\times 2h} = \begin{bmatrix} -m_{hh}^{-1} d_{hh} & -m_{hh}^{-1} k_{hh} \\ I_{hh} & 0_{hh} \end{bmatrix} \qquad b_{2h\times i} = \begin{bmatrix} m_{hh}^{-1}\Phi_{zh}^T T_{fz}^T R_{if}^T \\ 0_{hi} \end{bmatrix}$$

The output state-space vector Y_{2j} is related to the state vector x_{2h} by

$$Y_{3j} = \begin{Bmatrix} \ddot{y}_j \\ \dot{y}_j \\ y_j \end{Bmatrix} = c_{3j\times 2h}\xi_{2h} + e_{3j\times i}P_i$$

where

$$c_{3j\times 2h} = \begin{bmatrix} -S_{jf}T_{fz}\Phi_{zh}m_{hh}^{-1}d_{hh} & -S_{jf}T_{fz}\Phi_{zh}m_{hh}^{-1}k_{hh} \\ S_{jf}T_{fz}\Phi_{zh} & 0_{jh} \\ 0_{jh} & S_{jf}T_{fz}\Phi_{zh} \end{bmatrix}$$

and

$$e_{3j\times i} = \begin{bmatrix} S_{jf}T_{fz}\Phi_{zh}m_{hh}^{-1}\Phi_{zh}^{T}T_{fz}^{T}R_{if}^{T} \\ 0_{jh} \\ 0_{jh} \end{bmatrix}$$

Partitioning Matrices

In the most general case, the input and output DOF for a control system can be either a dependent or an independent DOF. For example, the DOF on a center node of an RBE3 spoke are dependent and can be used as an output DOF to get an average output of points connected by the spoke. Similarly, the same node can be used as an input DOF to distribute a load over an area.

The full model DOF size (g-set) can be partitioned for the output DOF (j-set) as follows:

$$y_j = \bar{S}_{jg}u_g$$

Equation 1-15.

The matrix $\bar{S}_{jg}$ is a partitioning matrix populated with 0s and 1s. Further partitioning Equation 1–15 into dependent DOF (m-set) and independent DOF (n-set) gives

$$y_j = \begin{bmatrix} \bar{S}_{jm} & \bar{S}_{jn} \end{bmatrix} \begin{Bmatrix} u_m \\ u_n \end{Bmatrix}$$

Equation 1-16.

The transformation G_{mn} relates the dependent DOF (m-set) to the independent DOF (n-set) as follows:

$$u_m = G_{mn}u_n$$

Utilizing this transformation in Equation 1–16 yields

$$y_j = \left[\overline{S}_{jm} G_{mn} + \overline{S}_{jn}\right] u_n = S_{jn}\, u_n$$

Equation 1-17.

Partitioning the n-set of Equation 1–17 into the free DOF (f-set) and restrained DOF (s-set) gives

$$y_j = \begin{bmatrix} S_{js} & S_{jf} \end{bmatrix} \begin{Bmatrix} u_s \\ u_f \end{Bmatrix}$$

Equation 1-18.

Typically, restrained DOF have zero displacement. Thus, a restrained DOF cannot be an output DOF and Equation 1–18 simplifies to

$$y_j = S_{jf}\, u_f$$

which is identical to Equation 1–5 when

$$S_{jf} = \quad f\text{–}set \;\; partition \;\; of \;\; \left[\overline{S}_{jm} G_{mn} + \overline{S}_{jn}\right]$$

If output DOF are only in the f-set, the partitioning matrix consists of 0s and 1s. If an output DOF is on a dependent DOF (m-set), the G_{mn} transformation matrix is used in the calculation and the partitioning factors may assume values other than 0 and 1.

The full model DOF size (g-set) can be partitioned to the force input DOF (i-set) using

$$P_g = \overline{R}_{ig}^{T} P_i$$

Equation 1-19.

Equation 1–19 can then be partitioned into dependent DOF (m-set) and independent DOF (n-set) as follows:

$$\begin{Bmatrix} P_m \\ \overline{P}_n \end{Bmatrix} = \begin{bmatrix} \overline{R}_{im} & \overline{R}_{in} \end{bmatrix}^T P_i = \begin{Bmatrix} \overline{R}_{im}^T P_i \\ \overline{R}_{in}^T P_i \end{Bmatrix}$$

Equation 1-20.

where bar overscripts indicate that the matrix was a vector prior to transformation. By transforming Equation 1–20, it can be shown that the dependent and independent forces are related as follows:

$$P_n = \overline{P}_n + G_{mn}^T P_m$$

Equation 1-21.

Substituting vectors $\overline{P}_n$ and P_m from Equation 1–20 into Equation 1–21 gives

$$P_n = \left[\overline{R}_{in}^T + G_{mn}^T \overline{R}_{im}^T\right] P_i = R_{in}^T P_i$$

Equation 1-22.

Partitioning the n-set of Equation 1–22 into the free DOF (f-set) and restrained DOF (s-set) gives

$$P_n = \begin{bmatrix} P_s & P_f \end{bmatrix} = \begin{bmatrix} R_{is}^T & R_{if}^T \end{bmatrix} P_i$$

Equation 1-23.

Because input forces are not specified on restrained DOF, a restrained DOF cannot be an input force DOF. Thus, Equation 1–23 simplifies to

$$P_f = R_{if}^T P_i$$

which is identical to Equation 1–6 when

$$R_{if}^T = \quad f\text{–}set \;\; partition \;\; of \;\; \left[\overline{R}_{in}^T + G_{mn}^T \overline{R}_{im}^T\right]$$

If the input DOF are only in the f-set, the partitioning matrix consists of 0s and 1s. If an input DOF is on a dependent DOF (m-set), the G_{mn} transformation matrix is used in the calculation and the partitioning factors may assume values other than 0 and 1.

Damping

In NX Nastran, damping can be defined as physical viscous (for example, CDAMP), physical structural (for example, GE on MAT bulk entries), and user-defined modal damping (for example, SDAMPING). NX Nastran combines the various damping inputs into a single viscous modal damping matrix using the following equation:

$$D_{ff} = D_{ff}^{(1)} + D_{ff}^{(2)} + \frac{G}{w_3} K_{ff} + \Sigma \frac{G_e}{w_4} K_{ff}^{(e)} + \frac{1}{w_4} K4_{ff}^{(2)}$$

Equation 1-24.

where $D^{(1)}{}_{ff}$ is physical damping from elements, $D^{(2)}{}_{ff}$ is physical damping from DMIG matrix input, $K^{(e)}{}_{ff}$ is the elemental stiffness matrix, $K4^{(2)}{}_{ff}$ is structural damping from DMIG input, G and G_e are structural damping coefficients, and w_3 and w_4 are frequency parameters used for converting structural damping to viscous damping.

When performing a modal reduction, Equation 1–24 is reduced to modal DOF using

$$d_{hh} = \Phi_{fh}^{T} D_{ff} \Phi_{fh}$$

Unlike the modal mass and stiffness matrices, the reduced damping matrix is generally not diagonal. If the component mode synthesis method is used, a second transformation is performed.

Often, modal damping from physical contributions is not used. Instead, user-defined modal damping is used. User-defined modal damping is given by

$$d_{hh} = 2 m_{hh} \omega_{hh} Z_{hh}$$

where Z_{hh} is a diagonal matrix of all the modal damping ratios and w$_{hh}$ is given by

$$\omega_{hh} = \begin{bmatrix} \ddots & & 0 \\ & \omega & \\ 0 & & \ddots \end{bmatrix}$$

where w are the natural frequencies. Because m_{hh} is a diagonal matrix, d_{hh} is also a diagonal matrix.

Therefore, the total modal damping from reduced physical and user-defined contributions is given by

$$d_{hh} = \Phi_{fh}^{T} D_{ff} \Phi_{fh} + 2 m_{hh} \omega_{hh} Z_{hh}$$

SOL 111 improved accuracy with modal damping and frequency dependent stiffness

When a model contains modal damping and elements with frequency dependent stiffness, you can now improve the accuracy of modal frequency response results by using the new SDAMPUP parameter. You can request modal damping with the SDAMP case control command and associated TABDMP1 table. You can also define frequency dependent stiffness properties for certain elements, using, for example, the PBUSHT and PELAST property entries. By default, the software computes modal damping only once, from the original modal solution. If modal damping is used together with frequency dependent stiffness, each response calculation uses an updated, frequency dependent modal stiffness, but frequency independent modal damping. This may produce inaccurate results.

If you specify PARAM,SDAMPUP,YES, the software updates the modal damping at each frequency thus improving the accuracy of the results. Although using this parameter improves accuracy, it may also increase the runtime, particularly if the number of modes is large.

Repeated eigenvalues and rigid body modes are not supported with this method. Rigid body modes may be removed either by adjusting F1 on the EIGRL bulk entry or by setting PARAM,LFREQ. If repeated eigenvalues are present, the new method is deactivated automatically and the default method is used instead.

Example

As an example, the performance of four different solution and damping combinations was evaluated using a relatively small model of approximately 3000 DOF that contained one scalar spring element with frequency dependent stiffness specified using a PELAST entry. The frequency response up to 133 Hz was computed using modes with frequencies up to 350 Hz. The results from each solution and damping combination are depicted in Figure 1-1.

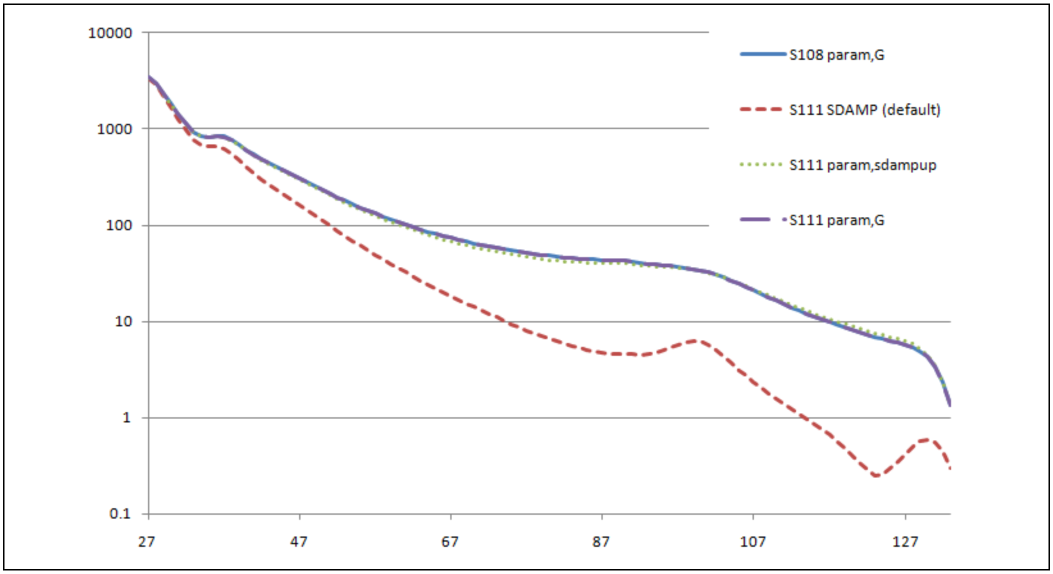


Figure 1-1. Frequency Response Predicted Using Various Combinations of Solutions and Damping.

The four solution and damping combinations were:

- Direct frequency response with damping specified using PARAM,G. (The "S108 param,G" curve in Figure 1-1.)

- Modal frequency response with modal damping specified using the SDAMPING case control command only. (The "S111 SDAMP (default)" curve in Figure 1-1.)

- Modal frequency response with modal damping specified using the SDAMPING case control command and PARAM,SDAMPUP,YES. (The "S111 param,sdampup" curve in Figure 1-1.)

- Modal frequency response with damping specified using PARAM,G. (The "S111 param,G" curve in Figure 1-1.) This solution is a direct solution to the reduced modal representation.

The results from all the solution and damping combinations were essentially identical except for those from the modal solution where PARAM,SDAMPUP,YES was not specified. Once PARAM,SDAMPUP,YES was specified, the modal damping solution was consistent with the others.

The models were run on an IBM Power4/1200MHz machine. By adding the PARAM,SDAMPUP,YES specification to the modal solution, the runtime increased from 0:19 to 0:36. By comparison, the runtime for the direct frequency response solution was 0:58.

Chapter 2 Acoustics

Acoustic source

An acoustic source defined with the ACSRCE bulk entry represents a pulsating sphere in infinite space. The ACSRCE entry, which defines the acoustic source strength as a function of frequency Q(f), references DAREA and TABLEDi entries to define a scale factor A and a power versus frequency table P(f), respectively.

The previous calculation of the source strength

$$\text{Source Strength} = Q(f) = \left[\frac{1}{2\pi f}\sqrt{\frac{8\pi CAP(f)}{\rho}}\right]e^{i(\theta + 2\pi f\tau)}$$

included the scale factor A under the radical. As a result, the scale factor linearly scaled the power P(f), and not the source strength Q(f) directly. This is inconsistent with the other dynamic load inputs defined with the TLOADi and RLOADi entries since the scale factor A on these entries does scale the load directly.

For consistency, the ACSRCE entry scale factor A is now included outside of the radical. The new source strength calculation is

$$\text{Source Strength} = Q(f) = A\left[\frac{1}{2\pi f}\sqrt{\frac{8\pi CP(f)}{\rho}}\right]e^{i(\theta + 2\pi f\tau)}$$

and the scale factor now scales the source strength Q(f) directly.

The new source strength calculation occurs by default, but you can revert to the previous behavior with system cell 503 set to zero. That is,

NASTRAN SYSTEM(503) = 0

See the updated ACSRCE bulk entry.

ACSRCE Acoustic Source Specification

Defines the source strength versus frequency curve for a simple acoustic source.

$$\text{Source Strength} = Q(f) = A\left[\frac{1}{2\pi f}\sqrt{\frac{8\pi C P(f)}{\rho}}\right] e^{i(\theta + 2\pi f \tau)}$$

$$C^2 = B/\rho$$

Format:

1	2	3	4	5	6	7	8	9	10
ACSRCE	SID	EXCITEID	DELAY	DPHASE	TC	RHO	B		

Example:

ACSRCE	103	11			12	1.0	15.0		

Fields:

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of a DAREA or SLOAD entry that defines the scale factor A. (Integer > 0)
DELAY	Identification number of a DELAY entry that defines t. (Integer ≥ 0 or blank)
DPHASE	Identification number of a DPHASE entry that defines q. q should be entered in degrees but is converted to radians during the solution. (Integer ≥ 0 or blank)
TC	Identification number of a TABLEDi entry that defines power versus frequency, P(f). (Integer ≥ 0 or blank)
RHO	Density of the fluid. (Real > 0.0)
B	Bulk modulus of the fluid. (Real > 0.0)

Remarks:

1. Acoustic sources must be selected in the Case Control Section with DLOAD=SID.
2. For additional remarks, see the RLOAD1 entry description.
3. SID must be unique for all ACSRSE, RLOADi, TLOADi, and SELOAD entries.
4. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.

SFE AKUSMOD fluid-structural coupling interface

The fluid-structural coupling created by SFE AKUSMOD can now be imported into a NX Nastran dynamic analysis.

To request the import of the AKUSMOD coupling file, the SFEF70 describer on the FLSTCNT case control command must be set to YES. For example,

FLSTCNT SFEF70=YES

When SFEF70=YES, NX Nastran does not compute the coupling, and instead uses the coupling definition from the external file. When SFEF70=NO (default), the external coupling is not imported.

NX Nastran expects the AKUSMOD file in the same directory where the job is being run, and expects the file name to be fort.70. An ASSIGN statement which uses UNIT=70 must be defined in the file management section of your input file if the coupling file is not named fort.70 or if it is in a location other than where the job is run. For example,

ASSIGN OUTPUT2='/directory_path/user_file_name.70' UNIT=70

See the updated FLSTCNT case control command.

FLSTCNT Control Parameters for Fluid-Structure Interaction

Miscellaneous control parameters for fluid-structure interaction.

Format:

$$\mathrm{FLSTCNT}\left[\mathrm{ACSYM}=\begin{Bmatrix}\boxed{\mathrm{YES}}\\ \mathrm{NO}\end{Bmatrix}\right],\left[\mathrm{ACOUT}=\begin{Bmatrix}\boxed{\mathrm{PEAK}}\\ \mathrm{RMS}\end{Bmatrix}\right],\left[\mathrm{ASCOUP}=\begin{Bmatrix}\boxed{\mathrm{YES}}\\ \mathrm{NO}\end{Bmatrix}\right],$$

$$\left[\mathrm{PREFDB}=\begin{Bmatrix}\boxed{1.0}\\ P\end{Bmatrix}\right],\left[\mathrm{SKINOUT}=\begin{Bmatrix}\boxed{\mathrm{NONE}}\\ \mathrm{PUNCH}\\ \mathrm{FREEFACE}\\ \mathrm{STOP}\end{Bmatrix}\right],\left[\mathrm{AGGPCH}=\begin{Bmatrix}\boxed{\mathrm{NO}}\\ \mathrm{YES}\end{Bmatrix}\right],$$

$$\left[\mathrm{SFEF70}=\begin{Bmatrix}\boxed{\mathrm{NO}}\\ \mathrm{YES}\end{Bmatrix}\right],$$

Examples:

```
FLSTCNT ACOUT=RMS PREFDB=1.0E-06
```

Describers:

Describer	Meaning
ACSYM	Symmetric (YES) or non-symmetric (NO) solution request for fluid-structure analysis. Default=YES.
ACOUT	PEAK or RMS output with the FORCE Case Control request. Default=PEAK.
ASCOUP	Requests a fluid-structure coupled solution when YES (default). When NO, the fluid and the structure are decoupled.
PREFDB	Specifies the reference pressure. Default=1.0.
SKINOUT	Specifies if a debug file with the pairing information is output. Default=NONE. If SKINOUT=PUNCH, a debug data deck is written out along with a debug punch file. SKINOUT=STOP works like the PUNCH option, but will stop immediately after the debug files are created. If SKINOUT=FREEFACE, a debug data deck is written out. See Remark 2.

Describer	Meaning
AGGPCH	Requests the output of the fluid-structure coupling matrix AGG to the punch file when YES. When NO (default), the coupling matrix is not written.
SFEF70	Requests the import of a fluid-structure coupling matrix created by SFE AKUSMOD when YES. When NO (default), the coupling matrix is not imported.
NONE	No debug deck or pairing information is generated.
PUNCH	Debug pairing information file is created. The name of the files will be the base name of the deck appended with "_acdbg.dat" and "_acdbg.pch".
FREEFACE	All free faces are written into a debug file.
STOP	The run is terminated as soon as the debug file with pairing information is created.

Remarks:

1. All the entries specified in this case control statements are available as PARAMETER statements.

2. When SKINOUT=PUNCH, both a punch file and a debug data deck are created. The punch file (*.pch) contains a list of the original structural and fluid element ID's which participated in the coupling. The debug data deck (*.dat) contains
   – dummy shell elements representing the *coupled* structural free faces, and are assigned to a dummy pshell with id=1.
   – dummy shell elements representing the *coupled* fluid free faces, and are assigned to a dummy pshell with id=2.

   When SKINOUT=FREEFACE, a debug data deck is created (no punch file) containing
   – dummy shell elements representing the *coupled* structural free faces, and are assigned to a dummy pshell property with id=1.
   – dummy shell elements representing the *coupled* fluid free faces, and are assigned to a dummy pshell property with id=2.
   – dummy shell elements representing the *uncoupled* structural free faces, and are assigned to a dummy pshell property with id=3.
   – dummy shell elements representing the *uncoupled* fluid free faces, and are assigned to a dummy pshell property with id=4.

3. When SFEF70=YES, NX Nastran does not compute the coupling, and instead uses the SFE AKUSMOD coupling definition from the external file. NX Nastran expects the AKUSMOD file in the same directory where the job is being run, and expects the file name to be fort.70. An ASSIGN statement which uses UNIT=70 must be defined in the file management section of your input file if the coupling file is not named fort.70 or if it is in a location other than where the job is run. For example,

   ASSIGN OUTPUT2='/directory_path/user_file_name.70' UNIT=70

Fluid-structural coupling punch output

You can now request the output of the fluid-structure coupling matrix AGG to the punch file with the new AGGPCH describer on the FLSTCNT case control command.

For example,

FLSTCNT AGGPCH=YES

will write the AGG matrix to the punch file.

You can then import and optionally scale this AGG matrix in a consecutive solution that includes the same model definition at the coupled interface. Use the A2GG case control command and the ASCOUP describer on the FLSTCNT case control command to request this import.

See the updated FLSTCNT case control command.

Chapter 3 External Superelements

Finite element geometry output

The new GEOM describer is available on the EXTSEOUT case control command. You can use this describer to request the software to include the full finite element geometry with the external superelement output. The full finite element geometry output can be used by pre and post processors for data recovery and post processing on the full FE model.

When you include the GEOM describer, the software will write the geometry data blocks GEOM1EXA, GEOM2EXA, and GEOM4EXA with the external superelement. The new describer is supported for the MATDB (or MATRIXDB), DMIGDB, and DMIGOP2 storage options.

See the updated EXTSEOUT case control command.

EXTSEOUT External Superelement Creation Specification

Specify the various requirements for the creation of an external superelement.

Format:

$$
\mathrm{EXTSEOUT}\left[\left(\left[\mathrm{STIFFNESS},\ \mathrm{MASS},\ \mathrm{DAMPING},\ \mathrm{K4DAMP},\ \mathrm{LOADS},\ \mathrm{GEOM},\ \mathrm{ASMBULK},\right.\right.\right.
$$

$$
\left.\left.\left.\mathrm{EXTBULK},\ \mathrm{EXTID}=\mathrm{seid},\ \mathrm{DMIGSFIX}=\left\{\begin{matrix} cccccc \\ EXTID \end{matrix}\right\}\right]\left[\begin{matrix}\mathrm{MATDB(or\,MATRIXDB)} \\ \mathrm{DMIGDB} \\ \mathrm{DMIGOP2=unit} \\ \mathrm{DMIGPCH} \\ \mathrm{MATOP4(or\,MATRIXOP4)=unit}\end{matrix}\right]\right)\right]
$$

Examples:

```
EXTSEOUT
EXTSEOUT(ASMBULK,EXTID=100)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=200)
EXTSEOUT(EXTBULK,EXTID=300)
EXTSEOUT(DMIGDB)
EXTSEOUT(ASMBULK,EXTID=400,DMIGOP2=21)
EXTSEOUT(EXTID=500,DMIGPCH)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=500,DMIGSFIX=XSE500,DMIGPCH)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=500,DMIGSFIX=EXTID,DMIGPCH)
EXTSEOUT(STIF,MASS,DAMP,EXTID=600,ASMBULK,EXTBULK,MATDB)
EXTSEOUT(STIF,MASS,DAMP,GEOM,EXTID=600)
```

Describers:

Describer	Meaning
STIFFNESS	Store the boundary stiffness matrix. See Remarks 1 and 2.
MASS	Store the boundary mass matrix. See Remark 1.
DAMPING	Store the boundary viscous damping matrix. See Remarks 1 and 2.
K4DAMP	Store the boundary structural damping matrix. See Remarks 1 and 2.
LOADS	Store the loads matrix and associated DTI,SELOAD entries. See Remarks 1, 2, and 21.
GEOM	Store geometry. See Remark 20.

Describer	Meaning
ASMBULK	Generate bulk entries related to the subsequent superelement assembly process and store them on the assembly punch file (.asm). This data is to be included in the main bulk portion of the subsequent assembly solution. See Remarks 4 and 13.
EXTBULK	Generate and store bulk entries for the external superelement on the standard punch file (.pch) when used in combination with one of either MATDB, DMIGDB, or DMIGOP2. This data is used in the BEGIN SUPER portion of the bulk section of the subsequent assembly solution. EXTBULK is ignored if either DMIGPCH or MATOP4 is specified. If EXTBULK is not specified, the subsequent assembly solution retrieves the required data for the external superelement from the medium on which the boundary matrices are stored. See Remarks 5 and 6.
EXTID = *seid*	*seid* (integer>0) is the superelement ID to be used in the SEBULK and SECONCT bulk entries stored on the assembly punch file (.asm) if ASMBULK is specified and in the BEGIN SUPER bulk entry stored on the standard punch file (.pch) if DMIGPCH or MATOP4 is specified. See Remarks 3, 4, 5, and 7.
DMIGSFIX = *cccccc*	*cccccc* is the suffix (up to six characters and must not = any EXTSEOUT keyword) that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH keyword is specified. See Remarks 8 through 11.
DMIGSFIX = EXTID	The *seid* defined by the EXTID keyword is the suffix that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH keyword is specified. See Remarks 8 through 11.
MATDB (or MATRIXDB)	Store the boundary matrices and other information on the database (default).
DMIGDB	Similar to MATDB (or MATRIXDB) except that the boundary matrices are stored as DMIG bulk entries on the database.
DMIGOP2 = *unit*	Store the boundary matrices as DMIG bulk entries on an OUTPUT2 file whose Fortran unit number if given by *unit* (integer>0). See Remark 14.

Describer	Meaning
DMIGPCH	Store the boundary matrices as DMIG bulk entries on the standard punch file (.pch). See Remarks 6 through 13.
MATOP4 = *unit* (or MATRIXOP4 = *unit*)	Store the boundary matrices on an OP4 file whose Fortran unit number is given by *unit* (Integer>0). See Remarks 3, 5, 6, 13, and 14.

Remarks:

1. If none of the describers STIFFNESS, MASS, DAMPING, K4DAMP, and LOADS are specified, then all of the boundary matrices are stored by default. If any subset of the describers STIFFNESS, MASS, DAMPING, K4DAMP, and LOADS are specified, then only the boundary matrices specified are stored.

2. STIFFNESS, DAMPING, K4DAMP, and LOADS may be abbreviated to STIF, DAMP, K4DA, and LOAD, respectively.

3. EXTID and an *seid* value must be specified if one or more of ASMBULK, EXTBULK, DMIGPCH, or MATOP4 are specified. If the DMIGSFIX=EXTID form is employed along with the DMIGPCH keyword, the value *seid* may not exceed 999999, since this value becomes part of the names given to the DMIG matrices generated on the standard punch file (.pch). See Remark 11.

4. If ASMBULK is specified, the following bulk entries are generated and stored on the assembly punch file (.asm):

   SEBULK *seid* ...

   SECONCT *seid* ...

   GRID entries for the boundary points

   CORD2x entries associated with the above GRID entries

5. If DMIGPCH is not specified, but MATOP4 or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) is specified, the following bulk entries are generated and stored on the standard punch file (.pch):

   BEGIN SUPER *seid*

   GRID entries for the boundary points

   GRID entries for the interior points referenced by PLOTEL entries

CORD2x entries associated with the above GRID entries

EXTRN

ASET/ASET1

QSET/QSET1

SPOINT

PLOTEL

6. If DMIGPCH or MATOP4 is specified, then EXTBULK is ignored even if it is specified.

7. If DMIGPCH is specified, the following bulk entries are generated and stored on the standard punch file (.pch):

   BEGIN SUPER seid

   GRID entries for the boundary points

   CORD2x entries associated with the above GRID entries

   ASET/ASET1

   SPOINT

   DMIG entries for the requested boundary matrices

8. The DMIGSFIX keyword is ignored if DMIGPCH is not specified.

9. If DMIGPCH is specified without the DMIGSFIX keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

   KAAX (boundary stiffness matrix)

   MAAX (boundary mass matrix)

   BAAX (boundary viscous damping matrix)

   K4AAX (boundary structural damping matrix)

   PAX (boundary load matrix)

10. If the DMIGSFIX = *cccccc* form is employed along with the DMIGPCH keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

K*cccccc* (boundary stiffness matrix)

M*cccccc* (boundary mass matrix)

B*cccccc* (boundary viscous damping matrix)

K4*cccccc* (boundary structural damping matrix)

P*cccccc* (boundary load matrix)

11. If the DMIGSFIX = EXTID form is employed along with the DMIGPCH keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

    K*seid* (boundary stiffness matrix)

    M*seid* (boundary mass matrix)

    B*seid* (boundary viscous damping matrix)

    K4*seid* (boundary structural damping matrix)

    P*seid* (boundary load matrix)

12. If the DMIGPCH option is specified, the boundary DMIG matrices generated and stored on the standard punch file (.pch) may not be as accurate as the boundary matrices resulting from other options (MATDB/MATRIXDB or DMIGOP2 or MATOP4/MATRIXOP4). Accordingly, this may result in decreased accuracy from the subsequent assembly job utilizing these DMIG matrices.

13. The punch output resulting from EXTSEOUT usage is determined by ASMBULK, EXTBULK, DMIGPCH, and MATOP4 as follows:

    - No ASMBULK, EXTBULK, DMIGPCH, or MATOP4 results in no punch output.

    - ASMBULK, but no DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output being generated and stored on the assembly punch file (.asm). See Remark 4.

    - No ASMBULK, but DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output being generated and stored on the standard punch file (.pch). See Remarks 5 or 7, as appropriate.

    - ASMBULK and DMIGPCH, MATOP4, or EXTBULK (in combination with MATDB, DMIGDB, or DMIGOP2) results in punch output consisting of two distinct and separate parts. One part is generated and stored on the assembly punch file (.asm) as indicated in Remark 4. The other part is generated and stored on the standard punch file (.pch) as indicated in Remark 5 or 7, as appropriate.

14. If DMIGOP2=*unit* or MATOP4=*unit* is specified, an appropriate ASSIGN OUTPUT2 or ASSIGN OUTPUT4 statement must be present in the File Management Section (FMS) for the *unit*.

15. The creation of an external superelement using EXTSEOUT involves running a non-superelement NX Nastran job, with the following additional data:

    - The data for the creation of the external superelement is specified by the EXTSEOUT case control command, which must appear above the subcase level. Appearing above the subcase level implies that this capability does not support the inclusion of differential stiffness for the external superelement.

    - The boundary points of the external superelement are specified by ASET/ASET1 bulk entries.

    - If the creation involves component mode reduction, the required generalized coordinates are specified using QSET/QSET1 bulk entries. The boundary data for the component mode reduction may be specified using the BNDFIX/BNDFIX1 and BNDFREE/BNDFREE1 bulk entries (or their equivalent BSET/BSET1 and CSET/CSET1 bulk entries). (The default scenario assumes that all boundary points are fixed for the component mode reduction.)

    - The output for the external superelement is generated in the assembly job. This output consists of displacements, velocities, accelerations, SPC forces, MPC forces, grid point force balances, stresses, strains, and element forces. However, in order for this output to be generated in the assembly job, the output requests must be specified in the external superelement creation run. Normally, the only output requests for the external superelement that are honored in the assembly job are those that are specified in the creation run. There is, however, one important exception to this: the displacement, velocity, acceleration, SPC forces, and MPC forces output for the boundary grid points as well as for all grid points associated with PLOTEL bulk entries can be obtained in the assembly job *even if there is no output request specified for these points in the creation run*.

    - If the assembly job involves the use of PARAM bulk entries, then the following points should be noted:

        o PARAM entries specified in the main bulk portion of the input data apply *only to the residual and not to the external superelement*.

- o PARAM entries specified in the BEGIN SUPER portion of the bulk section for an external superelement apply *only to the superelement*.

- o The most convenient way of ensuring that PARAM entries apply not only to the residual but also to all external superelements is to specify such PARAM entries in Case Control, not in the main bulk section. This is particularly relevant for such PARAMs as POST.

16. Output transformation matrices (OTMs) are generated for the following outputs requested in the in external superelement run with EXTSEOUT:

    - DISPLACEMENT
    - VELOCITY
    - ACCELERATION
    - SPCFORCE
    - MPCFORCES
    - GPFORCE
    - STRESS
    - STRAIN
    - FORCE

    Only these external superelement results can be output in the system analysis run. PARAM,OMID,YES is not applicable to the OTMs.

17. If a PARAM,EXTOUT or PARAM,EXTUNIT also exist, they will be ignored. The existence of the EXTSEOUT case control command takes precedence over PARAM,EXTOUT and PARAM,EXTUNIT.

18. This capability is enabled in SOLs 101, 103, 105, 107-112, 114, 115, 118, 129, 144-146, 159, 187, and 200. This capability is not enabled for thermal analyses. For SOLs 101, 103, 111, and 112, this capability will create the external superelement and then perform the user requested analysis. For the other solution sequences, this capability will only create the external superelement. Once the external superelement is created, no other analyses will be performed. Superelement results can be recovered in the second step (i.e. superelement assembly, analysis, and data recovery) for SOLs 101, 103, 105, 107-112, and 144-146.

19. The run creating the external superelement using this capability is not a superelement run. No superelement designations are allowed (i.e. SUPER, SEALL, SESET, BEGIN SUPER, etc.).

20. The GEOM describer will output geometry data blocks GEOM1EXA, GEOM2EXA, and GEOM4EXA containing all of the external superelement geometry to support post-processing. This describer only works for the MATDB (or MATRIXDB), DMIGDB, and DMIGOP2 storage options. By default, the full geometry will not be exported; the GEOM describer must be explicitly defined to have these geometry data blocks written.

21. The LOADS describer will output load information in the $[P_a]$ matrix along with associated DTI,SELOAD bulk entries for each load represented in the $[P_a]$ matrix. DTI,SELOAD bulk entries will not be output for the DMIGPCH option; the use of the DMIGPCH option requires the use of the P2G case control command in the system analysis in order to access the load information defined in the $[P_a]$ matrix that is stored in DMIG format. Thermal loads and enforced motion loads using the SPCD bulk data definition method are not supported. The load information and DTI,SELOAD bulk entries that are output depend on the method in which loads are defined and referenced. The creation run must be a dynamic solution, for example, SOL 103. The following conditions apply.

    - In a creation run, a column in the $[P_a]$ matrix is created for each load (not enforced motion) defined on RLOADi and TLOADi bulk entries whether or not they are referenced in the case control. The corresponding LIDSE and EXCSE on the DTI,SELOAD bulk entries will both be the value of EXCITEID on the RLOADi or TLOADi bulk entry.

      To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

    - In a creation run, the LOADSET = n case control command can be used to create multiple loads. Each definition of an LSEQ bulk entry will create a column in the [Pa] matrix. A LSEQ bulk entry must be referenced by a LOADSET case control command in order to generate a column in the $[P_a]$ matrix. The corresponding LIDSE and EXCSE on an DTI,SELOAD bulk entry will be the value of LID and EXCITEID, respectively, on an LSEQ bulk entry.

      To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

External superelement loads

A new method is available for storing loads with an external superelement. When you include an external superelement created using the new method in a system run, you can optionally select and scale the individual loads.

In the external superelement creation run, the new LOAD describer is available on the EXTSEOUT case control command. Use this describer to request the new load storage method. When the LOAD describer is included, NX Nastran

- Writes the load matrix $[P_a]$ into the output format you defined on the EXTSEOUT command.

- Organizes the load matrix $[P_a]$ such that each column corresponds to a unique load ID.

- Writes the new DTI,SELOAD bulk entry into the output format you defined on the EXTSEOUT command. Each row in the DTI,SELOAD entry corresponds to a unique load ID. This entry maps the original load ID's to the loads in the $[P_a]$ matrix.

When you include an external superelement created using the new method in a system solution, you can include the new SELOAD bulk entry in the residual structure portion of your input file to select the external superelement loads. The input format for the SELOAD entry is

1	2	3	4	5	6	7	8	9	10
SELOAD	LIDS0	SEID	LIDSE						

where LIDS0 is the load ID used in the residual structure, SEID is the external superelement ID, and LIDSE is the original load ID stored in the $[P_a]$ matrix. LIDS0 must be referenced by a loading request to be active. For example, LIDS0 can match the value of *n* on a LOAD case control command, or match the EXCITEID on a RLOADi bulk entry.

Creating and selecting the load matrix $[P_a]$

The load information stored in the load matrix and on the DTI,SELOAD bulk entry depends on the method in which loads are defined and referenced in the superelement creation run. The creation run must be a dynamic solution, for example, SOL 103. The following conditions apply.

- In a creation run, a column in the $[P_a]$ matrix is created for each load (not enforced motion) defined on RLOADi and TLOADi bulk entries whether or not they are referenced in the case control. The corresponding LIDSE and EXCSE on the DTI,SELOAD bulk entries will both be the value of EXCITEID on the RLOADi or TLOADi bulk entry.

To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

- In a creation run, the LOADSET = n case control command can be used to create multiple loads. Each definition of an LSEQ bulk entry will create a column in the [Pa] matrix. A LSEQ bulk entry must be referenced by a LOADSET case control command in order to generate a column in the [P_a] matrix. The corresponding LIDSE and EXCSE on an DTI,SELOAD bulk entry will be the value of LID and EXCITEID, respectively, on an LSEQ bulk entry.

  To select these loads in a system run, the LIDSE field on the SELOAD entry should equal the value of an EXCITEID from the creation run.

Limitations

- The new method does not support SPCD enforced motion loads or thermal loads.

- The DMIGPCH output option on the EXTSEOUT command does not support the new method. A load matrix [P_a] is written when the DMIGPCH output option is used, but only for static loads and without any unique load identifiers. The resulting static load matrix can be selected in a static system run by setting the P2G case control command equal to “PA”. For example,

  P2G = PA

See the updated EXTSEOUT case control command, the new DTI,SELOAD bulk entry, and the new SELOAD bulk entry.

DTI,SELOAD External Superelement Load Set IDs

Usage overrides automatic generation of DTI,SELOAD definitions created through the use of the EXTSEOUT case control command.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SELOAD	LIDSE	EXCSE	“0”	DESCRIPT	DESCRIPT (cont.)	DESCRIPT (cont.)	DESCRIPT (cont.)	
	-etc.-	“ENDREC”							

Example:

DTI	SELOAD	1	100	0	FORCE_AT	_GRID_2	ENDREC	ELEM_100	

Fields:

Field	Contents
LIDSE	Load set identification number in the external superelement. See Remark 2. (Integer > 0)
EXCSE	Excitation identification number in the external superelement. See Remark 2. (Integer > 0)
DESCRIPT	Alphanumeric description of the loading. The description continues until ENDREC is encountered in a field. The description must be defined such that the first non-blank character in each field is not numeric. If the first non-blank character in a field is numeric, the field may be interpreted as numeric. If the field then contains any alphabetic character, an error will be reported. If the data in the field is all numeric, it will be shifted to the right. Imbedded blanks within a field may produce an error. To avoid this problem, use underscores to represent blank characters. (Character)

Remarks:

1. External superelement loads are not supported in static solutions.

2. The values of LIDSE and EXCSE depend on the describers specified on the EXTSEOUT case control command in the external superelement creation run. The creation run must be a dynamic solution, for example, SOL 103.

   - If the loading is referenced in the case control using the LOAD case control command, LIDSE and EXCSE are both equal to the set identification number of the LOAD case control command. If the loading is referenced in the case control using DTI,SELOAD 93 External Superelement Load Set IDs the LOADSET case control command, LIDSE and EXCSE are the values corresponding to the values of LID and EXCITEID, respectively, on each LSEQ bulk entry referenced by the LOADSET case control command. If both LOAD and LOADSET case control commands exist, LOADSET takes precedence.

- LIDSE and EXCSE are both equal to the EXCITEID value for each load (not enforced motion) defined on RLOADi and TLOADi bulk entries, regardless of whether or not they are referenced in the case control.

- Thermal loads and enforced motion loads defined using the SPCD bulk data method are not supported.

3. The use of DTI,SELOAD is optional. If a DTI,SELOAD bulk entry does not exist for an external superelement load set (LIDSE), one will be automatically generated. The description (DESCRIPT) of the automatically generated entry will have the form “SE_LOAD_xx”, where “xx” is the character representation of the external superelement load set.

4. DTI,SELOAD can be considered for use when the EXTSEOUT case control command specifies one of the following data storage options: MATDB (or MATRIXDB), DMIGDB, DMIGOP2 = unit, or MATOP4 (or MATRIXOP4) = unit. The data storage option DMIGPCH is only applicable to static analysis and requires the use of the P2G case control command. See the EXTSEOUT case control command for additional information.

SELOAD External Superelement Load Mapping to Residual

Maps loads from an external superelement to a specified load set for the residual structure.

Format:

1	2	3	4	5	6	7	8	9	10
SELOAD	LIDS0	SEID	LIDSE						

Example:

SELOAD	10010	100	10						

Fields:

Field	Contents
LIDS0	Identification number of the mapped load set to be used in the residual structure. (Integer > 0)
SEID	Partitioned identification number of the partitioned external superelement representing the external superelement. (Integer > 0)
LIDSE	Load set identification number used in the external superelement. (Integer > 0)

Remarks:

1. SELOAD can only be specified in the main bulk section and is ignored after the BEGIN SUPER = *n* command. Because this load is applied at the boundary grids of the superelement, the superelement boundary grids must be connected (via the SECONCT bulk data entry for example) directly to a residual grid. If this is not done, the superelement loading cannot be properly mapped to the residual and an error will be issued (e.g. user fatal message 2008; load set 2 references undefined grid point 201).

2. SELOAD only works if PART superelements (BEGIN SUPER) exist that represent external superelements created using the EXTSEOUT case control command.

3. LIDSE is the identification number of a load set defined in the external superelement creation run using the EXTSEOUT case control command.

4. Multiple SELOAD commands can be used for the same SEID, but a specific LIDSE can only be mapped once.

5. LIDS0 must be unique for the residual structure.

6. The mapped load sets LIDS0s can be referenced by other load bulk entries in the system solution. Examples of load bulk entries include LOAD, LSEQ, TLOADi, and RLOADi.

Chapter

4 Element enhancements

Solid element support in SOL 106 and SOL 129

Beginning with NX Nastran 8, the material nonlinear capabilities of solid elements has expanded.

- Parabolic (variable noded) versions of the CHEXA and CPENTA elements now support plasticity, nonlinear elasticity, and creep.
- The linear and parabolic (variable noded) versions of the CPYRAM element now support plasticity, nonlinear elasticity, and creep.

The following table summarizes the nonlinear capabilities of solid elements in SOL 106 and SOL 129.

Table 4-1. SOL 106 and 129 Solid Element Support

Element	Number of Grids	Geometric Nonlinear Conditions	Material Nonlinear Conditions	
			Plasticity, Nonlinear Elastic, Creep	Hyperelasticity
CHEXA	8	X	X	X
	9 – 20	X	X (new)	X
CTETRA	4	X	X	X
	5 – 10	X	X	X
CPENTA	6	X	X	X
	7 – 15	X	X (new)	X
CPYRAM	5	X	X (new)	
	6 – 13	X	X (new)	

For additional information, see the updated CPYRAM bulk entry.

CPYRAM Five-Sided Solid Element Connection

Defines the connection of the five-sided solid element with five to thirteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPYRAM	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

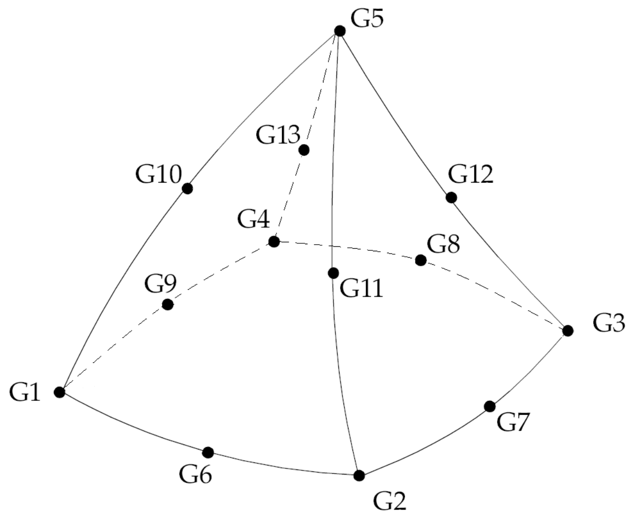


Figure 4-1. CPYRAM Element Connection

Example:

CPYRAM	111	203	31	32	33	34	35	36	
	37	38	39	40	41	42	43		

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSOLID or PLSOLID (SOL 601 only) entry. (Integer > 0; Default=EID)

Field	Contents
Gi	Grid point identification numbers of connected points. (Unique integers > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. The other four faces are triangles. G5 is the vertex and must be opposite with the quadrilateral face.
3. The edge points G6 to G13 are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.
4. Components of stress are output in the material coordinate system.
5. The CPYRAM element coordinate system is the same as the basic coordinate system.
6. It is recommended that the edge grids be located within the middle third of the edge.
7. Only h-version formulation is available; p-version formulation is not supported.
8. The CPYRAM element does not support hyperelasticity in SOL 106 and 129.
9. By default, all eight edges of the element are considered straight unless any of G6 through G13 is specified.

Remarks related to SOLs 601 and 701:

1. For SOL 701, only elements with 5 grid points are allowed.
2. For SOL 601, 13-node CPYRAM elements may be converted to 14-node CPYRAM elements (1 additional node on the centroid of the quadrilateral face of the element) by specifying ELCV=1 in the NXSTRAT entry. 14-node CPYRAM elements are more effective than 13-node CPYRAM for analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson’s ratio close to 0.5.

Total strain output in SOLs 106 and 129

When you used SOL 106 and SOL 129 in versions prior to NX Nastran 8, the software reported only mechanical strain values for the total strain reported under the label “NONLINEAR STRESSES”. Beginning with NX Nastran 8, the software reports the sum of the mechanical strain and thermal strain.

You can override this new behavior and revert to the pre-NX Nastran 8 behavior by including:

```
NASTRAN SYSTEM(500) = 1
```

in the input file.

ELSUM case control command enhancements

The ELSUM case control command is used to request output of an element property summary. Prior to NX Nastran 8, the element property summary could only be grouped according to element type. Beginning with NX Nastran 8, you can request that the element property summary:

- Be grouped according to property type.
- Include only mass totals for the element type or property type grouping.
- Include non-structural mass in the mass totals for the element type or property type grouping.

The new functionality for the ELSUM command is controlled by specifying one or more of the following describers: EID, PID, BOTH, EIDSUM, PIDSUM, SUMMARY, and NSMCONT.

Also beginning with NX Nastran 8, the ELSUM case control command is supported in SOLs 601 and 701.

For more information, see the updated ELSUM case control command.

ELSUM Element Summary Output Request

Requests output of an element property summary.

Format:

$$\text{ELSUM}[(\text{EID}, \text{PID}, \text{BOTH}, \text{EIDSUM}, \text{PIDSUM}, \text{SUMMARY}, \text{NSMCONT})] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

Examples:

```
ELSUM = ALL
ELSUM(PID) = 9
ELSUM(EIDSUM) = ALL
```

Describers:

Describer	Meaning
EID	Group element summary output by element type. (Default)
PID	Group element summary output by element property type.
BOTH	Output element summaries for both the EID and PID groupings.
EIDSUM	Output only a summary of mass totals for the EID grouping.
PIDSUM	Output only a summary of mass totals for the PID grouping.
SUMMARY	Output only a summary of mass totals for both the EID and PID groupings.
NSMCONT	Add non-structural mass from NSM, NSM1, NSML, and NMSL1 bulk entries to the EID and/or PID element summaries.
ALL	Element summary output for all elements.
n	Set identification of a previously appearing SET command. Only produces output for elements whose identification numbers appear on this SET command. (Integer>0)
NONE	No element summary output.

Remarks:

1. The ELSUM command is ignored in heat transfer solutions.

2. ELSUM output is only available in the F06 file. PUNCH and PLOT options are not supported.

3. The ELSUM command produces a summary of properties for elements. The properties include element-id, material-id, length or thickness, area, volume, structural mass, non-structural mass, total mass, and the product of the total mass and the WTMASS parameter. Total mass is the sum of structural and non-structural mass.

4. Certain element types produce only partial data. For these element types, output will contain element-id, length or thickness, volume, and area without mass data. Totals will thus not include those elements for which mass data was not generated.

5. Mass data will be computed for the following elements: CBAR, CBEAM, CBEND, CBUSH1D, CHEXA, CMASSi, CONMi, CONROD, CPENTA, CPLSTNi, CPLSTSi, CPYRAM, CQUAD4, CQUAD8, CQUADR, CQUADX4, CQUADX8, CROD, CSHEAR, CTETRA, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, CTRIAX6, CTUBE.

6. EIDSUM takes precedence over EID if both describers are used.

7. PIDSUM takes precedence over PID if both describers are used.

8. If the NSMCONT describer is specified in combination with the EID describer, a table is produced that identifies the contribution of each non-structural mass bulk entry to the total element non-structural mass. If the NSMCONT describer is specified in combination with the PID describer, a table is produced that identifies non-structural mass from all NSM type bulk entries for all element property types by property identification number.

Remarks related to SOLs 601 and 701:

1. NSM will be zero in the summary tables.

Enhancements for plane stress and plane strain elements

The CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 plane strain elements and the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 plane stress elements were introduced in NX Nastran 7.1 and were only applicable to SOL 601. With the release of NX Nastran 8, the capabilities of these elements have been enhanced. Now, you can:

- Use these elements in linear structural solutions (SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 187, and 200) and heat transfer solutions (SOLs 153 and 159). For these solutions, you must use PPLANE bulk entries to define the element properties.

- Specify different thicknesses at each corner node of the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 plane stress elements.

- Model nearly incompressible materials better by using a mean dilatational element formulation for the CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 elements. To do so, specify the new FOROPT field on the PPLANE bulk entry.

For more information, see the updated versions of the CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, MAT3, and PPLANE bulk entries.

CPLSTN3 Plane Strain Triangular Element Connection

Defines a plane strain triangular element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTN3	EID	PID	G1	G2	G3	THETA			

Example:

CPLSTN3	111	203	31	74	75	30.0			

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The grid points must lie in an X-Z plane of the basic coordinate system.
3. The element coordinate system is the basic coordinate system.
4. The reference coordinate system for the output of linear stress/strain is the material coordinate system.
5. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.
6. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

7. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTN4 Plane Strain Quadrilateral Element Connection

Defines a plane strain quadrilateral element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTN4	EID	PID	G1	G2	G3	G4	THETA		

Example:

CPLSTN3	111	203	31	74	75	32	15.0		

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The grid points must lie in an X-Z plane of the basic coordinate system.
3. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
4. All interior angles must be less than 180^{o}.
5. The element coordinate system is the basic coordinate system.
6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. Incompatible modes are used for this element. Incompatible modes may be turned off by specifying ICMODE=0 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTN6 Plane Strain Triangular Element Connection

Defines a plane strain triangular element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTN6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA								

Example:

CPLSTN6	302	3	31	33	71	32	51	52	

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. G1, G2, G3 are the corner grid points and G2, G4, G6 are the midside grid points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The grid points must lie in an X-Z plane of the basic coordinate system.
3. Grid points G1 through G6 must be numbered as shown in CTRIA6.
4. The element coordinate system is the basic coordinate system.
5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 6-node triangular elements may be converted to 7-node triangular elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTN8 Plane Strain Quadrilateral Element Connection

Defines a plane strain quadrilateral element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTN8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	THETA						

Example:

CPLSTN3	207	3	31	33	73	71	32	51	
	53	72	45.0						

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. G1 to G4 are the corner grid points and G5 to G8 are the midside grid points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points must lie in an X-Z plane of the basic coordinate system.

3. Grid points G1 through G8 must be numbered as shown in CQUAD8.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. The 9-node plane strain element is more effective in the analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson’s ratio close to 0.5.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTS3 Plane Stress Triangular Element Connection

Defines a plane stress triangular element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTS3	EID	PID	G1	G2	G3		THETA		
				TFLAG	T1	T2	T3		

Example:

CPLSTN3	111	203	31	74	75		30.0		
				1	1.2	1.2	1.2		

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points must lie in an X-Z plane of the basic coordinate system.

3. T1, T2, and T3 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTS4 Plane Stress Quadrilateral Element Connection

Defines a plane stress quadrilateral element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTS4	EID	PID	G1	G2	G3	G4	THETA		
				TFLAG	T1	T2	T3	T4	

Example:

CPLSTS4	111	203	31	74	75	32	30.0		
					0.8	0.8	0.8	0.8	

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points must lie in an X-Z plane of the basic coordinate system.

3. T1, T2, T3 and T4 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. The element coordinate system is the basic coordinate system.

5. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

6. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

7. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

8. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. Incompatible modes are used for this element. Incompatible modes may be turned off by specifying ICMODE=0 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTS6 Plane Stress Triangular Element Connection

Defines a plane stress triangular element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTN6	EID	PID	G1	G2	G3	G4	G5	G6	
			THETA	TFLAG	T1	T2	T3		

Example:

CPLSTN6	302	3	31	33	71	32	51	52	
			25.0	1	0.2	0.2	0.2		

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points must lie in an X-Z plane of the basic coordinate system.

3. T1, T2, and T3 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

4. Grid points G1 through G6 must be numbered as shown in CTRIA6.

5. The element coordinate system is the basic coordinate system.

6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 6-node triangular elements may be converted to 7-node triangular elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

CPLSTS8 Plane Stress Quadrilateral Element Connection

Defines a plane stress quadrilateral element for use in linear or nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CPLSTS8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	THETA	TFLAG	T1	T2	T3	T4	

Example:

CPLSTS8	207	3	31	33	73	71	32	51	
	53	72	70.0	1	0.5	0.5	0.5	0.5	

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PPLANE or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of connected points. (Unique integers > 0)
THETA	Material property orientation angle in degrees. It is the angle between x-direction of the material coordinate system and x-direction of the element coordinate system. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
TFLAG	Integer flag which specifies how Ti is used to define element thickness. (0, 1, or blank)
Ti	When Ti > 0.0 and TFLAG = 0 or blank, Ti overrides T on the PPLANE or PLPLANE if > 0.0. When Ti > 0.0 and TFLAG = 1, Ti is a multiplier of T on the PPLANE or PLPLANE. (Real > 0.0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. The grid points must lie in an X-Z plane of the basic coordinate system.

3. Grid points G1 through G8 must be numbered as shown in CQUAD8.

4. T1, T2, T3 and T4 are optional. If Ti is blank, then it will be set equal to the value of T on the PPLANE or PLPLANE entry.

5. The element coordinate system is the basic coordinate system.

6. The reference coordinate system for the output of linear stress/strain is the material coordinate system.

7. The element behaves linearly if used in a SOL 106 or SOL 129 analysis.

8. The cyclic solution types (SOLs 114,115,116,118) and the aero solution types (SOLs 144,145,146) are not supported.

9. GPSTRESS or GPSTRAIN output is not supported.

Remarks related to SOL 601:

1. The reference coordinate system for the output of linear stress/strain is the material coordinate system except when MAT1 is assigned to the element. Then the material coordinate system is ignored and the linear stress/strain is output in the basic coordinate system.

2. The reference coordinate system for the output of nonlinear stress/strain is the undeformed element coordinate system, which is the same as the basic coordinate system.

3. The reference coordinate system for the output of hyperelastic stress/strain is the basic coordinate system.

4. 8-node elements may be converted to 9-node elements (with 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry.

5. When THETA is defined, the positive element normal direction, which is defined by the G1, G2, and G3 connectivity using the right-hand-rule, must be consistent with the negative y-direction of the basic system.

MAT3 Axisymmetric Element Orthotropic Material Property Definition

Defines the material properties for linear orthotropic materials used by the CQUADX4, CQUADX8, CTRAX3, CTRAX6, and CTRIAX6 axisymmetric elements. Also defines the material properties for linear orthotropic materials used by the CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 planar membrane elements.

Format for Axisymmetric Elements:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	Eq	EZ	NUXq	NUqZ	NUZX	RHO	
			GZX	AX	Aq	AZ	TREF	GE	

Format for Planar Membrane Elements:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	EY	EZ	NUXY	NUYZ	NUZX	RHO	
			GZX	AX	AY	AZ	TREF	GE	

Example:

MAT3	23	1.0+7	1.1+7	1.2+7	.3	.25	.27	1.0-5	
			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

Fields:

Field	Contents
MID	Material identification number. (Integer > 0)
EX, Eq, EZ	Young's moduli in the X-, q-, and Z-directions, respectively, for axisymmetric elements. (Real > 0.0)
EX, EY, EZ	Young's moduli in the X-, Y-, and Z-directions, respectively, for planar membrane elements. (Real > 0.0)

Field	Contents
NUXq, NUqZ, NUZX	Poisson’s ratios for axisymmetric elements: (Real) NUXq = Poisson’s ratio for strain in the q-direction when stressed in the X-direction. NUqZ = Poisson’s ratio for strain in the Z-direction when stressed in the q-direction. NUZX = Poisson’s ratio for strain in the X-direction when stressed in the Z-direction.
NUXY, NUYZ, NUZX	Poisson’s ratios for planar membrane elements: (Real) NUXY = Poisson’s ratio for strain in the Y-direction when stressed in the X-direction. NUYZ = Poisson’s ratio for strain in the Z-direction when stressed in the Y-direction. NUZX = Poisson’s ratio for strain in the X-direction when stressed in the Z-direction.
RHO	Mass density. (Real)
GZX	Shear modulus in the XZ-plane. (Real > 0.0)
AX, Aq, AZ	Thermal expansion coefficients in the X-, q-, and Z-directions, respectively, for axisymmetric elements. (Real)
AX, AY, AZ	Thermal expansion coefficients in the X-, Y-, and Z-directions, respectively, for planar membrane elements. (Real)
TREF	Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark 9. (Real or blank)
GE	Structural element damping coefficient. See Remarks 10 and 11. (Real)

Remarks:

1. The material identification number must be unique with respect to the collection of all MAT1, MAT2, MAT3, MAT8, MAT9, and MAT11 entries.

2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE case control command.

3. Values for all seven elastic constants, EX, Eq (or EY), EZ, NUXq (or NUXY), NUqZ (or NUYZ), NUZX, and GZX, must be present.

4. A warning message will be issued if any value of NUXq (or NUXY) or NUqZ (or NUYZ) has an absolute value greater than 1.0.

5. The X-direction, the q- or Y-direction, and the Z-direction are principal material directions. Each element supporting the use of MAT3 contains a “THETA” or “TH” field to relate the principal material directions (i.e. the material coordinate system) to the basic coordinate system.

6. For axisymmetric elements CQUADX4, CQUADX8, CTRAX3, CTRAX6, and CTRIAX6, the strain-stress relationship is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \dfrac{1}{EX} & -\left(\dfrac{NU\theta X}{E\theta}\right) & -\left(\dfrac{NUZX}{EZ}\right) & 0 \\ -\left(\dfrac{NUX\theta}{EX}\right) & \dfrac{1}{E\theta} & -\left(\dfrac{NUZ\theta}{EZ}\right) & 0 \\ -\left(\dfrac{NUXZ}{EX}\right) & -\left(\dfrac{NU\theta Z}{E\theta}\right) & \dfrac{1}{EZ} & 0 \\ 0 & 0 & 0 & \dfrac{1}{GZX} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - TREF) \begin{Bmatrix} AX \\ A\theta \\ AZ \\ 0 \end{Bmatrix}$$

where

$$\frac{NUX\theta}{EX} = \frac{NU\theta X}{E\theta}$$
$$\frac{NU\theta Z}{E\theta} = \frac{NUZ\theta}{EZ}$$
$$\frac{NUZX}{EZ} = \frac{NUXZ}{EX}$$

7. For the plane stress planar membrane elements CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 with the Y-direction as the transverse direction, the strain-stress relationship is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{\mathrm{EX}} & -\left(\frac{\mathrm{NUZX}}{\mathrm{EZ}}\right) & 0 \\ -\left(\frac{\mathrm{NUXZ}}{\mathrm{EX}}\right) & \frac{1}{\mathrm{EZ}} & 0 \\ 0 & 0 & \frac{1}{\mathrm{GZX}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - \mathrm{TREF}) \begin{Bmatrix} \mathrm{AX} \\ \mathrm{AZ} \\ 0 \end{Bmatrix}$$

where

$$\frac{\mathrm{NUXY}}{\mathrm{EX}} = \frac{\mathrm{NUYX}}{\mathrm{EY}}$$
$$\frac{\mathrm{NUYZ}}{\mathrm{EY}} = \frac{\mathrm{NUZY}}{\mathrm{EZ}}$$
$$\frac{\mathrm{NUZX}}{\mathrm{EZ}} = \frac{\mathrm{NUXZ}}{\mathrm{EX}}$$

8. For the plane strain planar membrane elements CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 with the Y-direction as the transverse direction, the strain-stress relationship is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1-(\mathrm{NUYX})(\mathrm{NUXY})}{\mathrm{EX}} & -\left(\frac{\mathrm{NUZX}+(\mathrm{NUYX})(\mathrm{NUZY})}{\mathrm{EZ}}\right) & 0 \\ -\left(\frac{\mathrm{NUXZ}+(\mathrm{NUYX})(\mathrm{NUXY})}{\mathrm{EX}}\right) & \frac{1-(\mathrm{NUYZ})(\mathrm{NUZY})}{\mathrm{EZ}} & 0 \\ 0 & 0 & \frac{1}{\mathrm{GZX}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_z \\ \tau_{zx} \end{Bmatrix}$$

$$+ (T - \mathrm{TREF}) \begin{Bmatrix} \mathrm{AX} + (\mathrm{NUYX})(\mathrm{AY}) \\ \mathrm{AZ} + (\mathrm{NUYZ})(\mathrm{AY}) \\ 0 \end{Bmatrix}$$

where

$$\frac{\mathrm{NUXY}}{\mathrm{EX}} = \frac{\mathrm{NUYX}}{\mathrm{EY}}$$
$$\frac{\mathrm{NUYZ}}{\mathrm{EY}} = \frac{\mathrm{NUZY}}{\mathrm{EZ}}$$
$$\frac{\mathrm{NUZX}}{\mathrm{EZ}} = \frac{\mathrm{NUXZ}}{\mathrm{EX}}$$

9. TREF is used for two different purposes:

   - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 11 in the MAT1 description.

   - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.

10. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

11. If PARAM,W4 is not specified, GE is ignored in a transient analysis.

Remarks related to SOL 601:

1. GE is ignored.

2. TREF is used only when MAT3 is made temperature dependent by use of the MATT3 entry.

PPLANE Properties of Plane Stress or Plane Strain Elements

Defines the properties of plane stress elements or plane strain elements.

Format:

1	2	3	4	5	6	7	8	9	10
PPLANE	PID	MID	T	NSM	FOROPT				

Examples:

PPLANE	203	204	0.11						

PPLANE	1	3	0.5	0.0	1				

Fields:

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Material identification number. (Integer >0)
T	Thickness for plane stress elements. If T is blank or zero then the thickness must be specified for Ti on the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 entries. T is ignored for plane strain elements. (Real ≥ 0.0 or blank)
NSM	Nonstructural mass per unit area. (Real)
FOROPT	Formulation option number. See Remark 6. (Integer; Default = 0)

Remarks:

1. All PPLANE property entries should have unique identification numbers with respect to all other property entries.

2. The entry is referenced by the CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, CPLSTN3, CPLSTN4, CPLSTN6, and CPLSTN8 entries via PID.

3. For structural problems, MID must reference a MAT1 or MAT3 material property entry.

4. For heat transfer problems, MID must reference a MAT4 or MAT5 material property entry.

5. In a model where there are many plane stress elements with different thicknesses, it is recommended that the thickness values be specified in the element entries (Ti value in CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8) instead of assigning one PPLANE entry to one thickness value and resulting in many PPLANE entries.

6. The default value of 0 uses the standard formulation. A value of 1 specifies the Mean Dilatational Formulation option for nearly incompressible materials.

7. The fields on the PPLANE cannot be associated to design variables in SOL 200.

Remarks related to SOL 601:

1. NSM and FOROPT are ignored.

2. For structural problems, MID must reference a MAT1, MAT8 or MATSMA material property entry.

Edge loads for CPLSTN and CPLSTS elements

Beginning with NX Nastran 8, a new bulk entry, PLOADE1, is introduced. You can use this bulk entry to define surface tractions along the edge of CPLSTN3, CPLSTN4, CPLSTN6, or CPLSTN8 plane strain elements, or along the edge of CPLSTS3, CPLSTS4, CPLSTS6, or CPLSTS8 plane stress elements. All the solution sequences that support the listed elements also support the use of the PLOADE1 bulk entry.

When using SOL 601, you can define the surface tractions such that they:

- Maintain their orientation relative to the element geometry or maintain their original orientation in a geometric nonlinear analysis.

- Are time-dependent or time-independent in a transient analysis.

For more information, see the new PLOADE1 bulk entry.

PLOADE1 Edge Load on Plane Strain and Plane Stress Elements

Defines a surface traction acting on an edge of a CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, or CPLSTS8 element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADE1	SID	EID	PA	PB	GA	GB	THETA		

Example:

PLOADE1	200	35	3.5	10.5	10	30	20.		

Fields:

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA. (Real)
PB	Surface traction at grid point GB. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0)

Remarks:

1. In static solution sequences, the load set ID (SID) is selected by the LOAD case control command. In dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the LOADSET case control command.

2. The surface traction varies linearly along the element edge between GA and GB.

3. The surface traction is input as force per unit length.

4. THETA is measured counter-clockwise from the inward normal of the straight line defined by grid points GA and GB, to the direction that the surface traction acts, as shown in Figures 4-2 and 4-3.

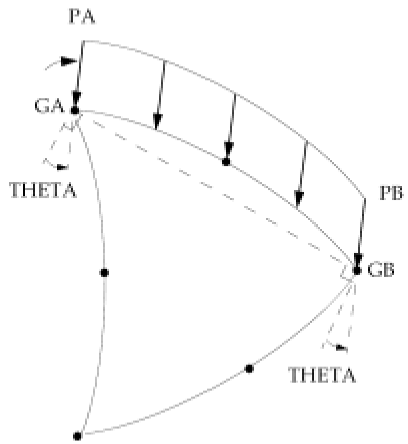


Figure 4-2. Pressure Load on CPLSTN3, CPLSTN6, CPLSTS3, and CPLSTS6 Elements

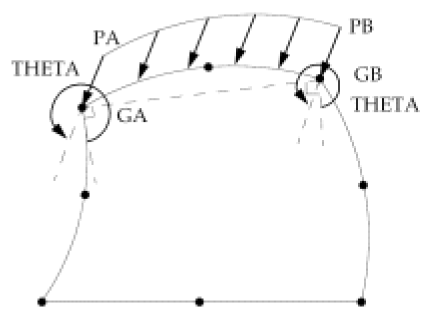


Figure 4-3. Pressure Load on CPLSTN4, CPLSTN8, CPLSTS4, and CPLSTS8 Elements

5. If THETA = 0.0 and the edge of the element is a straight line, the surface traction is simply an applied pressure loading.

Remarks related to SOL 601:

1. To define a surface traction as time-independent, use LOAD = SID in the case control.

2. To define a surface traction as time-dependent, reference the SID in the EXCITEID field of a TLOAD1 entry and include a DLOAD case control command that references the TLOAD1 entry.

3. In large deformation analysis, the direction of the surface traction follows the deformation of the element by default. The use of LOADOPT = 0 in an

NXSTRAT entry causes the surface tractions to act in their original direction throughout the analysis.

Composite laminate solid elements

Beginning with NX Nastran 8, you can assign composite laminate properties to CHEXA and CPENTA solid elements. To support this capability, two new bulk entries have been created.

- Use the new PCOMPS bulk entry to:

  Define the orientation, stacking, and thickness of each ply.

  Assign MAT1, MAT9, or MAT11 material property entries to each ply.

  Define reference temperature and damping coefficient for the laminate.

  Specify ply and inter-laminar failure theories and define allowable inter-laminar shear and normal stresses.

- Use the new MATFT bulk entry to define material constants for use in the Hill, Hoffman, Tsai-Wu, Maximum Strain, Maximum Stress, and Maximum Transverse Shear Stress ply failure theories.

The STRESS and STRAIN case control commands have been enhanced to allow you to specify whether the stress and strain output for each ply is at the bottom, middle, or top of the ply, or at a combination of these locations.

For additional information, see the new PCOMPS and MATFT bulk entries. Also see the updated CHEXA and CPENTA bulk entries, and the updated STRAIN and STRESS case control commands.

CHEXA Six-Sided Solid Element Connection

Defines the connections of the six-sided solid element with eight to twenty grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	0	0	30	31	53	54	
	55	56	57	58	59	60			

Fields:

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID, PLSOLID, or PCOMPS entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

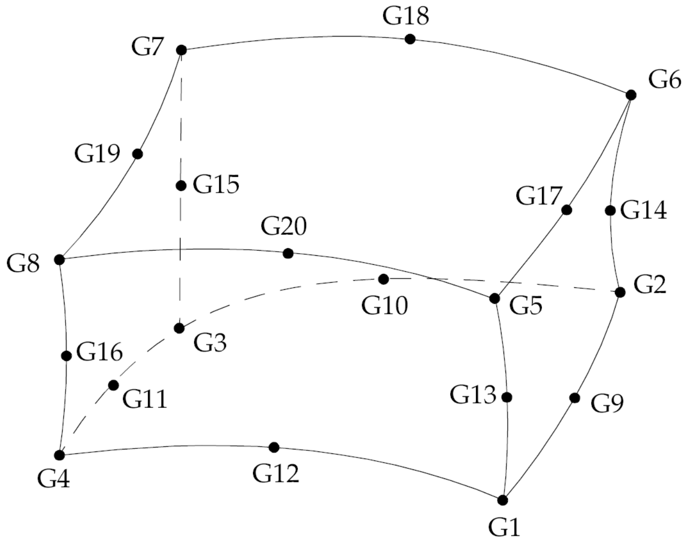


Figure 4-4. CHEXA Element Connection

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.

3. The edge points G9 to G20 are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.

4. Components of stress are output in the material coordinate system. See Remark 8 on the PSOLID bulk entry for hyperelastic and nonlinear exceptions. See Remark 6 on the PCOMPS bulk entry for composite laminate exception.

5. The second continuation is optional.

6. Except when used as a hyperelastic element or as a composite laminate solid element, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.

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

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

T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system, a 3 x 3 positive-definite symmetric matrix would be produced.)

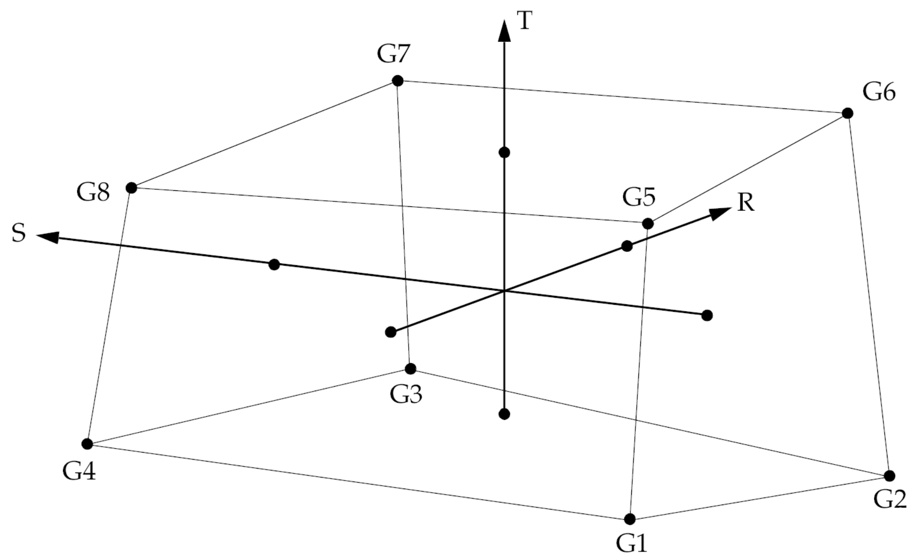


Figure 4-5. CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.

8. For hyperelastic elements, the plot codes are specified under the CHEXAFD element name in “Item Codes”.

9. If a CHEXA element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.

   - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.

- Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.

10. By default, all twelve edges of the element are considered straight unless:

    - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.

    - For h-elements, any of G9 through G20 are specified.

Remarks related to SOLs 601 and 701:

1. For SOL 601, only elements with 8 or 20 grid points are allowed, i.e., either all edge points G9 to G20 are specified or no edge points are specified. For SOL 701, only elements with 8 grid points are allowed.

2. For SOL 601, 20-node CHEXA elements may be converted to 27-node CHEXA elements (6 additional nodes on the centroid of the six faces and 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. 27-node CHEXA elements are especially effective in the analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson’s ratio close to 0.5.

CPENTA Five-Sided Solid Element Connection

Defines the connections of a five-sided solid element with six to fifteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CPENTA	112	2	3	15	14	4	103	115	
	5	16	8				120	125	
	130								

Fields:

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID, PLSOLID, or PCOMPS entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

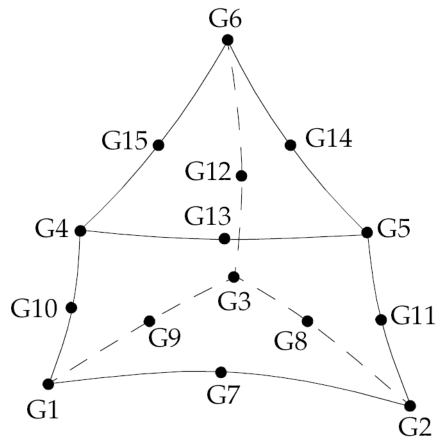


Figure 4-6. CPENTA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.

2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.

3. The edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.

4. Components of stress are output in the material coordinate system. See Remark 8 on the PSOLID bulk entry for hyperelastic and nonlinear exceptions. See Remark 6 on the PCOMPS bulk entry for composite laminate exception.

5. Except when used as a hyperelastic element or as a composite laminate solid element, the element coordinate system for the CPENTA element is defined as follows: The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

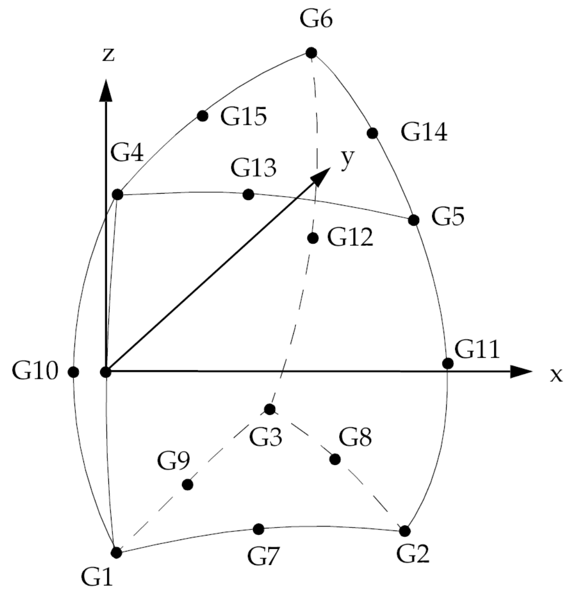


Figure 4-7. CPENTA Element Coordinate System

6. It is recommended that the edge grid points be located within the middle third of the edge.

7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in “Item Codes”.

8. If a CPENTA element is referenced on a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.

   - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.

   - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.

9. By default, all of the nine edges of the element are considered straight unless:

   - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.

   - For h-elements any of G7 through G15 are specified.

Remarks related to SOLs 601 and 701:

1. For SOL 601, only elements with 6 or 15 grid points are allowed, i.e., either all edge points G7 to G15 are specified or no edge points are specified. For SOL 701, only elements with 6 grid points are allowed.

2. For SOL 601, 15-node CPENTA elements may be converted to 21-node CPENTA elements (5 additional nodes on the centroid of the five faces and 1 additional node at the centroid of the element) by specifying ELCV=1 in the NXSTRAT entry. 21-node CPENTA elements are more effective than 15-node CPENTA for analysis of incompressible media and inelastic materials, e.g., rubber-like materials, elasto-plastic materials, and materials with Poisson's ratio close to 0.5.

PCOMPS Layered Composite Element Property for Solid Elements

Defines the properties of an n-ply composite material laminate for CHEXA and CPENTA solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPS	PID	CORDM	PSDIR	SB	NB	TREF	GE		
	GPLYIDi	MIDi	TRi	THETAi	FTi	ILFTi	SOUTi		

Example:

PCOMPS	20	2	13	10000.					
	2	1	0.02	90.	TSAI	NB	YES		
	3	2	0.03	45.	HILL	SB	YES		

Fields:

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
CORDM	Identification number of the material coordinate system. Enter “0” or leave blank to use the basic coordinate system. (Integer; Default = 0)
PSDIR	Ply and stack directions in the material coordinate system. Enter the X-, Y-, and Z-directions of the material coordinate system as 1, 2, and 3, respectively. (Integer; 12,13,21,23,31,32; Default = 13)
SB	Allowable inter-laminar shear stress of the bonding material. See Remark 1. (Real > 0.0 or blank)
NB	Allowable inter-laminar normal stress of the bonding material. See Remark 1. (Real > 0.0 or blank)
TREF	Reference temperature. (Real; Default = 0.0)
GE	Damping coefficient. (Real; Default = 0.0)
GPLYIDi	Global ply IDs. (Integer > 0)
MIDi	Material ID of the various plies. The MIDs must refer to MAT1, MAT9, or MAT11 bulk entries. (Integer > 0 or blank)

Field	Contents
TRi	Ply thickness. See Remark 2. (Real > 0.0)
THETAi	Ply orientation angle. See Remark 3. (Real; Default = 0.0)
FTi	Ply failure theory. Allowable entries are: Blank for no failure theory. "HILL" for the Hill failure theory. "HOFF" for the Hoffman failure theory. "TSAI" for the Tsai-Wu failure theory. "STRN" for the Maximum Strain failure theory. "STRS" for the Maximum Stress failure theory. "TS" for the Maximum Transverse Shear Stress failure theory. See Remark 4. For a detailed explanation of each failure theory, see "Laminates" in the *NX Nastran User's Guide*. (Character or blank)
ILFTi	Inter-laminar failure theory. Allowable entries are: Blank for no failure index. "SB" for transverse shear stress failure index. "NB" for normal stress failure index. (Character or blank)
SOUTi	Controls individual ply stress and strain output. See Remark 7. Allowable entries are: "NO" for do not compute. (Default) "YES" for compute. (Character or blank)

Remarks:

1. If SB and NB are not specified, then inter-laminar failure indices and/or strength ratios will not be computed.

2. The laminate thickness is adjusted at the corners to coincide with the distance between grid points. The thickness of each ply in the laminate is adjusted proportionally.

3. Ply orientation and stack direction will be determined from the material coordinate system (CORDM). The ply orientation direction will be at an angle relative to the local X-direction of the ply. The local X-direction is the projection of the n-direction of the CORDM onto the ply, where n is the first number in the PSDIR field. The stack direction corresponds to the m-direction of the CORDM, where m is the second number in the PSDIR field.

4. FTi is failure theory for i-th ply. The material properties used in the failure theories are specified by a MATFT bulk entry.

5. To compute a ply and/or bonding failure index, the STRESS case control command must be present, SOUTi on the PCOMPS bulk entry must be set to "YES", and the following must be defined.

   For a stress or strain ply failure index:

   a. FTi on the PCOMPS bulk entry.

   b. The stress or strain allowables on the referenced MATFT bulk entry.

   For a stress bonding failure index:

   a. ILFTi on the PCOMPS bulk entry.

   b. The stress allowables SB or NB on the PCOMPS bulk entry.

   By default, failure index output prints in the f06 file even when using the PLOT or PUNCH describers on the STRESS and STRAIN case control commands. The parameter entry PARAM,NOFISR,1 can be used to turn off the printing of the failure index output. See the parameter NOFISR.

6. Ply stress and strain results are always computed in the ply coordinate system.

7. To request that ply stress and/or strain be computed, the STRESS and/or STRAIN case control command must be defined with the appropriate PRINT, PUNCH, or PLOT output option, and the SOUTi field must equal "YES". The STRESS and STRAIN commands also include the CPLYMID, CPLYBT, and CPLYBMT describers to specify stress or strain recovery at the bottom, middle, or top of the plies. See the remarks on the STRESS and STRAIN case control commands.

8. GPSTRESS or GPSTRAIN output is not supported.

9. Glue or contact definitions defined on composite solid faces which are perpendicular to the stack direction (edge faces) may produce poor stress continuity. If the glue/contact definition is between edge faces belonging to different PCOMPS definitions, and if the number of plies on each PCOMPS definition is small and the same, and the ply thicknesses are similar, the stress continuity should be fairly smooth. This also applies to the results requested with the BCRESULTS and BGRESULTS case control commands.

MATFT Material Properties for Composite Ply Failure Theories

Defines material properties for use with composite ply failure theories.

Format 1: Hill Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"HILL"	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

Format 2: Hoffman Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"HOFF"	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

Format 3: Tsai-Wu Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"TSAI"	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23	F12	F13	F23			

Format 4: Maximum Strain Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"STRN"	Xet	Xec	Yet	Yec	Zet	Zec	Se12	
		Se13	Se23						

Format 5: Maximum Stress Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"STRS"	Xt	Xc	Yt	Yc	Zt	Zc	S12	
		S13	S23						

Format 6: Maximum Transverse Shear Stress Failure Theory

1	2	3	4	5	6	7	8	9	10
MATFT	MID								
	"TS"	S13	S23						

Example:

1	2	3	4	5	6	7	8	9	10
MATFT	20								
	HILL	1.0E5	5.4E6	5.0E4	2.0E4	2.0E4	2.0E4		
	HOFF	7.2E6	5.6E6	5.2E6	4.4E6	4.0E5	3.5E5	2.1E6	
		2.1E6	2.1E6						

Fields:

Field	Contents
MID	Material identification number. See Remark 1. (Integer > 0)
FTi	Ply failure theory. Allowable entries are: "HILL" for the Hill failure theory. "HOFF" for the Hoffman failure theory. "TSAI" for the Tsai-Wu failure theory. "STRN" for the Maximum Strain failure theory. "STRS" for the Maximum Stress failure theory. "TS" for the Maximum Transverse Shear Stress failure theory. For a detailed explanation of each failure theory, see "Laminates" in the *NX Nastran User's Guide*. (Character or blank)
Xc, Yc, Zc	Allowable compressive stress in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xc for Yc and Zc)
Xt, Yt, Zt	Allowable tensile stress in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xt for Yt and Zt)
S12, S13, S23	Allowable in-plane shear stress (S12) and transverse shear stresses (S13 and S23). (Real > 0.0)
F12, F13, F23	Interaction terms in the tensor polynomial theory of Tsai-Wu. Required if the failure index is desired and either F12, F13, or F23 are non-zero. See Remark 5. (Real; Default = 0.0)

Field	Contents
Xec, Yec, Zec	Allowable compressive strain in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xec for Yec and Zec)
Xet, Yet, Zet	Allowable tensile strain in the longitudinal, lateral, and transverse principal material directions, respectively. (Real > 0.0; Default = Xet for Yet and Zet)
Se12, Se13, Se23	Allowable in-plane shear strain (Se12) and transverse shear strains (Se13 and Se23). (Real > 0.0)

Remarks:

1. The MID on a MATFT bulk entry must be the same as the MID on a MAT1, MAT9, or MAT11 bulk entry. Only one MATFT bulk entry is allowed per MID.

2. When a MATFT bulk entry is used in combination with a MAT1 bulk entry, any stress limits defined on the MAT1 bulk entry are ignored and the stress limits defined on the MATFT bulk entry are used.

3. Multiple failure theories can be supported on a single MATFT bulk entry, but only one set of failure moduli is allowed for each failure theory, and each failure theory can only be specified once.

4. If any failure modulus related to the transverse direction is not specified, failure indices and strength ratios are only calculated for the two-dimensional plane of the ply. If any failure modulus related to the longitudinal or lateral direction is not specified, failure indices and strength ratios are not calculated.

5. If you enter a nonzero value for F_{12} and the first stability criterion below is not satisfied, the software uses F_{12} = 0.0. This behavior is the same for F_{23} and F_{13} using the second and third stability criterion below, respectively. Each is evaluated independently.

$$\left(\frac{1}{Xt\ Xc}\right)*\left(\frac{1}{Yt\ Yc}\right)-F_{12}^2>0$$

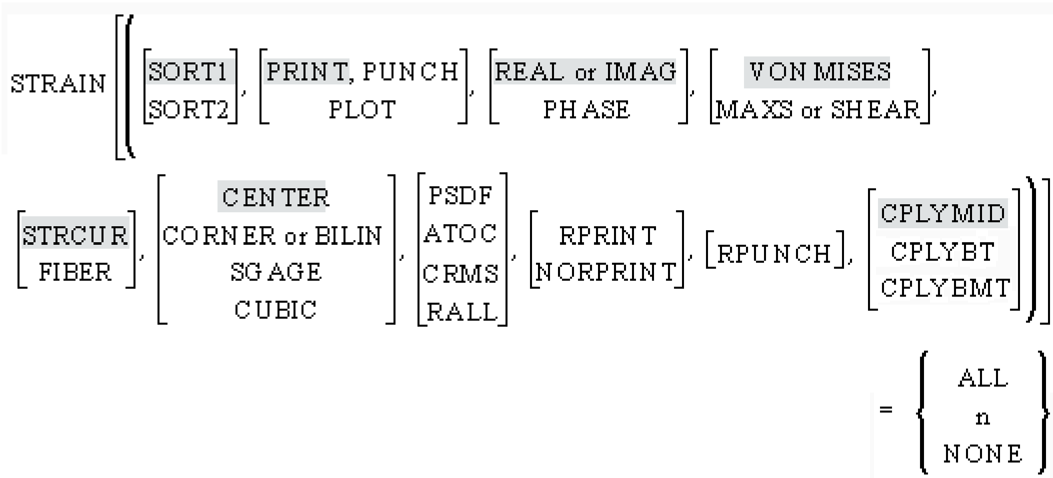
$$\left(\frac{1}{Zt\ Zc}\right)*\left(\frac{1}{Yt\ Yc}\right)-F_{23}^2>0$$

$$\left(\frac{1}{Xt\ Xc}\right)*\left(\frac{1}{Zt\ Zc}\right)-F_{13}^2>0$$

STRAIN Element Strain Output Request

Requests the form and type of strain output. Note: ELSTRAIN is an equivalent command.

Format:



Examples:

```
STRAIN=5
STRAIN(CORNER)=ALL
STRAIN(PRINT,PHASE)=15
STRAIN(PLOT)=ALL
```

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates strain for the requested set but no printer output.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be made in the case control. See Remark 10.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 10.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 10.
RPUNCH	Writes random analysis results to the punch file. See Remark 10.
VONMISES	von Mises strain is output.
MAXS or SHEAR	Maximum shear strains are output.
STRCUR	Strain at the reference plane and curvatures is output for plate elements.

Describer	Meaning
FIBER	Strain at locations Z1, Z2 is computed for plate elements.
CENTER	Outputs strains at the center of the element only. See Remark 15.
CORNER or BILIN	Outputs strains at the center and corner grid points using extrapolation. See Remark 16.
SGAGE	Outputs strains at the center and corner grid points using strain gage approach. See Remark 17.
CUBIC	Outputs strains at the center and corner grid points using cubic bending correction. See Remark 18.
CPLYMID	Requests element strains at the middle of each ply for elements referencing PCOMPS property entries. See Remarks 11, 12, 13, and 14.
CPLYBT	Requests element strains at the bottom and top of the ply for elements referencing PCOMPS property entries. See Remarks 11, 12, 13, and 14.
CPLYBMT	Requests element strains at the bottom, middle, and top of each ply for elements referencing PCOMPS property entries. See Remarks 11, 12, 13, and 14
ALL	Output strain for all elements.
n	Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	No element strain will be output.

Remarks:

1. In SOLs 106 and 129, nonlinear strains for nonlinear elements are requested by the STRESS/NLSTRESS commands and appear in the nonlinear stress output. The STRAIN command will generate additional output for total strain except for hyperelastic elements. The additional STRAIN output request will also be ignored for nonlinear material elements when the parameter LGDISP is -1, which is the default (strains will appear in the nonlinear stress output).

2. Both PRINT and PUNCH may be requested.

3. STRAIN=NONE overrides an overall output request.

4. The PLOT option is used when strains are requested for postprocessing but no printer output is desired.

5. Definitions of stress, strain, curvature, and output locations are given in the *NX Nastran Element Library Reference Manual*.

6. If the STRCUR option is selected, the values of Z1 will be set to 0.0 and Z2 will be set to -1.0 on the output.

7. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.

8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also the FORCE command for further discussion. These options are discussed in “Understanding Plate and Shell Element Output” in the *NX Nastran Element Library Reference*.

9. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

   - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

   - SORT2 is not supported for ply strains on the PCOMP, PCOMG, and PCOMPS bulk entries.

10. The following applies to SOL 111 PSD solutions:

    - Frequency response output occurs in addition to any random output. The PRINT,PUNCH,PLOT describers control the frequency response output. The RPRINT,NORPRINT,RPUNCH describers control the random output.

    - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the parameter RPOSTS.

    - Any combination of the PSDF, ATOC, and CRMS describers can be selected. The RALL describer selects all three.

    - When requesting PSDF, CRMS, ATOC, or RALL, both the overall RMS and the Number of Zero Crossing tables are always calculated.

11. If some combination of the CPLYMID, CPLYBT, or CPLYBMT describers are specified, the describer producing the most output data is used.

12. If different CPLYMID, CPLYBT, or CPLYBMT describers are specified on STRESS and STRAIN case control commands in the same input file, the describer specified on the STRESS case control command takes precedence.

13. Failure indices and strength ratios for in-plane ply failure are output for the locations corresponding to the CPLYMID/CPLYBT/CPLYBMT specification and the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification. Failure indices and strength ratios for inter-laminar failure are always output at the top and bottom of each ply at the locations corresponding to the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification.

14. For elements referencing PCOMPS property entries, the REAL, IMAG, PHASE, VONMISES, MAXS, SHEAR, STRCUR, FIBER, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, NOPRINT, and RPUNCH describers are not supported.

15. For the CENTER option, strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center of the element for each ply.

16. For the CORNER (or BILIN) option, strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center and corner grid locations for each ply.

17. For the SGAGE option, strains are output at the center and grid points for CQUAD4 elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

18. For the CUBIC option, strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

STRESS Element Stress Output Request

Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.

Format:

$$\text{STRESS}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{VONMISES}\\ \text{MAXS or SHEAR}\end{bmatrix},\right.\right.$$

$$\left.\left.\begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, \left[\text{RPUNCH}\right], \begin{bmatrix}\text{CPLYMID}\\ \text{CPLYBT}\\ \text{CPLYBMT}\end{bmatrix}\right)\right]$$

$$= \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Examples:

```
STRESS=5
STRESS(CORNER)=ALL
STRESS (SORT1,PRINT,PUNCH,PHASE)=15
STRESS(PLOT)=ALL
```

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates stresses for requested set but no printer output.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 13.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 13.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be made in the case control. See Remark 13.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 13.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 13.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 13.
RPUNCH	Writes random analysis results to the punch file. See Remark 13.
VONMISES	von Mises stress is output.
MAXS or SHEAR	Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.
CENTER	Outputs stresses at the center of the element only. See Remark 18.

Describer	Meaning
CORNER or BILIN	Outputs stresses at the center and corner grid points using extrapolation. See Remark 19.
SGAGE	Outputs stresses at center and corner grid points using strain gage approach. See Remark 20.
CUBIC	Outputs stresses at the center and corner grid points using cubic bending correction. See Remark 21.
CPLYMID	Requests element stresses at the middle of each ply for elements referencing PCOMPS property entries. See Remarks 14, 15, 16, and 17.
CPLYBT	Requests element stresses at the bottom and top of the ply for elements referencing PCOMPS property entries. See Remarks 14, 15, 16, and 17.
CPLYBMT	Requests element stresses at the bottom, middle, and top of each ply for elements referencing PCOMPS property entries. See Remarks 14, 15, 16, and 17.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output. (Integer >0)
NONE	No element stress will be output.

Remarks:

1. Both PRINT and PUNCH may be requested.

2. ALL should not be used in a transient problem due to excessive output.

3. The defaults for SORT1 and SORT2 depend on the type of analysis:

   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

- SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

- XY plot requests will force SORT2 format thus overriding SORT1 format requests.

- SORT2 is not supported for ply stresses on the PCOMP, PCOMG, and PCOMPS bulk entries.

4. ELSTRESS is an alternate form and is entirely equivalent to STRESS.

5. STRESS=NONE overrides an overall output request.

6. The PLOT option is used when contour plots of stresses are requested but no printer output of stresses is desired. However in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.

7. The VONMISES option is ignored for ply stresses.

8. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.

9. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also the FORCE command for further discussion. These options are discussed in the section “Understanding Plate and Shell Element Output” in Chapter 4 of the *NX Nastran Element Library Reference*.

10. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).

11. For nonlinear analysis, the grid point option for CQUAD4 elements will be reset to the default (CENTER) option for nonlinear elements.

12. MAXS for shell elements is not an equivalent stress.

13. The following applies to SOL 111 PSD solutions:

- Frequency response output occurs in addition to any random output. The PRINT,PUNCH,PLOT describers control the frequency response output. The RPRINT,NORPRINT,RPUNCH describers control the random output.

- The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the parameter RPOSTS.

- Any combination of the PSDF, ATOC, and CRMS describers can be selected. The RALL describer selects all three.

- When requesting PSDF, CRMS, ATOC, or RALL, both the overall RMS and the Number of Zero Crossing tables are always calculated.

14. If some combination of the CPLYMID, CPLYBT, or CPLYBMT describers are specified, the describer producing the most output data is used.

15. If different CPLYMID, CPLYBT, or CPLYBMT describers are specified on STRESS and STRAIN case control commands in the same input file, the describer specified on the STRESS case control command takes precedence.

16. Failure indices and strength ratios for in-plane ply failure are output for the locations corresponding to the CPLYMID/CPLYBT/CPLYBMT specification and the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification. Failure indices and strength ratios for inter-laminar failure are always output at the top and bottom of each ply at the locations corresponding to the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification.

17. For elements referencing PCOMPS property entries, the REAL, IMAG, PHASE, VONMISES, MAXS, SHEAR, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, NOPRINT, and RPUNCH describers are not supported.

18. For the CENTER option, stresses are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that are referenced by a PSHELL entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply stresses are always reported at the center of the element. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center of the element. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply stresses are output at the center of the element for each ply.

19. For the CORNER (or BILIN) option, stresses are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a

PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply stresses are output at the center and corner grid locations for each ply. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points.

20. For the SGAGE option, stresses are output at the center and grid points for CQUAD4 elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points of the element.

21. For the CUBIC option, stresses are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN. For CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry and PARAM,NOCOMPS is greater than or equal to 0 (see PARAM,NOCOMPS), the homogeneous stresses (based on the smeared representation of the laminate properties) will be output at the center and grid points of the element.

Remarks related to SOLs 601 and 701:

1. Output is restricted to REAL format and CENTER or CORNER for CQUAD4 element stresses. IMAG, PHASE, CUBIC, SGAGE, PSDF, ATOC and RALL are ignored.

2. The same request is used for element strain output in both linear and nonlinear analysis, i.e., the STRAIN command is ignored for linear analysis.

3. For both linear and nonlinear analysis, stress and strain results at grid points may be requested for single-ply elements by specifying the CORNER option.

4. If there is any nonlinearity in the model, i.e., large displacements, contact, or nonlinear material models, elements results are output in the op2 file using nonlinear element stress/strain data block, including elements with linear material models. Exceptions are the CQUAD8 and CTRIA6 elements, where both linear and nonlinear element stress/strain data blocks are output. However, in a future release, the output of linear stress/strain data blocks for CQUAD8 and CTRIA6 elements may be discontinued for a nonlinear analysis.

Alternate material coordinate system specification method for solid elements

Beginning with NX Nastran 8, a new bulk entry, MATCID, is introduced. The MATCID bulk entry assigns the coordinate system that you enter in the CORDM field as the material coordinate system for elements you list in the EIDi fields. Use this new bulk entry to override the material coordinate systems for the following:

- The CHEXA, CPENTA, CTETRA, and CPYRAM solid elements that are specified on PSOLID property entries referenced by the solid elements.

- The CHEXA and CPENTA solid elements that are specified on PCOMPS property entries referenced by the solid elements.

Consider using MATCID bulk entries when your model includes groups of solid elements that have many different material coordinate systems, but otherwise have identical element properties. For such a case, you can map a group of elements to a single PSOLID or PCOMPS property entry and use MATCID bulk entries to assign each material coordinate system to the appropriate solid elements. Because the number of PSOLID and PCOMPS property entries is minimized, the size of the input file is reduced and the model data is handled more efficiently.

For additional information, see the new MATCID bulk entry.

MATCID Material Coordinate System for Solid Elements

Overrides the material coordinate system for CHEXA, CPENTA, CTETRA, and CPYRAM solid elements when the elements reference a PSOLID property. Also overrides the material coordinate system for CHEXA and CPENTA solid elements when the elements reference a PCOMPS property.

Format:

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	
	EID8	EID9	EID10	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
MATCID	20	101	102	103	104	105			

Alternate Formats and Examples:

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	"THRU"	EID2					
MATCID	20	101	THRU	105					

1	2	3	4	5	6	7	8	9	10
MATCID	CID	EID1	"THRU"	EID2	"BY"	N			
MATCID	20	101	THRU	105	BY	2			

1	2	3	4	5	6	7	8	9	10
MATCID	CID	"ALL"							
MATCID	20	ALL							

Fields:

Field	Contents
CID	Material coordinate system identification number. (Integer > -2) See Remarks 1 and 2.
EIDi	Element identification number. (Integer > 0 or "ALL" or "THRU" or "THRU" with "BY"; For "THRU" options, EID1 < EID2)
N	Element selection increment for use with "THRU" with "BY" option. (Integer ≥ 1)

Remarks:

1. For CHEXA, CPENTA, CTETRA, and CPYRAM solid elements that reference a PSOLID property, CID = -1 refers to the element coordinate system and CID = 0 refers to the basic coordinate system.

2. For CHEXA and CPENTA solid elements that reference a PCOMPS property, CID = -1 and CID = 0 are treated the same with both referring to the basic coordinate system. Ply stresses and strains are output in the ply coordinate system rather than the CID coordinate system.

Rigid element processing

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. With the release of NX Nastran 8, two new parameters, LMSTAT and LMDYN, are available to control artificial stiffness when using 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:

$$\begin{bmatrix} M_{GG} & 0 \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \ddot{x}_G \\ \ddot{\lambda}_M \end{Bmatrix} + \begin{bmatrix} K_{GG} & RG^T \\ RG & 0 \end{bmatrix} \begin{Bmatrix} x_G \\ \lambda_M \end{Bmatrix} = \begin{Bmatrix} p_G \\ 0 \end{Bmatrix}$$

Because the resulting augmented stiffness matrix is often singular, NX Nastran optionally adds artificial stiffness terms that connect the DOF in each constraint. 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_l 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_l 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_l 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_l 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.

For more information on rigid element processing, see “Rigid element processing options” in the *NX Nastran Element Library Reference*.

Element, property, and material selection tables

Beginning with NX Nastran 8, the *NX Nastran User’s Guide* contains:

- An element applicability table to assist you in selecting which of the most commonly used elements are suited for a particular type of analysis.

- Cross-reference tables to assist you in selecting property and material bulk entries that are compatible with a given element type.

The applicability table and cross-reference tables are valid for SOLs 101–200.

Element applicability table

Table 4-2 is the *Element Applicability Table for Solutions 101–200*. It provides you with a concise reference for determining which of the most commonly used elements have formulations suited to a particular analysis type. After each element listing there is a footnote. The footnote describes how the element will behave in analyses that the table indicates are unsuitable for the element type.

For example, Table 4-2 indicates that the CBAR element is not formulated for nonlinear structural analysis. However, the footnote for the CBAR element indicates that you can use CBAR elements in a nonlinear structural analysis, but when you do so:

- The CBAR elements will behave linear elastically in a material nonlinear analysis.
- The stiffness of the CBAR elements will not reformulate in a geometric nonlinear analysis.

Table 4-2. Element Applicability for Solutions 101–200

Type	Element	Linear Structural SOLs (1)	Nonlinear Structural, SOLs 106 and 129					Heat Transfer SOLs 153 and 159
			Material Nonlinear (2)				Geometric Nonlinear (3)	
			Plasticity	Nonlinear Elastic	Hyper-elasticity	Creep		
1D	CBAR[a]	X						X
	CBEAM[b]	X	X(4)				X	X
	CBEND[a]	X						X
	CONROD[b]	X	X(5)	X(5)			X	X
	CROD[b]	X	X(5)	X(5)		X(5)	X	X
	CTUBE[b]	X	X(5)	X(5)			X	X

Table 4-2. Element Applicability for Solutions 101–200

Type	Element	Linear Structural SOLs [1]	Nonlinear Structural, SOLs 106 and 129					Heat Transfer SOLs 153 and 159
			Material Nonlinear [2]				Geometric Nonlinear [3]	
			Plasticity	Nonlinear Elastic	Hyper-elasticity	Creep		
2D	CPLSTN3[a]	X						X
	CPLSTN4[a]	X						X
	CPLSTN6[a]	X						X
	CPLSTN8[a]	X						X
	CPLSTS3[a]	X						X
	CPLSTS4[a]	X						X
	CPLSTS6[a]	X						X
	CPLSTS8[a]	X						X
	CQUAD[c]				X			
	CQUAD4[d]	X	X	X	X	X	X	X
	CQUAD8[e]	X			X			X
	CQUADR[a]	X						X
	CRAC2D[a]	X						
	CSHEAR[a]	X						
	CTRIA3[d]	X	X	X	X	X	X	X
	CTRIA6[e]	X			X			X
	CTRIAR[a]	X						X
Axi-sym	CCONEAX[a]	X						
	CQUADX[c]				X			
	CQUADX4[e]	X			X			X
	CQUADX8[e]	X			X			X
	CTRAX3[e]	X			X			X
	CTRAX6[e]	X			X			X
	CTRIAX[c]				X			
	CTRIAX6[a]	X						X
3D	CHEXA[d]	X	X	X	X	X	X	X
	CPENTA[d]	X	X	X	X	X	X	X
	CPYRAM[b]	X	X	X		X	X	X
	CRAC3D[a]	X						
	CTETRA[d]	X	X	X	X	X	X	X

Table 4-2. Element Applicability for Solutions 101–200

Type	Element	Linear Structural SOLs (1)	Nonlinear Structural, SOLs 106 and 129					Heat Transfer SOLs 153 and 159
			Material Nonlinear (2)				Geometric Nonlinear (3)	
			Plasticity	Nonlinear Elastic	Hyper-elasticity	Creep		
Element Footnotes:								
(a) Behaves linear elastically if used in a material nonlinear analysis. Stiffness does not reformulate if used in a geometric nonlinear analysis.								
(b) Limited material nonlinear capability that is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness reformulates if used in a geometric nonlinear analysis.								
(c) Only supported for hyperelastic analysis.								
(d) Comprehensive material nonlinear capability that is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness reformulates if used in a geometric nonlinear analysis.								
(e) Hyperelastic capability is accessed through certain material bulk entries. Otherwise, behaves linear elastically if used in a material nonlinear analysis. Stiffness does not reformulate if used in a geometric nonlinear analysis.								
Other footnotes:								
(1) Includes SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 187, and 200.								
(2) Hyperelasticity is the only nonlinear capability that allows axisymmetric analysis.								
(3) Small strain only. For large strain, refer to the hyperelasticity column.								
(4) Elastic-perfectly plastic material behavior used to model the ends of beams as plastic hinges.								
(5) Nonlinear capability for axial deflections only.								

Element, property, material cross-reference tables

Tables 4-4 through 4-8 list the properties and materials you can use with a given element for Solutions 101–200. To assist you in navigating to the correct table, Table 4-3 is provided.

Table 4-3. Summary of Material, Property, and Element Cross-Reference Tables

Element Type	Elements	Table
1D	CBAR, CBEAM, CBEND, CONROD, CROD, CTUBE	4-4
2D	CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8, CQUAD, CQUAD4, CQUAD8, CQUADR, CRAC2D, CSHEAR, CTRIA3, CTRIA6, CTRIAR	4-5
Axisymmetric	CCONEAX, CQUADX, CQUADX4, CQUADX8, CTRAX3, CTRAX6, CTRIAX, CTRIAX6	4-6
3D	CHEXA, CPENTA, CPYRAM, CRAC3D, CTETRA	4-7
Special Purpose	CAABSF, CAERO1, CAERO2, CAERO3, CAERO4, CAERO5, CAXIFi, CBUSH, CBUSH1D, CDAMP1, CDAMP2, CDAMP3, CDAMP4, CDAMP5, CDUMi, CELAS1, CELAS2, CELAS3, CELAS4, CFAST, CFLUIDi, CGAP, CHACAB, CHACBR, CHBDYE, CHBDYG, CHBDYP, CMASS1, CMASS2, CMASS3, CMASS4, CONM1, CONM2, CONV, CONVM, CSLOT3, CSLOT4, CVISC, CWELD	4-8

Table 4-4. 1D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CBAR	PBAR	MAT1			X								
		MAT4						X					
		MAT5							X				
	PBARL	MAT1			X								
		MAT4						X					
		MAT5							X				

Table 4-4. 1D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CBEAM	PBEAM	MAT1		X[1]	X								
		MAT4						X					
		MAT5							X				
	PBEAML	MAT1			X								
		MAT4						X					
		MAT5							X				
	PBCOMP	MAT1		X[1]	X								
		MAT4						X					
		MAT5							X				
CBEND	PBEND	MAT1			X								
		MAT4						X					
		MAT5							X				
CONROD	N/A	MAT1		X[2]	X								
		MAT4						X					
		MAT5							X				
CROD	PROD	MAT1		X[2]	X								X[2]
		MAT4						X					
		MAT5							X				
CTUBE	PTUBE	MAT1		X[2]	X								
		MAT4						X					
		MAT5							X				

[1] For material nonlinear analysis, elastic-perfectly plastic material behavior only. Strain hardening is not supported. The ends of the element are modeled as plastic hinges.

[2] Nonlinear capability for axial deflections only.

Table 4-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CPLSTN3	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTN4	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTN6	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTN8	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTS3	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTS4	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CPLSTS6	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				

Table 4-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CPLSTS8	PPLANE	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
CQUAD	PLPLANE	MATHP											
CQUAD4	PSHELL	MAT1		X	X								X
		MAT2		X		X							X
		MAT4						X					
		MAT5							X				
		MAT8		X						X			X
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			
	PLPLANE	MATHP											
	N/A	MFLUID											
CQUAD8	PSHELL	MAT1			X								
		MAT2				X							
		MAT4						X					
		MAT5							X				
		MAT8								X			
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			
	PLPLANE	MATHP											

Table 4-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CQUADR	PSHELL	MAT1			X								
		MAT2				X							
		MAT4						X					
		MAT5							X				
		MAT8								X			
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			
CRAC2D	PRAC2D	MAT1			X								
		MAT2				X							
		MAT8								X			
CSHEAR	PSHEAR	MAT1											
CTRIA3	PSHELL	MAT1		X	X								X
		MAT2		X		X							X
		MAT4						X					
		MAT5							X				
		MAT8		X						X			X
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			
	PLPLANE	MATHP											
	N/A	MFLUID											

Table 4-5. 2D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CTRIA6	PSHELL	MAT1			X								
		MAT2				X							
		MAT4						X					
		MAT5							X				
		MAT8								X			
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			
	PLPLANE	MATHP											
CTRIAR	PSHELL	MAT1			X								
		MAT2				X							
		MAT4						X					
		MAT5							X				
		MAT8								X			
	PCOMP	MAT1			X								
		MAT2				X							
		MAT8								X			
	PCOMPG	MAT1			X								
		MAT2				X							
		MAT8								X			

Table 4-6. Axisymmetric Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CCONEAX	PCONEAX	MAT1			X								
		MAT2				X							
CQUADX	PLPLANE	MATHP											
CQUADX4	PSOLID	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
		MAT9									X		
	PLSOLID	MATHP											
CQUADX8	PSOLID	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
		MAT9									X		
	PLSOLID	MATHP											
CTRAX3	PSOLID	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
		MAT9									X		
	PLSOLID	MATHP											
CTRAX6	PSOLID	MAT1			X								
		MAT3					X						
		MAT4						X					
		MAT5							X				
		MAT9									X		
	PLSOLID	MATHP											
CTRIAX	PLPLANE	MATHP											
CTRIAX6	N/A	MAT1			X								
		MAT3					X						
		MAT4						X					

Table 4-7. 3D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CHEXA	PSOLID	MAT1		X	X								X
		MAT4						X					
		MAT5							X				
		MAT9		X							X		X
		MAT10											
		MAT11		X								X	X
	PLSOLID	MATHP											
	PCOMPS	MAT1	X		X								
		MAT9	X								X		
		MAT11	X									X	
CPENTA	PSOLID	MAT1		X	X								X
		MAT4						X					
		MAT5							X				
		MAT9		X							X		X
		MAT10											
		MAT11		X								X	X
	PLSOLID	MATHP											
	PCOMPS	MAT1	X		X								
		MAT9	X								X		
		MAT11	X									X	
CPYRAM	PSOLID	MAT1		X	X								X
		MAT4						X					
		MAT5							X				
		MAT9		X							X		X
		MAT10											
		MAT11		X								X	X
CRAC3D	PRAC3D	MAT1			X								
		MAT9									X		

Table 4-7. 3D Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CTETRA	PSOLID	MAT1		X	X								X
		MAT4						X					
		MAT5							X				
		MAT9		X							X		X
		MAT10											
		MAT11		X								X	X
	PLSOLID	MATHP											

Table 4-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CAABSF	PAABSF	N/A											
CAERO1	PAERO1	N/A											
CAERO2	PAERO2	N/A											
CAERO3	PAERO3	N/A											
CAERO4	PAERO4	N/A											
CAERO5	PAERO5	N/A											
CAXIFi (1)	N/A	N/A											
CBUSH	PBUSH (2)	N/A											
CBUSH1D	PBUSH1D	N/A											
CDAMP1	PDAMP	N/A											
CDAMP2	N/A	N/A											
CDAMP3	PDAMP	N/A											
CDAMP4	N/A	N/A											
CDAMP5	PDAMP5	MAT4						X					
		MAT5							X				
CDUMi (3)	PDUMi (3)	(4)											
CELAS1	PELAS	N/A											

Table 4-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200

Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
CELAS2	N/A	N/A											
CELAS3	PELAS	N/A											
CELAS4	N/A	N/A											
CFAST	PFAST	N/A											
CFLUIDi (1)	N/A	N/A											
CGAP	PGAP	N/A											
CHACAB	PACABS	N/A											
CHACBR	PACBAR	N/A											
CHBDYE	N/A	N/A											
CHBDYG	N/A	N/A											
CHBDYP	PHBDY	N/A											
CMASS1	PMASS	N/A											
CMASS2	N/A	N/A											
CMASS3	PMASS	N/A											
CMASS4	N/A	N/A											
CONM1	N/A	N/A											
CONM2	N/A	N/A											
CONV	PCONV	MAT4						X					
CONVM	PCONVM	MAT4						X					
CSLOT3	N/A	N/A											
CSLOT4	N/A	N/A											
CVISC	PVISC	N/A											
CWELD	PWELD	MAT1			X								
RBAR	N/A	N/A											
RBEi (5)	N/A	N/A											

(1) i = 2, 3, or 4

(2) Use a PBUSHT entry with the same PID as a PBUSH entry to define frequency dependent or stress dependent properties.

(3) i = 1 through 9

(4) Per user definition.

Table 4-8. Special Purpose Elements: Supplemental Materials by Primary Material, Property, and Element for Solutions 101–200													
Element	Property	Primary Material	Supplemental Material										
			MATi										CREEP/ MATTC
			FT	S1	T1	T2	T3	T4	T5	T8	T9	T11	
(5) i = 1, 2, or 3													

Chapter 5 Contact for linear solutions

Contact conditions with inertia relief

Inertia relief is an NX Nastran option that allows you to simulate unconstrained structures in a static analysis. For example, an aircraft in flight is unconstrained, yet it can be analyzed in a static analysis. With inertia relief, the mass of the structure resists the applied loadings such that the structure is in a state of static equilibrium even though it is unconstrained in any or all degrees of freedom.

Contact conditions and the automatic inertia relief option can now be used together in a SOL 101 static solution.

The automatic inertia relief option is requested by including PARAM,INREL,-2 in the bulk data section.

Since INREL=-2 requires that six rigid body modes exist (a free-free structure), active elements in the contact source and target regions need to be in close proximity to each other. If the initial distance between these active elements exceeds that allowed by small displacement theory, the applied contact stiffness will produce a grounding effect, and the 6 rigid body modes will no longer exist. This usually results in all contact elements becoming inactive or in a solution with excessive penetration (the contact condition is not enforced).

See the “Inertia Relief in Linear Static Analysis” chapter in the *NX Nastran User’s Guide* for more information.

Contact with Static Condensation

Static condensation is an optional NX Nastran method of partitioning and reformulating the stiffness matrix with the goal of reducing the solution time. This method requires the model be partitioned into the analysis set (A-set) and omitted set (O-set). In a static analysis, the results using static condensation are numerically exact. The partitioned solution merely changes the order of the operations of the unpartitioned solution.

An automatic static condensation option is now available to use with models which include contact conditions. To select this option, set the new parameter CNTASET to “YES”. For example,

PARAM,CNTASET,YES

If you select this option, you should not manually select degrees of freedom with the ASET bulk entry. When PARAM,CNTASET,YES is defined, NX Nastran automatically places the degrees of freedom which are part of the contact portion of the solution into the A-set. The remaining degrees of freedom are then automatic placed into the O-set. The result is that a static condensation is performed to reduce the full K_{gg} matrix to the K_{aa} matrix which contains only the contact degrees of freedom. The contact iterations are then performed using the resulting K_{aa} matrix.

This new solution option shows the best improvement when the number of contact degrees of freedom is small compared to the overall number of degrees of freedom in the model. As the reduced A-set stiffness matrix becomes larger and more dense, the performance benefit decreases.

This option is only supported in a linear statics solution (SOL 101). The reduced mass at the contact degrees of freedom may not be an accurate representation of the original mass distribution. As a result, the software will issue a WARNING and continue without the A-set reduction if you attempt to use the PARAM,CNTASET,YES option in the dynamic solutions 103, 111, or 112.

Because PARAM,CNTASET,NO is the default option, automatic condensation does not occur by default.

Restrictions when PARAM,CNTASET,YES is defined

- If multiple subcases exist, the same contact set and the same constraint set must be used by all subcases. This can be achieved by including the BCSET and SPC case control commands in the global subcase. If the subcases use different contact and constraint sets, the software will continue without the A-set reduction, although singularities are likely.

- Bolt preload conditions are not supported.

- The iterative solver is not supported.

- You cannot select additional degrees of freedom with the ASET bulk entry to include in the K_{aa} matrix. The software determines all A-set and O-set degrees of freedom.

- The dynamic solutions 103, 111, or 112 are not supported.

See the “Understanding Sets and Matrix Operations” chapter in the *NX Nastran User’s Guide* for more information on static condensation.

Chapter

6 Glue enhancements

Glue surface tractions

You can now request glue surface tractions for solid and shell elements using the new case control command, BGRESULTS. The glue traction request is supported in a linear static solution (SOL 101), in a normal modes solution (SOL 103) but only for a static preload subcase if present, and in a linear buckling solution (SOL 105). The glue results from SOL 105 are a result of the applied static loads which are not necessarily the loads at which buckling occurs.

The new glue tractions, which are similar to the existing contact results, are calculated and stored at the grids which are on the glue surfaces. The normal component of the tractions is a scalar, while the in-plane (tangential) tractions are output in the basic coordinate system. For an edge-to-surface glue pair, only forces and not tractions are recovered.

See the new BGRESULTS case control command.

BGRESULTS Glue Result Output Request (SOLs 101, 103, 105)

Format:

$$\text{BGRESULTS}\left[\left(\text{TRACTION}, \text{FORCE}, \begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix}, \begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Examples:

```
BGRESULTS=ALL
BGRESULTS(FORCE,PLOT)=ALL
BGRESULTS(TRACTION,FORCE,PLOT)=ALL
```

Describers:

Describer	Meaning
TRACTION	Glue normal traction (scalar) and in-plane glue tractions (vector in basic coordinate system) are output for each glue grid point.
FORCE	Glue force vector is output for each glue grid point.
PRINT	The printer will be the output medium.
PLOT	Computes and puts glue results in OP2 file only.
PUNCH	The punch file is the output media.
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue or time, depending of the solution sequence.
SORT2	Output will be presented as a tabular listing of load,frequency or time for each grid point.
ALL	Glue results at all contact grid points will be output.
NONE	Glue results will not be output.
n	Set identification of a previously appearing SET command. Only glue grid points with identification numbers that appear on this SET command will be output. (Integer>0)

REMARKS:

1. The glue traction request is supported in a linear static solution (SOL 101), in a normal modes solution (SOL 103) but only for a static preload subcase if present, and in a linear buckling solution (SOL 105). The glue results from SOL 105 are a result of the applied static loads which are not necessarily the loads at which buckling occurs.

REMARKS RELATED TO SOLS 601 AND 701:

1. SOLs 601 and 701 do not support BGRESULTS requests.

Chapter 7 Advanced nonlinear

Advanced nonlinear enhancements

A summary of the advanced nonlinear enhancements is included here. See the *NX Nastran Advanced Nonlinear Theory and Modeling Guide* and the *NX Nastran Quick Reference Guide* for details on these enhancements.

Viscoelastic material models

New viscoelastic material models are available for SOL 601.

- Use the new MATHEV bulk entry to specify a viscoelastic effect (Holzapfel model) for a hyperelastic material.

- Use the new MATVE and TABVE bulk entries to specify a viscoelastic effect for a non-hyperelastic material.

Mullins effect with hyperelastic material

SOL 601 now supports the Mullins effect (Ogden-Roxburgh model) for a hyperelastic material. Use the new MATHEM bulk entry to specify this capability.

Creep enhancement

In releases prior to NX Nastran 8, creep and elasto-plastic material models were supported, but they could not be combined in the same analysis. Now they can both be included within the same SOL 601 analysis.

CBUSH element support in SOL 601 and SOL 701

The generalized spring and damper element defined with the CBUSH bulk entry is now supported in both SOL 601 and SOL 701. Both “K” and “B” on the PBUSH bulk entry are supported, and “KN” on the PBUSHT bulk entry is supported. For geometric nonlinear analysis, the stiffness and damping direction remains constant in the undeformed configuration throughout the analysis.

Edge load support for plane stress and plane strain elements

Use the new PLOADE1 bulk entry to define an in-plane distributed load along the edge of a plane stress or a plane strain element in SOL 601. Enter the distributed load in terms of force per unit length on the PLOADE1 entry.

MAT11 and MATCID support in SOL 601 and SOL 701

The MAT11 bulk entry, which is used to define orthotropic materials for solid elements, is now supported in both SOL 601 and SOL 701. In addition, you can use the new MATCID bulk entry in SOL 601 to overwrite the MID defined on the PSOLID bulk entry. The MATCID entry allows you to avoid a large number of PSOLID entries due to different material orientations.

DMIG support in SOL 601

SOL 601 now supports matrices defined with the DMIG (Direct Matrix Input at Grids) bulk entry. Use the K2GG, M2GG, and B2GG case control commands to select DMIG entries defining the stiffness, mass, and viscous damping matrices, respectively. You can use the CK2, CM2, and CB2 parameters to scale these matrices.

DMIG matrices can optionally be created in an initial, statics reduction solution (SOL 101) which includes the EXTSEOUT case control command along with the ASET bulk entry. See the *NX Nastran Superelement User's Guide* for details on external superelement creation.

Stress output at Gauss points for solid and axisymmetric elements

In releases prior to NX Nastran 8, nonlinear stress and strain for solid and axisymmetric elements were only calculated and stored at the grids during a SOL 601 or SOL 701 analysis. Now the option to request the nonlinear stress and strain at the Gauss points is available. To output nonlinear stress and strain at the Gauss points, select the new GAUSS option in the STRESS field of the PSOLID bulk entry.

Variable thickness support for plane stress elements

In NX Nastran 7.1, the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 plane stress elements were introduced for use in SOL 601. These elements were restricted to constant thickness. This restriction has now been removed. You can now define the thickness at each corner of these elements by entering the thicknesses in the Ti fields of the CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 bulk entries.

Miscellaneous changes for SOL 601

- The MAXDISP parameter, which is defined on the NXSTRAT bulk entry, specifies a limit for the maximum incremental displacement that is allowed for any grid in any equilibrium iteration. Now, when contact is defined and MAXDISP=0.0 (default) in a static run, and when no nonlinear conditions exist, a maximum incremental displacement of 1% of model length is used.

- In NX Nastran 7.1, stiffness matrix stabilization was automatically used when mesh glue conditions were defined. It has been found that this could lead to incorrect results in a linear analysis, or poor convergence in a nonlinear analysis. Hence, in this release, stiffness matrix stabilization is not automatically used for mesh gluing.

Chapter

8 Optimization

Designable frequency dependent properties

Previous versions of NX Nastran did not allow design variables to be associated with frequency dependent properties. Using the DVPREL1 and DVPREL2 bulk entries, you can now make frequency dependent properties designable by associating design variables or design variable combinations with them.

The PBUSHT, PDAMPT, and PELAST bulk entries define frequency dependent properties. When you include these bulk entries in the input data and request a frequency response analysis, the nominal values of properties defined by the PBUSH, PDAMP, and PELAS entries are replaced by those pointed at by the PBUSHT, PDAMPT, PELAST entries, as applicable.

PBUSHT, PDAMPT, PELAST entries specify property values by referencing TABLEDi bulk entries, which provide values as functions of frequencies. The analysis frequencies are matched by appropriate interpolation or extrapolation over the table values.

The main difference between the frequency dependent properties and their frequency independent counterparts is the frequency at which each of the former are defined. To make the frequency dependent properties designable, you can add a frequency to either or both of the DVPREL1 and DVPREL2 bulk entries. The added frequency indicates that the property being designed is defined at that particular frequency. Thus, in general:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	C0	**FREQ**	
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

or

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	EQID	**FREQ**	
	“DESVAR”	DVID1	DVID2	DVID3	-etc.-				
	“DTABLE”	LABL1	LABL2	LABL3	-etc.-				

where FREQ is the frequency at which you define the designed property referenced by DVPREL1 or DVPREL2. If the FREQ field is left blank, it defaults to “all frequencies (including 0.0)” as before. Thus the relevant DVPRELi should only be for the frequency independent property, as in PBUSH, PDAMP, or PELAS, or any other property bulk entry. However, currently this cannot be done for frequency response analysis if there already exists a directly corresponding frequency dependent property, for example TKID when the DVPRELi is for Ki of same PID.

The frequencies specified with the DVPREL1 or DVPREL2 bulk entries do not have to match the table frequencies. The property values computed by the substitution of the design variable initial values into the DVPREL1 or DVPREL2 relations also do not have to match the TABLEDi obtained values. Instead, the TABLEDi are re-generated, based on the DVPREL1 or DVPREL2 defined values. Thus, when you design frequency dependent properties, it is essential to have as many DVPREL1 and DVPREL2 entries as necessary to re-create the relevant TABLEDi values with a sufficient number of data points.

With NX Nastran 8, only TABLED1 are available for use with designed frequency dependent properties, and the use of TABLED1 for this case is limited to the LINEAR, LINEAR default options for XAXIS and YAXIS. For frequency dependent properties not associated with design variables, other options and other TABLEDi can be used.

Also, currently, a frequency specified with a DVPREL1 or DVPREL2 bulk entry has to match an analysis frequency, unless no frequency response analysis is requested.

Example 1

Consider a scalar spring connection (EID = 222) that has frequency dependent stiffness and is used in design optimization involving frequency response analysis (ANALYSIS = MFREQ).

```
CELAS1, 222, 333, 101, 1, 102, 1
PELAS, 333, 1.21
PELAST, 333, 444
TABLED1, 444
       , 3.0, 0.7, 5.0, 0.95, 7.0, 1.32, 9.0, 1.46
       , ENDT
FREQ, 777, 4.5, 6.5, 8.5
```

Therefore, in the MFREQ analysis, at a given frequency, the stiffness value determined by PELAST and TABLED1 supersedes the nominal value given by the PELAS bulk entry.

If you need to design the frequency dependent stiffnesses, then design variables can be associated with the PELAST bulk entry in the following manner. The assumption in the example is that you will design the PELAST based stiffness values at the three analysis frequencies.

```
DVPREL1, 567, PELAST, 333, TKID,  ,  ,  , 4.5
       , 865, 0.75, 866, 0.34, 867, 1.31
DVPREL1, 568, PELAST, 333, TKID,  ,  ,  , 6.5
```

```
       , 865, 0.87, 866, 0.74
DVPREL2, 569, PELAST, 333, TKID,  ,  , 888, 8.5
       , DESVAR, 866, 867,
       , DTABLE, ABC1, ABC4, ABC6
```

From the above example, note that the only change to the previous use of the DVPREL1 and DVPREL2 bulk entries has been to allow you to associate a frequency with each instance of such data when designed frequency dependent properties exist.

Also in the above example, the updated TABLED1 bulk entry will now contain only three points. These points represent the values of the TKID property of the PELAST bulk entry at the three analysis frequencies of the FREQ bulk entry, and the three property values will be designable. That is, their values will in general change from one design cycle to the other.

At the end of a SOL 200 run that has designable frequency dependent properties, the updated TABLED1 bulk entry may be output into a punch file in bulk data format for the best or last design cycle, as you specified in the ECHO case control command. The TABLED1 entry is output into the punch file as updated analysis model data.

Example 2

Updated analysis and design model data for a job as it appears in the punch file – including the designed PELAST TKID entry as given by the updated TABLED1 entry – is as follows:

```
$   UPDATED DESIGN MODEL DATA ENTRIES
$
DESVAR *             865DESVAR1          8.00000012E-01  8.00000012E-01+D    1V
*D   1V   1.00999999E+00  1.00000000E+00
DESVAR *             866DESVAR2          1.00999999E+00  1.00999999E+00+D    2V
*D   2V   1.40999997E+00  1.00000000E+00
DESVAR *             867DESVAR3          6.00000024E-01  6.00000024E-01+D    3V
*D   3V   1.40999997E+00  1.00000000E+00
$
$   UPDATED ANALYSIS MODEL DATA ENTRIES

PELAST*              333             444               0               0
TABLED1*             444          LINEAR          LINEAR
*
*           .45000000E+01   .17294000E+00   .65000000E+01   .14434000E+01
*           .85000000E+01   .76909089E+00   ENDT
```

DVPREL1 Design Variable to Property Relation

Defines the relation between an analysis model property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	C0	FREQ	
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

Examples:

DVPREL1	12	PBAR	612	6	0.2	3.0			
	4	0.25	20	20.0	5	0.3			

DVPREL1	73	PELAST	319	TKID			0.725	3.5	
	18	0.375	28	1.52					

Fields:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a property entry, such as “PBAR”, “PBEAM”, etc. (Character)
PID	Property entry identification number. (Integer>0)
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location, then the default value for PMIN is -1.0 + 35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.0E-20)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer>0)

Field	Contents
COEFi	Coefficient of linear relation. (Real)
FREQ	Frequency at which a frequency dependent property given by PNAME/FID is defined. See Remark 8. (Real)

Remarks:

1. The relationship between the analysis model property and design variables is given by:

$$P_i = C0 + \sum_i \text{COEFi} \cdot \text{DVIDi}$$

2. The continuation entry is required.

3. PTYPE = “PBEND” is not supported. PTYPE= “PBEAML” is not supported with the FID option.

4. FID may be either a positive or a negative number. However, for PTYPE=“PBEAM” or “PBUSH”, FID must be negative. If FID>0, it identifies the field position on a property entry. If FID<0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either FID = +4 or FID = -3 can be used. In general, use of PNAME is recommended.

5. Designing PBEAML or PBEAM requires specification of both property name and station. Table 8-1 shows several examples.

Table 8-1.

PTYPE	Property Name	END A	END B	i-th Station
PBEAML	DMI1	DIM1 or DIM1(A)	DIM1(B)	DIM1(i)
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, using an END B property name on a DVPREL1 entry when the referenced PBEAM is a constant section is not allowed.

6. The PWELD and PFAST property types are not supported.

7. The PPLANE property type is not supported.

8. FREQ (Field 9) is applicable only for TYPE=PBUSHT/PDAMPT/PELAST type properties. A FREQ value must currently match an analysis frequency, though there is no need to match a TABLED1 entry. Properties should be designed at a minimum of two frequencies for a TABLED1 referenced by a designed frequency dependent property PNAME/FID, because these tables are currently re-generated, not added to.

DVPREL2 Design Variable to Property Relation

Defines the relation between an analysis model property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	EQID	FREQ	
	“DESVAR”	DVID1	DVID2	DVID3	-etc.-				
	“DTABLE”	LABL1	LABL2	LABL3	-etc.-				

Examples:

DVPREL2	13	PBAR	712	5	0.2	4			
	DESVAR	4	11	13	5				
	DTABLE	PI	YM						

DVPREL2	17	PBUSHT	122	TBID2	0.01	0.03	200	6.0	
	DESVAR	111	112	113	114				
	DTABLE	XYZ11	XYZ13	XYZ15					

Fields:

Field	Contents
ID	Unique identification number. (Integer>0)
TYPE	Name of a property entry, such as PBAR, PBEAM, etc. (Character)
PID	Property entry identification number. (Integer>0)
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.E-20)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E20)

Field	Contents
EQID	DEQATN entry identification number. (Integer>0)
"DESVAR"	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer>0)
"DTABLE"	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character. See "Bulk Data Syntax Rules".)
FREQ	Frequency at which a frequency dependent property given by PNAME/FID is defined. See Remark 9. (Real)

Remarks:

1. The variables identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where N = m+n) are assigned in the order DVID1, DVID2, ..., DVIDn, LABL1, LABL2, ..., LABLm.

2. If both "DESVAR" and "DTABLE" are specified in field 2, "DESVAR" must appear first.

3. FID may be either a positive or a negative number. However, for PTYPE="PBEAM" or "PBUSH", FID must be negative. If FID>0, it identifies the field position on a property entry. If FID<0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended.

4. PTYPE = "PBEND" is not supported.

5. Designing PBEAM requires specification of both property name and station. Table 8-2 shows one example.

Table 8-2.

PTYPE	Property Name	END A	END B	i-th Station
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, using an END B property name on a DVPREL2 entry when the referenced PBEAM is a constant section is not allowed.

6. PWELD and PFAST property types are not supported.

7. PBARL and PBEAML property types are not supported.

8. The PPLANE property type is not supported.

9. FREQ (Field 9) is applicable only for TYPE=PBUSHT/PDAMPT/PELAST type properties. A FREQ value must currently match an analysis frequency, though there is no need to match a TABLED1 entry. Properties should be designed at a minimum of two frequencies for a TABLED1 referenced by a designed frequency dependent property PNAME/FID, because these tables are currently re-generated, not added to.

Undocumented legacy inputs

Design sensitivity is supported with SOL 200 using the DSAPRT case control command. As a result, the following legacy inputs for design sensitivity analysis with SOLs 101, 103, and 105 are undocumented.

- Case control commands: SENSITY, SET2
- Parameters: EIGD, NORM
- Bulk entries: DSCONS, DVAR, DVSET

Because design sensitivity analysis occurs automatically during design optimization, it is not necessary to use the DSAPRT command unless specific sensitivity information is requested as part of the output, or only design sensitivity analysis is requested.

Design cycle data block

SOL 200 now writes the new DESCYC data block to the .op2 file. This data block defines which design cycle the results data blocks that follow it, up to the next DESCYC data block, belong to.

The data block format has the header at record 0. The first word in record 1 contains the design cycle number based on the summary table at the end of the .f06 file. The second word in record 1:

- Contains a “D” if the results that follow are for a discretized design cycle.

- Is blank if the results that follow are for a continuous design cycle.

A discretized design cycle has discrete design variables. For example, if the possible discrete values of a design variable are 0.2, 0.4, and 0.6, that design variable may not take on the value of 0.635 in a discretized design cycle. A continuous design variable, in contrast, can take on any value within its lower and upper limits. See also DISBEG and DISCOD under the DOPTPRM bulk entry, as well as the DDVAL bulk entry.

The same information is repeated in the trailer. Thus, the second word in the trailer (the first word being a 1) is the design cycle number, and the third word is a 0 or a 1 depending on whether the results that follow are for a continuous design cycle or a discretized design cycle, respectively.

How often DESCYC gets written is based on the PARAM, NASPRT parameter, similar to what is done for the results data blocks.

The following list demonstrates how DESCYC is written to the .op2 file (unit 12). The list is condensed .f06 output. You will see similar lines throughout your .f06 output, but not together as shown below.

```
…
…
     DATA BLOCK DIT      WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DYNAMICS WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK EQEXIN   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Cycle 0: "INITIAL")
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Continuous Cycle 1)
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Continuous Cycle 3)
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Continuous Cycle 5)
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Discretized Cycle 5D)
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DESCYC   WRITTEN ON FORTRAN UNIT 12, TRL = (Final, Discretized Cycle 6D)
     DATA BLOCK BOUGV1   WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK OES1     WRITTEN ON FORTRAN UNIT 12, TRL =
     DATA BLOCK DBCOPT   WRITTEN ON FORTRAN UNIT 12, TRL =
                                 END-OF-DATA SIMULATION ON FORTRAN UNIT 12
```

The following example shows an ASCII conversion of the DESCYC data block for a discretized cycle (5D in this case).

```
0RECORD NO.    0  HEADER
      NAME        NAME
  1)         DESC         YC5
                              END OF       2 WORD RECORD.
```

```
0RECORD NO.    1
     DESCYCL      CONDIS      CONDIS
  1)       5        D
                              END OF       3 WORD RECORD.
0END OF FILE
0TRAILER WORD1=  1 WORD2=  5 WORD3=  1 WORD4 =  0 WORD5=  0 WORD6=  0
```

Improved FREQ3, FREQ4, FREQ5 handling

You can now use different FREQ3, FREQ4, and FREQ5 bulk entries in SOL 200 across modal frequency response subcases (ANALYSIS = MFREQ) with different boundary conditions and/or different frequency sets.

Additionally, in releases prior to NX Nastran 8, a failure could occur when an ANALYSIS = MFREQ type subcase included the FREQ3, FREQ4, and FREQ5 bulk entries, together with all of the following conditions:

- Additional subcase types existed, for example ANALYSIS = STATICS or MTRAN.
- Boundary conditions changed across the subcases.
- The sequence of the subcases were in a certain order.

As a result, you now have more flexibility and robustness in the use of the FREQ3, FREQ4, and FREQ5 bulk entries.

Improvements in PBARL/PBEAML handling

Sensitivity analysis for responses in models with designed PBARL or PBEAML cross-sectional dimensions has been modified to significantly improve both computational speed and efficiency of disk use.

Chapter

9 RDMODES enhancements

RDMODES enhancements

The multi-level recursive domain Lanczos method (RDMODES) was originally delivered in NX Nastran 6.1 as an improvement over the automatic component modal synthesis method (ACMS) for very large models. Prior to NX Nastran 8, RDMODES was enabled for normal modes analysis (SOL 103) and modal frequency response analysis (SOL111). Beginning with NX Nastran 8:

- RDMODES is supported for modal transient response analysis (SOL 112) and modal analysis with optimization (SOL 200).
- RDMODES is supported for restarts when using SOLs 103, 111, or 112 in either serial or distributed memory parallel (DMP) runs.
- The logic for RDMODES is improved to minimize I/O usage. This improvement is similar to the partitioning performance improvements delivered in NX Nastran 7.1.
- The shared memory parallel (SMP) implementation of vendor supplied math kernels for x86-64 Linux machines is improved.

No additional inputs are required to utilize these enhancements. However, the following limitations apply:

- The software selects the sparse eigenvector recovery method option automatically based on the output requests that you define. However, the sparse eigenvector recovery method option is never used with SOL 200.
- Enforced motion and mode acceleration are not supported with the sparse eigenvector recovery method option in SOL 111 and SOL 112. If enforced motion or mode acceleration are needed, you should disable the sparse eigenvector recovery method option by specifying PARAM,RDSPARSE,NO in the bulk section of the input file.
- For restarts to work correctly, the same value for the Nastran keyword ‘nrec’ should be used in both the cold and restart runs.

- RDMODES does not support the constraint mode formulation of enforced motion. If enforced motion is requested with RDMODES, the absolute displacement formulation (system(422)=1) is automatically selected by the software.

RDMODES inputs

Multi-level RDMODES can run in either serial, SMP, or DMP configurations. A DMP with SMP hybrid configuration is also possible with the proper hardware. You activate RDMODES by entering the Nastran keyword ‘nrec’ on the command line. To specify the desired parallel functionality, you can also enter the Nastran keywords ‘dmp’ or ‘smp’. Sample command line entries include:

Serial: NASTRAN nrec = *m*

SMP: NASTRAN nrec = *m* smp = *p*

DMP: NASTRAN nrec = *m* dmp = *p*

In the above entries, *m* is the number of external partitions and *p* is the number of processors. There is no default value for the number of external partitions. Depending on the model size, the following values are suggested.

Number of grids	Suggested number of external partitions
1 ~ 5,000	4
5,000 ~ 40,000	8
15,000 ~ 80,000	16
30,000 ~ 150,000	32
60,000 ~ 300,000	64
120,000 ~ 600,000	128
250,000 ~ 1,200,000	256
> 1,200,000	512

Note that this table does not suggest a unique nrec value for a given model. The best choice will depend on the model and your machine configuration.

You can optionally use the Nastran keyword ‘rdscale’ to modify the frequency range specified by an EIGRL bulk entry for eigensolutions of each substructure. Beginning in NX Nastran 8, the default value for ‘rdscale’ is 2.0, which is different from the default value of 3.5 that was used in previous versions of NX Nastran.

Models that use ACMS can easily be adapted to use RDMODES. To do so, follow these steps:

1. From the input file, remove the DOMAINSOLVER executive control statement.

2. At the command line, enter the Nastran keyword ‘nrec’ and, optionally, the appropriate Nastran keywords. Enter:

   - ‘smp’ or ‘dmp’ to specify the desired parallel functionality.

   - ‘rdscale’ to modify the frequency range.

In the DOMAINSOLVER executive control statement, you can use the values specified for NUMDOM to specify values for ‘nrec’ and the values specified for UPFACT to specify values for ‘rdscale’. For example, suppose that the input file for a job run using ACMS contains:

```
DOMAINSOLVER[ACMS(NUMDOM=128 UPFACT=2.5 TREE=MULTI)]
```

The equivalent Nastran keyword entry for an RDMODES run would be:

```
NASTRAN nrec=128 rdscale=2.5
```

RDMODES performance study

A performance study was completed using a vehicle model containing approximately 546,000 grid points, 500,000 elements, and 3,000,000 DOF. A SOL 103 normal modes analysis was performed that calculated modes up to 400 Hz using a workstation that had two quad-core Intel Nehalem 2.93GHz processors (for a total of 8 cores), two large local disks, and 18 GB of memory (RAM). The 8-core workstation was a single machine with multiple processors. Consequently, SMP was the natural choice for parallel implementation. However, it could also have run in DMP with properly assigned local disks.

The following table summarizes the results of solving the model using multi-level RDMODES for serial, SMP, and DMP configurations with both NX Nastran 7.1 and NX Nastran 8. For all runs, nrec = 256 and RDSCALE = 2.0.

Method	**RDMODES NXN 7.1**		**RDMODES NXN 8**		
Processor	Serial	DMP = 4	Serial	SMP = 8	DMP = 4
Elapsed time, min:sec	77:37	36:28	59:04	42:12	31:56
I/O, GB	2000.7	417.8	1002.1	987.3	343.5

As the tabular results clearly indicate:

- The RDMODES enhancements implemented in NX Nastran 8 significantly improve performance.

- When the available machine is either a distributed machine, a multi-core machine, or both, either parallel implementation gives better performance results than a serial configuration.

Chapter 10 Numerical Enhancements

Frequency response improvement

The FRRDRU module has been improved with additional support for structural damping. Structural damping is widely used in frequency response solutions (SOL 111). With the improvement, FRRDRU is efficient when the number of modes is larger than 2000, and the number of viscous damping elements is small.

FRRDRU is selected by setting system cell 462 to “2”. For example:

```
nastran sys462 = 2
```

When performing modal analysis on a large model, use the FRRDRU module in combination with RDMODES to obtain the best performance.

Performance Example

- Model: Car body
- Model size:

    1,096,000 grid points

    1,080,000 elements

    5,912,000 DOF

- Machine: 2.9 GHz Xeon with 7.8 GB memory
- Output:

    2,500 modes

    400 frequency responses

    285 subcases

- Parallel option: Shared memory parallel (SMP) (Parallel = 8)

Method	FRRD1 (default)	FRRDRU (sys462 = 2)
Elapsed time (min:sec)	75:08	20:18

Method	FRRD1 (default)	FRRDRU (sys462 = 2)
I/O (GB)	270.6	111.6
CPU (sec)(1)	32980	3287
(1) Because SMP is used, times are approximate.		

Real eigensolver factor caching

Beginning with NX Nastran 8, factor caching is introduced for the Lanczos real eigensolver.

A typical real eigensolution spends approximately equal time among factorization, FBS, and reorthogonalization. I/O cost from FBS can be reduced by storing the factor matrix in memory, provided sufficient free memory exists to store at least part of the factor. If not, requesting factor caching has no effect.

You can activate factor caching by setting system cell 492 to “1”. For example:

```
nastran sys492 = 1
```

Performance Example

- Model: Car body
- Model size:

  268,000 grid points

  275,000 elements

  1,584,000 DOF
- Machine: 2.9 GHz Xeon with 2.0 GB memory
- Output: Modes up to 200 Hz

sys492	0 (default)	1 (factor cached)
Elapsed time (min:sec)	94:02	86:21
I/O (GB)	2478.3	1247.2
CPU (sec)	4867.7	4433.2

Chapter

11 Output control for EPT and MPT data blocks

Output control for EPT and MPT data blocks

By default, the EPT and MPT data blocks are output to the .op2 file when you use any of the following parameters: PARAM,POST,-1, PARAM,POST,-2, PARAM,POST,-4. However, some external optimization algorithms require the output from the EPT and MPT data blocks to be disabled. Beginning with NX Nastan 8, you can:

- Disable the output from the EPT data block by including PARAM,OEPT,NO in either the case control or bulk data sections of the input file.

- Disable the output from the MPT data block by including PARAM,OMPT,NO in either the case control or bulk data sections of the input file.

The OEPT and OMPT parameters are valid for all solutions except SOL 601 and SOL 701.

OEPT parameter

The OEPT parameter can be placed in either the bulk data or case control sections of the input file and is valid for all solutions except SOL 601 and SOL 701.

OEPT	Default = YES By default, the EPT data block is output to the .op2 file when any of the following parameters are used: PARAM,POST,-1, PARAM,POST,-2, PARAM,POST,-4. PARAM,OEPT,NO can be used to disable the output of the EPT data block to the .op2 file.

OMPT parameter

The OMPT parameter can be placed in either the bulk data or case control sections of the input file and is valid for all solutions except SOL 601 and SOL 701.

OMPT	Default = YES By default, the MPT data block is output to the .op2 file when any of the following parameters are used: PARAM,POST,-1, PARAM,POST,-2, PARAM,POST,-4. PARAM,OMPT,NO can be used to disable the output of the MPT data block to the .op2 file.

Chapter

12 Upward compatibility

Updated data blocks

AXIC

Element property table.

Updated Record – BDYLIST(8915,89,213)

Fluid boundary list.

Note

The triplet for BDYLIST has changed from (8915,89,212) to (8915,89,213). This change is a documentation correction only.

AXIC68

Element property table (Pre-MSC Nastran Version 69).

Updated Record – BDYLIST(8915,89,213)

Fluid boundary list.

Note

The triplet for BDYLIST has changed from (8915,89,212) to (8915,89,213). This change is a documentation correction only.

CASECC

Case control information

Updated Record – REPEAT

Word	Name	Type	Description
......			
......			
333	EXSEGEOM	I	External superelement geometry flag (EXTSEOUT)
......			
......			
487	BGRESU	I	Glue results output
488	BGMEDIA	I	Glue results media code
489	UNDEF	None	
490	BGTYPE	I	Glue results type
491	RSVCOMP	I	Residual vector component flag
492	RSVOPTC	I	Residual vector component options
493	RSVSYST	I	Residual vector system flag
494	RSVOPTS	I	Residual vector system options
495	PLSLOC	I	Ply strain or stress locations
496	ELSMOP	I	ELSUM output option
497	UNDEF(103)	None	
......			
......			

DVPTAB

Designed property table

Updated Record – REPEAT

By ascending internal property identification number order. Type one properties are first and type two follow.

Word	Name	Type	Description
......			
......			
3	EPPNT	I	Property type (1 or 2 or 3) (Type 1 includes both DVPREL1 defined properties and internally generated PBAR/PBEAM properties that are linearly dependent on designed PBARL/PBEAML dimensions. Type 2 is for DVPREL2 defined properties. Type 3 is for internally generated PBAR/PBEAM properties that are nonlinearly dependent on designed PBARL/PBEAML dimensions.)
......			
......			
12	FREQ	RS	Frequency, if relevant, for which the property is defined

Updated Record – TRAILER

Word	Name	Type	Description
......			
2	NENT1	I	Number of Type 1 entries (see definition of EPPNT in “Repeat” record above)
3	NENT2	I	Number of Type 2 entries (see definition of EPPNT in “Repeat” record above)

Word	Name	Type	Description
4	NENT3	I	Number of Type 3 entries (see definition of EPPNT in “Repeat” record above)
5	UNDEF(2)	None	

Updated Notes:

There are as many records as there are designed properties. (NPROP = NENT1 + NENT2 + NENT3)

DYNAMIC

Table of Bulk Data entry images related to dynamics

New Record – NLRGAP(3707,37,556)

Defines a nonlinear transient radial gap.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	GA	I	Inner grid identification number
3	GB	I	Outer grid identification number
4	PLANE	I	Radial gap orientation plane (xy = 100; yz = 200, zx = 300)
5	TABK	I	Table identification number for gap stiffness vs. time
6	TABG	I	Table identification number for radial gap clearance
7	TABU	I	Table identification number for radial coefficient
8	RADIUS	RS	Shaft radius

EPT

Element property table.

New Record – MATCID(17006,170,901)

Defines material coordinate system for solid elements.

Word	Name	Type	Description
1	CID	I	Material coordinate system identification number
2	SPECOPT	I	Specification option
SPECOPT=1		Select individual element identification numbers	
3	EID	I	Element identification number
Word 3 repeats until -1 occurs			
SPECOPT=2		Select all element identification numbers	
3	ALL(2)	CHAR4	Keyword for selecting 'ALL' option
Words 3 repeats until -1 occurs			
SPECOPT=3		Select element identification numbers using a 'THRU' range without the 'BY' option	
3	EID	I	Element identification number
4	THRU(2)	CHAR4	Keyword for selecting 'THRU' option
6	EID	I	Element identification number
Words 3 through 5 repeat until -1 occurs			
SPECOPT=4		Select element identification numbers using a 'THRU' range with the 'BY' option	
3	EID	I	Element identification number
4	THRU(2)	CHAR4	Keyword for selecting 'THRU' option
6	EID	I	Element identification number

Word	Name	Type	Description
7	BY(2)	CHAR4	Keyword for selecting ‘BY’ option
9	N	I	Element selection increment
Words 3 through 9 repeat until -1 occurs			

New Record – PCOMPS(16006,160,903)

Defines the properties of an n-ply composite material laminate for solid elements.

Word	Name	Type	Description
1	PID	I	Property identification number
2	CORDM	I	Material coordinate system identification number
3	PSDIR	I	Stack and ply directions in the material coordinate system
4	SB	RS	Allowable shear stress of the bonding material
5	NB	RS	Allowable normal stress of the bonding material
6	TREF	RS	Reference temperature
7	GE	RS	Damping coefficient
8	UNDEF	None	
9	GPLYIDi	I	Global ply identification number
10	MID	I	Material identification number
11	TR	RS	Thicknesses of the ply
12	THETA	RS	Orientation angle of the longitudinal direction of the ply
13	FT	I	Failure theory

Word	Name	Type	Description
14	ILFT	I	Inter-laminar failure theory
15	SOUT	I	Stress or strain output request of the ply
16	TFLAG	I	Flag of ‘ABS’ or ‘REL’
Words 9 through 16 repeat until (-1,-1,-1,-1,-1,-1,-1,-1) occurs			

New Record – PLPLANE(4606,46,375)

Word	Name	Type	Description
1	PID	I	Property identification number
2	MID	I	Material identification number
3	CID	I	Coordinate system identification number
4	STR	CHAR4	Location of stress and strain output
5	T	RS	Default membrane thickness for Ti on the connection entry
6	UNDEF(6)	None	

Updated Record – PPLANE(3801,38,979)

Word	Name	Type	Description
……			
……			
5	FOROPT	I	Formulation option number
6	UNDEF(3)	None	

Note

The triplet for PPLANE was formerly incorrectly documented as (3801,38,978).

EPT705

Element property table (Pre-MSC Nastran 2001).

New Record – MATCID(17006,170,901)

Defines material coordinate system for solid elements.

Word	Name	Type	Description
1	CID	I	Material coordinate system identification number
2	SPECOPT	I	Specification option
SPECOPT=1		Select individual element identification numbers	
3	EID	I	Element identification number
Word 3 repeats until -1 occurs			
SPECOPT=2		Select all element identification numbers	
3	ALL(2)	CHAR4	Keyword for selecting 'ALL' option
Words 3 repeats until -1 occurs			
SPECOPT=3		Select element identification numbers using a 'THRU' range without the 'BY' option	
3	EID	I	Element identification number
4	THRU(2)	CHAR4	Keyword for selecting 'THRU' option
6	EID	I	Element identification number
Words 3 through 5 repeat until -1 occurs			
SPECOPT=4		Select element identification numbers using a 'THRU' range with the 'BY' option	
3	EID	I	Element identification number
4	THRU(2)	CHAR4	Keyword for selecting 'THRU' option
6	EID	I	Element identification number

Word	Name	Type	Description
7	BY(2)	CHAR4	Keyword for selecting 'BY' option
9	N	I	Element selection increment
Words 3 through 9 repeat until -1 occurs			

New Record – PCOMPS(16006,160,903)

Defines the properties of an n-ply composite material laminate for solid elements.

Word	Name	Type	Description
1	PID	I	Property identification number
2	CORDM	I	Material coordinate system identification number
3	PSDIR	I	Stack and ply directions in the material coordinate system
4	SB	RS	Allowable shear stress of the bonding material
5	NB	RS	Allowable normal stress of the bonding material
6	TREF	RS	Reference temperature
7	GE	RS	Damping coefficient
8	UNDEF	None	
9	GPLYIDi	I	Global ply identification number
10	MID	I	Material identification number
11	TR	RS	Thicknesses of the ply
12	THETA	RS	Orientation angle of the longitudinal direction of the ply
13	FT	I	Failure theory

Word	Name	Type	Description
14	ILFT	I	Inter-laminar failure theory
15	SOUT	I	Stress or strain output request of the ply
16	TFLAG	I	Flag of 'ABS' or 'REL'
Words 9 through 16 repeat until (-1,-1,-1,-1,-1,-1,-1,-1) occurs			

New Record – PLPLANE(4606,46,375)

Word	Name	Type	Description
1	PID	I	Property identification number
2	MID	I	Material identification number
3	CID	I	Coordinate system identification number
4	STR	CHAR4	Location of stress and strain output
5	T	RS	Default membrane thickness for Ti on the connection entry
6	UNDEF(6)	None	

Updated Record – PPLANE(3801,38,979)

Word	Name	Type	Description
......			
......			
5	FOROPT	I	Formulation option number
6	UNDEF(3)	None	

Note

The triplet for PPLANE was formerly incorrectly documented as (3801,38,978).

GEOM1

Table of Bulk Data entry images related to geometry.

New Record – CORD3G(501,5,43)

Word	Name	Type	Description
1	CID	I	Coordinate system identification number
2	METHOD(2)	CHAR4	Methods
4	FORM(2)	CHAR4	Forms
6	THETAID(3)	I	Identification number for DEQATN or TABLE
9	CIDREF	I	Coordinate system identification number

New Record – SELOAD(1127,11,461)

Word	Name	Type	Description
1	LID0	I	New mapped load set identification number to be used in the residual structure
2	SEID	I	Superelement identification number of the external superelement
3	LIDSE	I	Load set identification number in the external superelement

GEOM2

Table of Bulk Data entries related to element connectivity.

GEOM2 also contains information on scalar points. ECT is identical in format to GEOM2 except all grid and scalar point external identification numbers are replaced by internal numbers. Also, ECT does not contain SPOINT records.

New Record – CHEXAL(7708,77,9944)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	UNDEF	None	
4	G(20)	I	Grid point identification numbers of connection points

Note

The CHEXAL(7708,77,9944) record replaces the previously unused CHEXAL(7708,77,369) record.

New Record – CPENTAL(7108,71,9943)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	UNDEF	None	
4	G(15)	I	Grid point identification numbers of connection points

Changes to CPLSTNi and CPLSTSi Records

The triplet number for most of the plane strain and plane stress elements have changed.

Record	Old Triplet Number	New Triplet Number
CPLSTN3	(8801,88,980)	(1701,17,980)
CPLSTN4	(9301,93,981)	(5701,57,981)
CPLSTN6	(9401,94,982)	(5801,58,982)

Record	Old Triplet Number	New Triplet Number
CPLSTN8	(9501,95,983)	(7201,72,983)
CPLSTS3	(9601,96,984)	(8801,88,984)
CPLSTS4	(8401,84,985)	No change
CPLSTS6	(9801,98,986)	(1801,18,986)
CPLSTS8	(9901,99,987)	(3601,36,987)

GEOM3

Table of Bulk Data entry images related to static and thermal loads

Updated Record – BOLTLD(7601,76,608)

The triplet for BOLTLD was (7601,76,577).

New Record – PLOADE1(6701,67,978)

Word	Name	Type	Description
1	SID	I	Load set identification number
2	EID	I	Element identification number
3	PA	RS	Surface traction at grid point GA
4	PB	RS	Surface traction at grid point GB
5	G(2)	I	Corner grid point identification numbers
7	THETA	RS	Angle between surface traction and inward normal

GEOM4

Table of Bulk Data entry images related to constraints.

Table of Bulk Data entry images related to constraints, degree-of-freedom membership and rigid element connectivity.

New Record – RVDOF(9801,98,609)

DOF specification for residual vector computations.

Word	Name	Type	Description
1	ID	I	Grid or scalar point identification number
2	C	I	Component numbers

New Record – RVDOF1(9901,99,610)

DOF specification for residual vector computations.

Word	Name	Type	Description
1	C	I	Component numbers
2	THRUFLAG	I	Thru flag range
THRUFLAG = 0		No	
3	ID	I	Grid or scalar point identification number
THRUFLAG = 1		Yes	
3	ID1	I	First grid or scalar point identification number
4	ID2	I	Second grid or scalar point identification number
End THRUFLAG			

GEOM168

Table of Bulk Data entry images related to geometry (Pre-MSC Nastran Version 69).

New Record – CORD3G(501,5,43)

Word	Name	Type	Description
1	CID	I	Coordinate system identification number
2	METHOD(2)	CHAR4	Methods
4	FORM(2)	CHAR4	Forms
6	THETAID(3)	I	Identification number for DEQATN or TABLE
9	CIDREF	I	Coordinate system identification number

GPL

Grid point list.

Updated Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name
3	SEID	I	Superelement identification number

MPT

Table of Bulk Data entry images related to material properties

New Record – MATFT(3403,34,902)

Word	Name	Type	Description
1	MID	I	Material identification number
2	FTOPT	I	Failure theory option
FTOPT=1		Hill failure theory	

Word	Name	Type	Description
3	XT	RS	Allowable longitudinal stress in tension
4	XC	RS	Allowable longitudinal stress in compression
5	YT	RS	Allowable lateral stress in tension
6	YC	RS	Allowable lateral stress in compression
7	ZT	RS	Allowable transverse stress in tension
8	ZC	RS	Allowable transverse stress in compression
9	SXY	RS	Allowable shear stress in the xy-plane
10	SYZ	RS	Allowable shear stress in the yz-plane
11	SZX	RS	Allowable shear stress in the zx-plane
FTOPT=2		Hoffman failure theory	
3	XT	RS	Allowable longitudinal stress in tension
4	XC	RS	Allowable longitudinal stress in compression
5	YT	RS	Allowable lateral stress in tension
6	YC	RS	Allowable lateral stress in compression
7	ZT	RS	Allowable transverse stress in tension
8	ZC	RS	Allowable transverse stress in compression
9	SXY	RS	Allowable shear stress in the xy-plane
10	SYZ	RS	Allowable shear stress in the yz-plane
11	SZX	RS	Allowable shear stress in the zx-plane
FTOPT=3		Tsai-Wu failure theory	

Word	Name	Type	Description
3	XT	RS	Allowable longitudinal stress in tension
4	XC	RS	Allowable longitudinal stress in compression
5	YT	RS	Allowable lateral stress in tension
6	YC	RS	Allowable lateral stress in compression
7	ZT	RS	Allowable transverse stress in tension
8	ZC	RS	Allowable transverse stress in compression
9	SXY	RS	Allowable shear stress in the xy-plane
10	SYZ	RS	Allowable shear stress in the yz-plane
11	SZX	RS	Allowable shear stress in the zx-plane
12	F12	RS	Interaction term in the tensor polynomial theory
13	F23	RS	Interaction term in the tensor polynomial theory
14	F31	RS	Interaction term in the tensor polynomial theory
FTOPT=4		Maximum strain failure theory	
3	XT	RS	Allowable longitudinal strain in tension
4	XC	RS	Allowable longitudinal strain in compression
5	YT	RS	Allowable lateral strain in tension
6	YC	RS	Allowable lateral strain in compression
7	ZT	RS	Allowable transverse strain in tension
8	ZC	RS	Allowable transverse strain in compression

Word	Name	Type	Description
9	SXY	RS	Allowable shear strain in the xy-plane
10	SYZ	RS	Allowable shear strain in the yz-plane
11	SZX	RS	Allowable shear strain in the zx-plane
FTOPT=5		Maximum stress failure theory	
3	XT	RS	Allowable longitudinal stress in tension
4	XC	RS	Allowable longitudinal stress in compression
5	YT	RS	Allowable lateral stress in tension
6	YC	RS	Allowable lateral stress in compression
7	ZT	RS	Allowable transverse stress in tension
8	ZC	RS	Allowable transverse stress in compression
9	SXY	RS	Allowable shear stress in the xy-plane
10	SYZ	RS	Allowable shear stress in the yz-plane
11	SZX	RS	Allowable shear stress in the zx-plane
FTOPT=6		Maximum transverse shear stress failure theory	
3	SXZ	RS	Allowable shear stress in the xz-plane
4	SYZ	RS	Allowable shear stress in the yz-plane
Words 2 thru the last word of the selected option repeat until -1 is reached			

Updated Record – MATHE(7910,79,596)

MATHE format for Mooney-Rivlin (default) model (Model = Mooney in MATHE bulk entry):

Word	Name	Type	Description
......			

Word	Name	Type	Description
3	UNDEF	None	
....			
....			
11	UNDEF	None	
......			

MATHE format for Arruda-Boyce model (Model = Aboyce in MATHE bulk entry):

Word	Name	Type	Description
......			
3	UNDEF	None	
......			
......			
11	UNDEF	None	
......			

MATHE format for Sussman-Bathe model (Model=Sussbat in MATHE bulk entry):

Word	Name	Type	Description
......			
3	UNDEF	None	
......			
......			
12	UNDEF(5)	None	
......			

New Record – MATHEM(3501,35,977)

Word	Name	Type	Description
1	MID	I	Material identification number
2	R	RS	Material constant of Ogden-Roxburgh model for Mullins effect
3	M	RS	Material constant of Ogden-Roxburgh model for Mullins effect
4	HGEN	RS	Fraction of energy dissipated by Mullins effect model
5	UNDEF(4)	None	

New Record – MATHEV(3601,36,976)

Word	Name	Type	Description
1	MID	I	Material identification number
2	SHIFT	I	Time-temperature superposition shift law
3	C1	RS	Material constant for the WLF or Arrhenius shift functions
4	C2	RS	Material constant for the WLF or Arrhenius shift functions
5	TREF	RS	Reference temperature used by the WLF or Arrhenius shift functions
6	UNDEF(3)	None	
9	BETA	RS	Beta factor for chain of the viscoelastic model
10	TAU	RS	Relaxation time for chain of the viscoelastic model
11	HGEN	RS	Fraction of energy dissipation that is considered as heat generation

Word	Name	Type	Description
12	USAGE	I	Usage of chain
Words 9 through 12 repeat until -1 occurs			

New Record – MATVE(3701,37,975)

Word	Name	Type	Description
1	MID	I	Material identification number
2	GFUNC	I	TABVE ID for shear modulus relaxation function
3	KFUNC	I	TABVE ID for bulk modulus relaxation function
4	RHO	RS	Mass density
5	ALPHA	RS	Coefficient of thermal expansion
6	UNDEF(3)	None	
9	SHIFT	I	Time-temperature superposition shift law
10	C1	RS	Material constant for the WLF or Arrhenius shift functions
11	C2	RS	Material constant for the WLF or Arrhenius shift functions
12	T0	RS	Reference temperature used by the WLF or Arrhenius shift functions
13	UNDEF(4)	None	

OEF

Table of element forces

Also contains composite failure indices and analysis types (real and complex), and SORT1 and SORT2 formats.

Updated Record - IDENT

Word	Name	Type	Description
......			
......			
13	UNDEF	None	
14	Q4CSTR(C)	I	Corner Stress Flag
15	PLSLOC(C)	I	Ply stress/strain location
16	UNDEF(7)	None	
......			
......			

Updated Record - DATA

Word	Name	Type	Description
ELTYPE =267		Ply failure for HEXAL	
2	PLY	I	Lamina number
PLSLOC=0		Center option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	V1	RS	Failure value 1
7	V2	RS	Failure value 2
8	V3	RS	Failure value 3
9	V4	RS	Failure value 4
10	V5	RS	Failure value 5

Word	Name	Type	Description
11	V6	RS	Failure value 6
PLSLOC=1		Corner option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	V1	RS	Failure value 1
7	V2	RS	Failure value 2
8	V3	RS	Failure value 3
9	V4	RS	Failure value 4
10	V5	RS	Failure value 5
11	V6	RS	Failure value 6
Words 3 through 11 repeat 5 times			

Word	Name	Type	Description
ELTYPE =268		Ply failure for PENTAL	
2	PLY	I	Lamina number
PLSLOC=0		Center option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	V1	RS	Failure value 1
7	V2	RS	Failure value 2
8	V3	RS	Failure value 3

Word	Name	Type	Description
9	V4	RS	Failure value 4
10	V5	RS	Failure value 5
11	V6	RS	Failure value 6
PLSLOC=1		Corner option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	V1	RS	Failure value 1
7	V2	RS	Failure value 2
8	V3	RS	Failure value 3
9	V4	RS	Failure value 4
10	V5	RS	Failure value 5
11	V6	RS	Failure value 6
Words 3 through 11 repeat 4 times			

Word	Name	Type	Description
ELTYPE =269		Inter-laminar failure for HEXAL	
2	PLY	I	Lamina number
PLSLOC=0		Center option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	FSV	RS	Failure value

Word	Name	Type	Description
PLSLOC=1		Corner option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	FSV	RS	Failure value
Words 3 through 6 repeat 5 times			

Word	Name	Type	Description
ELTYPE =270		Inter-laminar failure for PENTAL	
2	PLY	I	Lamina number
PLSLOC=0		Center option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	FSV	RS	Failure value
PLSLOC=1		Corner option	
3	FLOC	CHAR4	Fiber location (BOT, MID, and TOP)
4	GRID	I	Edge grid ID (0=center)
5	THEORY	CHAR4	Failure theory
6	FSV	RS	Failure value
Words 3 through 6 repeat 4 times			

Updated Notes:

The comment regarding force output for CDAMPi and CVISC elements has been removed.

OES

Table of element stresses or strains

For all analysis types (real and complex) and SORT1 and SORT2 formats.

Updated Record - IDENT

Word	Name	Type	Description
....			
....			
15	PLSLOC(C)	I	Ply stress/strain location
16	UNDEF(35)	None	
....			
....			

Updated Record - DATA

ELTYPE =269		Composite HEXA element (CHEXAL) – Center and corners	
1	PLY	I	Lamina number
SCODE,6=0		Strain	
PLSLOC=0		Center option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	EX1	RS	Normal strain in the 1-direction
5	EY1	RS	Normal strain in the 2-direction

6	ET1	RS	Shear strain in the 12-plane
7	EZ1	RS	Normal strain in the 3-direction
8	EL1	RS	Shear strain in the 13-plane
9	EL2	RS	Shear strain in the 23-plane
10	ETMAX1	RS	Von Mises strain
PLSLOC=1		Corner option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	EX1	RS	Normal strain in the 1-direction
5	EY1	RS	Normal strain in the 2-direction
6	ET1	RS	Shear strain in the 12-plane
7	EZ1	RS	Normal strain in the 3-direction
8	EL1	RS	Shear strain in the 13-plane
9	EL2	RS	Shear strain in the 23-plane
10	ETMAX1	RS	Von Mises strain
Words 2 through 10 repeat 5 times			
SCODE,6=1		Stress	
PLSLOC=0		Center option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	SX1	RS	Normal stress in the 1-direction
5	SY1	RS	Normal stress in the 2-direction
6	ST1	RS	Shear stress in the 12-plane

7	SZ1	RS	Normal stress in the 3-direction
8	SL1	RS	Shear stress in the 13-plane
9	SL2	RS	Shear stress in the 23-plane
10	STMAX1	RS	Von Mises stress
PLSLOC=1		Corner option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	SX1	RS	Normal stress in the 1-direction
5	SY1	RS	Normal stress in the 2-direction
6	ST1	RS	Shear stress in the 12-plane
7	SZ1	RS	Normal stress in the 3-direction
8	SL1	RS	Shear stress in the 13-plane
9	SL2	RS	Shear stress in the 23-plane
10	STMAX1	RS	Von Mises stress
Words 2 through 10 repeat 5 times			
End SCODE,6			

ELTYPE =270		Composite PENTA element (CPENTAL) – Center and corners	
1	PLY	I	Lamina number
SCODE,6=0		Strain	
PLSLOC=0		Center option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)

4	EX1	RS	Normal strain in the 1-direction
5	EY1	RS	Normal strain in the 2-direction
6	ET1	RS	Shear strain in the 12-plane
7	EZ1	RS	Normal strain in the 3-direction
8	EL1	RS	Shear strain in the 13-plane
9	EL2	RS	Shear strain in the 23-plane
10	ETMAX1	RS	Von Mises strain
PLSLOC=1		Corner option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	EX1	RS	Normal strain in the 1-direction
5	EY1	RS	Normal strain in the 2-direction
6	ET1	RS	Shear strain in the 12-plane
7	EZ1	RS	Normal strain in the 3-direction
8	EL1	RS	Shear strain in the 13-plane
9	EL2	RS	Shear strain in the 23-plane
10	ETMAX1	RS	Von Mises strain
Words 2 through 10 repeat 4 times			
SCODE,6=1		Stress	
PLSLOC=0		Center option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	SX1	RS	Normal stress in the 1-direction

5	SY1	RS	Normal stress in the 2-direction
6	ST1	RS	Shear stress in the 12-plane
7	SZ1	RS	Normal stress in the 3-direction
8	SL1	RS	Shear stress in the 13-plane
9	SL2	RS	Shear stress in the 23-plane
10	STMAX1	RS	Von Mises stress
PLSLOC=1		Corner option	
2	FLOC	CHAR4	Fiber location (BOT, MID, TOP)
3	GRID	I	Edge grid ID (center=0)
4	SX1	RS	Normal stress in the 1-direction
5	SY1	RS	Normal stress in the 2-direction
6	ST1	RS	Shear stress in the 12-plane
7	SZ1	RS	Normal stress in the 3-direction
8	SL1	RS	Shear stress in the 13-plane
9	SL2	RS	Shear stress in the 23-plane
10	STMAX1	RS	Von Mises stress
Words 2 through 10 repeat 4 times			
End SCODE,6			

ELTYPE =271		Triangle plane strain (CPLSTN3) – Center	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	EX	RS	Normal strain in x
3	EY	RS	Normal strain in y

4	EZ	RS	Normal strain in z
5	EXZ	RS	Shear strain in xz
6	EMAX	RS	Von Mises strain
TCODE,7=1		Real / Imaginary	
2	EXR	RS	Normal strain in x – real part
3	EXI	RS	Normal strain in x – imaginary part
4	EYR	RS	Normal strain in y – real part
5	EYI	RS	Normal strain in y – imaginary part
6	EZR	RS	Normal strain in z – real part
7	EZI	RS	Normal strain in z – imaginary part
8	EXYR	RS	Shear strain in xz – real part
9	EXYI	RS	Shear strain in xz – imaginary part
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	SX	RS	Normal stress in x
3	SY	RS	Normal stress in y
4	SZ	RS	Normal stress in z
5	SXZ	RS	Shear stress in xz
6	SMAX	RS	Von Mises stress
TCODE,7=1		Real / Imaginary	
2	SXR	RS	Normal stress in x – real part
3	SXI	RS	Normal stress in x – imaginary part

4	SYR	RS	Normal stress in y – real part
5	SYI	RS	Normal stress in y – imaginary part
6	SZR	RS	Normal stress in z – real part
7	SZI	RS	Normal stress in z – imaginary part
8	SXYR	RS	Shear stress in xz – real part
9	SXYI	RS	Shear stress in xz – imaginary part
End TCODE,7			
End SCODE,6			

ELTYPE =272		Quadrilateral plane strain (CPLSTN4) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z
7	EXZ	RS	Shear strain in xz
8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	"CEN"

3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y
6	SZ	RS	Normal stress in z
7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	

2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part
11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
End SCODE,6			

ELTYPE =273		Triangle plane strain (CPLSTN6) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z

7	EXZ	RS	Shear strain in xz
8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 4 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part
Words 3 through 11 repeat 4 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y

6	SZ	RS	Normal stress in z
7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 4 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part
11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 4 times			
End TCODE,7			
End SCODE,6			

ELTYPE =274		Quadrilateral plane strain (CPLSTN8) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”

3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z
7	EXZ	RS	Shear strain in xz
8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	

2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y
6	SZ	RS	Normal stress in z
7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part
11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
End SCODE,6			

ELTYPE =275		Triangle plane stress (CPLSTS3) – Center	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	EX	RS	Normal strain in x
3	EY	RS	Normal strain in y
4	EZ	RS	Normal strain in z
5	EXZ	RS	Shear strain in xz
6	EMAX	RS	Von Mises strain
TCODE,7=1		Real / Imaginary	
2	EXR	RS	Normal strain in x – real part
3	EXI	RS	Normal strain in x – imaginary part
4	EYR	RS	Normal strain in y – real part
5	EYI	RS	Normal strain in y – imaginary part
6	EZR	RS	Normal strain in z – real part
7	EZI	RS	Normal strain in z – imaginary part
8	EXYR	RS	Shear strain in xz – real part
9	EXYI	RS	Shear strain in xz – imaginary part
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	SX	RS	Normal stress in x
3	SY	RS	Normal stress in y
4	SZ	RS	Normal stress in z

5	SXZ	RS	Shear stress in xz
6	SMAX	RS	Von Mises stress
TCODE,7=1		Real / Imaginary	
2	SXR	RS	Normal stress in x – real part
3	SXI	RS	Normal stress in x – imaginary part
4	SYR	RS	Normal stress in y – real part
5	SYI	RS	Normal stress in y – imaginary part
6	SZR	RS	Normal stress in z – real part
7	SZI	RS	Normal stress in z – imaginary part
8	SXYR	RS	Shear stress in xz – real part
9	SXYI	RS	Shear stress in xz – imaginary part
End TCODE,7			
End SCODE,6			

ELTYPE =276		Quadrilateral plane stress (CPLSTS4) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z
7	EXZ	RS	Shear strain in xz

8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y
6	SZ	RS	Normal stress in z

7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part
11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
End SCODE,6			

ELTYPE =277		Triangle plane stress (CPLSTS6) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”

3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z
7	EXZ	RS	Shear strain in xz
8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 4 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part
Words 3 through 11 repeat 4 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	

2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y
6	SZ	RS	Normal stress in z
7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 4 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	"CEN"
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part
11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 4 times			
End TCODE,7			
End SCODE,6			

ELTYPE =278		Quadrilateral plane stress (CPLSTS8) – Center and Corners	
SCODE,6=0		Strain	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EX	RS	Normal strain in x
5	EY	RS	Normal strain in y
6	EZ	RS	Normal strain in z
7	EXZ	RS	Shear strain in xz
8	EMAX	RS	Von Mises strain
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	EXR	RS	Normal strain in x – real part
5	EXI	RS	Normal strain in x – imaginary part
6	EYR	RS	Normal strain in y – real part
7	EYI	RS	Normal strain in y – imaginary part
8	EZR	RS	Normal strain in z – real part
9	EZI	RS	Normal strain in z – imaginary part
10	EXZR	RS	Shear strain in xz – real part
11	EXZI	RS	Shear strain in xz – imaginary part

Words 3 through 11 repeat 5 times			
End TCODE,7			
SCODE,6=1		Stress	
TCODE,7=0		Real	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SX	RS	Normal stress in x
5	SY	RS	Normal stress in y
6	SZ	RS	Normal stress in z
7	SXZ	RS	Shear stress in xz
8	SMAX	RS	Von Mises stress
Words 3 through 8 repeat 5 times			
TCODE,7=1		Real / Imaginary	
2	TERM	CHAR4	“CEN”
3	GRID	I	Grid identification number; 0 for centroid
4	SXR	RS	Normal stress in x – real part
5	SXI	RS	Normal stress in x – imaginary part
6	SYR	RS	Normal stress in y – real part
7	SYI	RS	Normal stress in y – imaginary part
8	SZR	RS	Normal stress in z – real part
9	SZI	RS	Normal stress in z – imaginary part
10	SXZR	RS	Shear stress in xz – real part

11	SXZI	RS	Shear stress in xz – imaginary part
Words 3 through 11 repeat 5 times			
End TCODE,7			
End SCODE,6			

OQG

Table of single or multipoint constraint forces. Also contact force results from SOL 101, SOL 601,106, SOL 601,129 and SOL 701.

For all analysis types (real and complex), and SORT1 and SORT2 formats. Contact force results are real only.

Updated Record - DATA

Word	Name	Type	Description
......			
......			
Data Format when Table Code = 67 (OQG glue force results for SOL 101 (ACODE=1) and SOLs 103 and 105 (ACODE=2))			
1	EKEY	I	Device code + 10*Point identification number
2	TYPE	I	Point type, Grid or Scalar (always Grid for glue force)
3	QF1	RS	Glue force in direction X (Base C.S.)
4	QF2	RS	Glue force in direction Y (Base C.S.)
5	QF3	RS	Glue force in direction Z (Base C.S.)
6	QM1	RS	Not used
7	QM2	RS	Not used
8	QM2	RS	Not used

Word	Name	Type	Description
Repeat word 1-8 for each grid point.			

OUGMC

Table of modal contributions for displacements, velocities, accelerations.

For all analysis types (real and complex), and SORT1 and SORT2 formats.

Updated Record - IDENT

Word	Name	Type	Description
......			
TCODE,1 = 02		Sort 2	
PANFLG = 1		Panel data	
6	PNAME(2)	CHAR4	Panel name
PANFLG = 0		Not panel data	
6	MODE	I	Mode number (0 for TOTAL; -1 for constraint modes)
7	MFREQ	RS	Modal frequency (Hz)
End PANFLG			
End TCODE,1			
8	DATTYP	I	Data Type (1=displacement, 2=velocity, 3=acceleration)
9	FCODE	I	Format Code
10	NUMWDE	I	Number of words per entry in DATA record
11	NOMC(C)	I	Number of modal contribution data sets
12	NUMPAN(C)	I	Number of panels

Word	Name	Type	Description
13	PANFLG	I	Panel flag (1 = panel data; 0 = mode data)
14	UNDEF(37)	None	
......	......	......	......

Updated Record - DATA

Note

The OUGMC data record has been changed to allow for panel contributions. In addition, the documentation for the OUGMC data record has been extensively revised to better facilitate interpretation.

Word	Name	Type	Description
TCODE,1 = 01		Sort 1	
1	MODE	I	Mode number (0 for TOTAL)
2	MFREQ	RS	Modal frequency (Hz)
TCODE,7 = 0 or 2		Real	
3	MC	RS	Modal contribution
4	MODE	I	Mode number (-1 for constraint modes)
5	MFREQ	RS	Modal frequency (Hz)
6	MC	RS	Modal contribution
Words 4–6 repeat NOMC times			
End TCODE,7 = 0 or 2			
TCODE,7 = 1		Real/ Imaginary	
3	MCR	RS	Modal contribution – real part
4	MCI	RS	Modal contribution – imaginary part

Word	Name	Type	Description
5	MODE	I	Mode number (-1 for constraint modes)
6	MFREQ	RS	Modal frequency (Hz)
7	MCR	RS	Modal contribution – real part
8	MCI	RS	Modal contribution – imaginary part
Words 5–8 repeat NOMC times			
The remaining definitions only exist if NUMPAN > 0			
9	MODE	I	Mode number (-1 for constraint modes)
10	MFREQ	RS	Modal frequency (Hz)
11	MCR	RS	Modal contribution - real part
12	MCI	RS	Modal contribution - imaginary part
13	PNAME(2)	CHAR4	Panel name
15	PCR	RS	Panel contribution – real part
16	PCI	RS	Panel contribution – imaginary part
Words 13–16 repeat NUMPAN times			
Words 9–16 repeat NOMC times			
End TCODE,7 = 1			
TCODE,1 = 02		Sort 2	
ACODE,4 = 05		Frequency response	
1	FREQ	RS	Frequency (Hz)
2	UNDEF	None	
ACODE,4 = 06		Transient response	
1	TIME	RS	Time step

Word	Name	Type	Description
2	UNDEF	None	
ACODE,4 = 09		Complex eigenvalues	
1	EIGR	RS	Eigenvalue – real part
2	EIGI	RS	Eigenvalue – imaginary part
End ACODE,4			
TCODE,7 = 0 or 2		Real	
3	MC	RS	Modal contribution
TCODE,7 = 1		Real/ Imaginary	
3	MCR	RS	Modal contribution – real part
4	MCI	RS	Modal contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2 = 10		Velocity	
TCODE,7 = 0 or 2		Real	
3	MC	RS	Modal contribution
TCODE,7 = 1		Real/ Imaginary	
3	MCR	RS	Modal contribution – real part
4	MCI	RS	Modal contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2 = 11		Acceleration	

Word	Name	Type	Description
TCODE,7 = 0 or 2		Real	
3	MC	RS	Modal contribution
TCODE,7 = 1		Real/ Imaginary	
3	MCR	RS	Modal contribution – real part
4	MCI	RS	Modal contribution – imaginary part
End TCODE,7			

New data blocks

OBG

Output glue normal and in-plane tractions for SOL 101, SOL 103 and SOL 105. SORT1 and SORT2 formats are supported.

Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name, for example, OBG1

Record – IDENT

Word	Name	Type	Description
1	ACODE(C)	I	Device code + 10* Approach Code
2	TCODE(C)	I	Table code (66 for Glue tractions)
3	UNDEF	None	
4	SUBCASE	I	
TCODE=1		SORT1	

Word	Name	Type	Description
ACODE=01		Statics	
5	LSDVMN	I	Load set number
6	UNDEF(2)	None	
ACODE=02		Real Eigenvalues	
5	MODE	I	Mode number
6	EIGN	RS	Eigenvalue
7	MODECYCL	RS	Mode or cycle
ACODE=03		Differential stiffness 0	
5	LSDVMN	I	Load set number
6	UNDEF(2)	None	
ACODE=04		Differential stiffness 1	
5	LSDVMN	I	Load set number
6	UNDEF(2)	None	
ACODE=05		Frequency	
5	FREQ	RS	Frequency
6	UNDEF(2)	None	
ACODE=06		Transient	
5	TIME	RS	Time step
6	UNDEF(2)	None	
ACODE=07		Buckling Phase 0 (pre-buckling)	
5	LSDVMN	I	Load set number
6	UNDEF(2)	None	

Word	Name	Type	Description
ACODE=08		Buckling Phase 1 (post-buckling)	
5	LSDVMN	I	Mode number
6	EIGN	RS	Eigenvalue
7	UNDEF	None	
ACODE=09		Complex eigenvalues	
5	MODE	I	Mode
6	EIGR	RS	Eigenvalue (real)
7	EIGI	RS	Eigenvalue (imaginary)
ACODE=10		Nonlinear statics	
5	LFTSFQ	RS	Time step
6	UNDEF(2)	None	
ACODE=11		Old geometric nonlinear statics	
5	LSDVMN	I	Load set number
6	UNDEF(2)	None	
ACODE=12		CONTRAN (may appear as ACODE = 6)	
5	TIME	RS	Time
6	UNDEF(2)	None	
End ACODE			
TCODE=2		SORT2	
5	LSDVMN	I	Load set, mode number
6	UNDEF(2)	None	
End TCODE			

Word	Name	Type	Description
8	LSDVMN	I	Load set
9	FCODE	I	Format code
10	NUMWDE	I	Number of words per entry in DATA record
11	UNDEF	None	
12	PID	I	Physical property identification number
13	UNDEF(38)	None	
51	TITLE(32)	CHAR4	Title
83	SUBTITL(32)	CHAR4	Subtitle
115	LABEL(32)	CHAR4	Label

Record – DATA

Word	Name	Type	Description
TCODE=1		SORT1	
1	EKEY	I	Device code + 10 * point identification number
TCODE=2		SORT2	
ACODE=01		SOL 101 Linear Statics	
1	LOADSET	I	Load set or zero
End ACODE			
End TCODE			
2	P	RS	Glue normal traction
3	T1	RS	Glue tangential traction in direction X (Base C.S.)

Word	Name	Type	Description
4	T2	RS	Glue tangential traction in direction Y (Base C.S.)
5	T3	RS	Glue tangential traction in direction Z (Base C.S.)
Repeat word 1-5 for each grid point.			

Record – TRAILER

Word	Name	Type	Description
1	UNDEF(6)	None	

Notes:

1. Glue results are grid point based results.
2. Glue tangential traction is expressed as a vector data (magnitude and direction) in X, Y and Z components of Basic Coordinate System.

OUGRC

Table of reciprocal panel contributions.

For frequency analysis types (complex), and SORT1 and SORT2 formats.

Record - HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name
3	WORD	I	No Def or Month, Year, One, One
Word 3 repeats until End of Record			

Record - IDENT

Word	Name	Type	Description
1	ACODE(C)	I	Device code + 10*Approach Code
2	TCODE(C)	I	Table Code
3	PCODE	I	Reciprocal panel contributions code: 1=absolute (modal), 2=normalized (modal), -1=absolute (direct), -2=normalized (direct)
4	SUBCASE	I	Subcase number
5	DCODE	I	Acoustic dof code (10*grid ID + direction)
TCODE,1=01		Sort 1	
ACODE,4=05		Frequency	
6	FREQ	RS	Frequency (Hz)
7	UNDEF	None	
End ACODE,4			
TCODE,1=02		Sort 2	
6	PNAME(2)	CHAR4	Panel name (0 for TOTAL)
End TCODE,1			
8	DATTYP	I	Data Type (1=displacement, 2=velocity, 3=acceleration)
9	FCODE	I	Format Code
10	NUMWDE	I	Number of words per entry in DATA record
11	UNDEF(40)	None	
51	TITLE(32)	CHAR4	Title

Word	Name	Type	Description
83	SUBTITL(32)	CHAR4	Subtitle
115	LABEL(32)	CHAR4	Label

Record - DATA

Word	Name	Type	Description
TCODE,1=01		Sort 1	
1	PNAME(2)	CHAR4	Panel name (0 for TOTAL)
TCODE,1=02		Sort 2	
1	FREQ	RS	Frequency (Hz)
2	UNDEF	None	
End TCODE,1			
TCODE,2=01		Displacement	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Reciprocal panel contribution – real part
4	PCI	RS	Reciprocal panel contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2=02		Velocity	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Reciprocal panel contribution – real part

Word	Name	Type	Description
4	PCI	RS	Reciprocal panel contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2=03		Acceleration	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Reciprocal panel contribution – real part
4	PCI	RS	Reciprocal panel contribution – imaginary part
End TCODE,7			

Record - TRAILER

Word	Name	Type	Description
1	NREC	I	Number of records
2	UNDEF(5)	None	

DISTL

Direct table input table for RS

Contains the load set labels on DTI bulk data entries.

Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name

Record – REPEAT

Word	Name	Type	Description
1	ID	I	Load set identification number
2	STRING(2)	CHAR4	String label
Words 2 and 3 repeat until End of Record			

Record – TRAILER

Word	Name	Type	Description
1	WORD1	I	= 32767
2	WORD2	I	= 32767
3	WORD3	I	= 32767
4	WORD4	I	= 32767
5	WORD5	I	= 32767
6	WORD6	I	= 32767

DESCYC

Defines the design cycle to which the results data blocks that follow belong, up to the next design cycle data block (i.e. the next DESCYC).

Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data Block Name

Record – DATA

Word	Name	Type	Description
1	DESCYCL	I	Design cycle number

Word	Name	Type	Description
2	CONDIS(2)	CHAR4	Design cycle type; “D” for discretized design cycle; Blank for continuous design cycle

Record – TRAILER

Word	Name	Type	Description
1	“1”	I	Always “1”
2	DESCYCL	I	Design cycle number
3	CYCTYP	I	Design cycle type; “0” for a continuous design cycle; “1” for a discretized design cycle
4	UNDEF(3)	None	

Previously undocumented data blocks

EDOM

SOL 200 design optimization and sensitivity analysis bulk entries.

Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name

Record – DCONADD(5106,51,471)

Design constraint set combinations.

Word	Name	Type	Description
1	DCID	I	Design constraint set identification number

Word	Name	Type	Description
2	DCi	I	DCONSTR entry identification number
Word 2 repeats until -1 occurs			

Record – DCONSTR(4106,41,362)

Design constraints.

Word	Name	Type	Description
1	DCID	I	Design constraint set identification number
2	RID	I	DRESPi entry identification number
3	LALLOW	RS	Lower bound on the response quantity. Undefined if LTID is nonzero
4	UALLOW	RS	Upper bound on the response quantity. Undefined if UTID is nonzero
5	LOWFQ	RS	Low end of frequency range in Hz
6	HIGHFQ	RS	High end of frequency range in Hz
7	LTID	I	Identification number of TABLEDi entry giving lower bound on the response quantity as a function of frequency or 0 if not specified
8	UTID	I	Identification number of TABLEDi entry giving upper bound on the response quantity as a function of frequency or 0 if not specified

Record – DDVAL(7000,70,563)

Discrete design variable values.

Word	Name	Type	Description
1	ID	I	Unique discrete value set identification number
2	DVALi	RS	Discrete values for the variable
Word 2 repeats until -1 occurs			

Record – DESVAR(3106,31,352)

Design variables.

Word	Name	Type	Description
1	ID	I	Unique design variable identification number
2	LABEL(2)	CHAR4	User-supplied name for printing purposes
4	XINIT	RS	Initial value
5	XLB	RS	Lower bound
6	XUB	RS	Upper bound
7	DELXV	RS	Fractional change allowed for the design variable during approximate optimization
8	DDVAL	I	ID of a DDVAL entry that provides a set of allowable discrete values

Record – DLINK(3206,32,353)

Multiple design variable linking.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	DDVID	I	Dependent design variable identification number
3	C0	RS	Constant term

Word	Name	Type	Description
4	CMULT	RS	Constant multiplier
5	IDVi	I	Independent design variable identification number
6	Ci	RS	Coefficient corresponding to IDV
Words 5 and 6 repeat until -1 occurs			

Record – DOPTPRM(4306,43,364)

Design optimization parameters.

Word	Name	Type	Description
1	APRCOD	I	Approximation method
2	IPRINT	I	Print control during approximate optimization phase with DOT
3	DESMAX	I	Maximum number of design cycles
4	METHOD	I	DOT optimization method
5	DELP	RS	Fractional change allowed in each property during any optimization design cycle
6	DPMIN	RS	Minimum move limit imposed
7	PTOL	RS	Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry
8	CONV1	RS	Relative objective function convergence criterion
9	CONV2	RS	Absolute objective function convergence criterion

Word	Name	Type	Description
10	GMAX	RS	Maximum constraint violation allowed at the converged optimum
11	DELX	RS	Fractional change allowed in each design variable during any optimization cycle
12	DXMIN	RS	Minimum absolute limit on design variable move
13	DELB	RS	Relative finite difference move parameter
14	GSCAL	RS	Constraint normalization factor
15	CONVDV	RS	Relative convergence criterion on design variables
16	CONVPR	RS	Relative convergence criterion on properties
17	P1	I	Design cycles in which output is printed
18	P2	I	Items to be printed at the design cycles defined by P1
19	CT	RS	Constraint tolerance
20	CTMIN	RS	Constraint violation threshold
21	DABOBJ	RS	DOT absolute objective function convergence criterion
22	DELOBJ	RS	DOT relative objective function convergence criterion
23	DOBJ1	RS	DOT 1–D search absolute objective limit
24	DOBJ2	RS	DOT 1–D search relative objective limit
25	DX1	RS	DOT 1–D search absolute DV limit
26	DX2	RS	DOT 1–D search relative DV limit
27	ISCAL	I	Design variables are rescaled every ISCAL iterations

Word	Name	Type	Description
28	ITMAX	I	Maximum DOT MFD iterations per cycle
29	ITRMOP	I	Maximum consecutive DOT MFD iterations at convergence
30	IWRITE	I	File number for DOT optimizer printout
31	IGMAX	I	Active constraint counter
32	JTMAX	I	Maximum DOT SLP iterations per cycle
33	ITRMST	I	Maximum consecutive DOT SLP iterations at convergence
34	JPRINT	I	SLP subproblem print within DOT
35	IPRNT1	I	Print scaling factors for design variable vector within DOT
36	IPRNT2	I	DOT 1–D search or miscellaneous information print
37	JWRITE	I	File number on which iteration history is written within DOT
38	STPSCL	RS	Scale factor for shape finite difference step sizes applied to all shape design variables
39	FSDMAX	I	Number of FSD cycles to be performed
40	FSDALP	RS	Relaxation parameter applied in FSD
41	DISCOD	I	Discrete processing method code
42	DISBEG	I	Design cycle ID for discrete variable processing initiation
43	PLVIOL	I	Flag for handling property limit violation
44	P2RSET	I	ID of a SET1 entry listing constrained responses to be printed if retained

Record – DRESP1(3806,38,359)

First level design response quantities. (If applicable, see the *Quick Reference Guide* for detailed information on response attributes.)

Word	Name	Type	Description
1	ID	I	Unique entry identifier
2	LABEL(2)	CHAR4	User-defined label
4	FLAG	I	Flag indicating response type
FLAG = 1		WEIGHT	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	MONE	I	Entry is -1
FLAG = 2		VOLUME	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	MONE	I	Entry is -1
FLAG = 3		LAMA	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute

Word	Name	Type	Description
10	MONE	I	Entry is -1
FLAG = 4		EIGN	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	MONE	I	Entry is -1
FLAG = 5		DISP	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 6		STRESS	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			

Word	Name	Type	Description
FLAG = 7		STRAIN	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 8		FORCE	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 9		CFAILURE	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or composite property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute

Word	Name	Type	Description
10	ATTi	I	Element numbers (if Word 5 is ELEM) or composite property IDs
Word 10 repeats until -1 occurs			
FLAG = 10		CSTRESS	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or composite property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or composite property IDs
Word 10 repeats until -1 occurs			
FLAG = 11		CSTRAIN	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or composite property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or composite property IDs
Word 10 repeats until -1 occurs			
FLAG = 12		FREQ	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	I	Response attribute
10	MONE	I	Entry is -1
FLAG = 13		SPCFORCE	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 14		ESE	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 15		CEIG	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute

Word	Name	Type	Description
10	MONE	I	Entry is -1
FLAG = 20		FRDISP	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 21		FRVELO	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 22		FRACCL	
5	UNDEF(2)	None	

Word	Name	Type	Description
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 23		FRSPCF	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 24		FRSTRE	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 25		FRFORC	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 26		RMSDISP	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Random ID
10	ATTi	I	Grid point IDs

Word	Name	Type	Description
Word 10 repeats until -1 occurs			
FLAG = 27		RMSVELO	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Random ID
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 28		RMSACCL	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Random ID
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 29		PSDDISP	
5	UNDEF	None	
6	PTYPE	I	Random ID
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 30		PSDVELO	
5	UNDEF	None	
6	PTYPE	I	Random ID
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 31		PSDACCL	
5	UNDEF	None	
6	PTYPE	I	Random ID
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Frequency value; -1 (integer) spawn for all frequencies in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 60		TDISP	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 61		TVELO	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 62		TACCL	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 63		TSPCF	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Grid point IDs
Word 10 repeats until -1 occurs			
FLAG = 64		TSTRE	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 65		TFORC	
5	PTYPE(2)	CHAR4	Element flag (ELEM) or property entry name
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute

Word	Name	Type	Description
9	ATTB	RS	Time value; -1 (integer) spawn for all time steps in set; -1.10000E+08 for SUM; -1.20000E+08 for AVG; -1.30000E+08 for SSQ; -1.40000E+08 for RSS; -1.50000E+08 for MAX; -1.60000E+08 for MIN
10	ATTi	I	Element numbers (if Word 5 is ELEM) or property IDs
Word 10 repeats until -1 occurs			
FLAG = 81		DIVERG	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATT1	RS	Mach number
11	MONE	I	Entry is -1
FLAG = 82		TRIM	
5	UNDEF(2)	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Entry is 0
10	ATT1	I	Entry is 0
11	MONE	I	Entry is -1
FLAG = 83		STABDER	
5	UNDEF(2)	None	

Word	Name	Type	Description
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATT1	I	Component
11	MONE	I	Entry is -1
FLAG = 84		FLUTTER	
5	METHOD	CHAR4	Analysis type: PK or PKNL
6	UNDEF	None	
7	REGION	I	Region identifier for constraint screening
8	ATTA	I	Response attribute
9	ATTB	I	Response attribute
10	ATTi	I	ATT1 is the identification number of a SET1 entry that specifies a set of modes; ATT2 is the identification number of an FLFACT entry that specifies a list of densities; ATT3 is the identification number of an FLFACT entry that specifies a list of Mach numbers; ATT4 is the identification number of an FLFACT entry that specifies a list of velocities
Word 10 repeats until -1 occurs			

Record – DRESP2(3906,39,360)

Design equation response quantities.

Word	Name	Type	Description
1	ID	I	Unique entry identifier
2	LABEL(2)	CHAR4	User-defined label

Word	Name	Type	Description
4	EQID	I	DEQATN entry identification number if integer > 0. Function to be applied to arguments (FUNC) if integer < 0: -2 for AVG; -3 for SSQ; -4 for RSS; -5 for MAX; -6 for MIN
5	REGION	I	Region identifier for constraint screening
6	FLAG	I	
FLAG = 1000		DESVAR	
7	DVIDi	I	DESVAR entry identification number
Word 7 repeats until -1000			
FLAG = 2000		DTABLE	
7	LABLi(2)	CHAR4	Label for a constant in the DTABLE entry
Word 7 repeats until -2000			
FLAG = 3000		DFRFNC	
7	DFRFIDi	I	Identification number for a DFRFNC record in the DTI, DFRFNC table
Word 7 repeats until -3000			
FLAG = 4000		DRESP1	
7	NRi	I	DRESP1 entry identification number
Word 7 repeats until -4000			
FLAG = 5000		DNODE	
7	Gi	I	Grid point identification number
8	Ci	I	Component number of grid point
Words 7 and 8 repeat until -5000			

Word	Name	Type	Description
FLAG = 6000		DVPREL1	
7	DPIPi	I	DVPREL1 entry identification number
Word 7 repeats until -6000			
FLAG = 7000		DVCREL1	
7	DCICi	I	DVCREL1 entry identification number
Word 7 repeats until -7000			
FLAG = 8000		DVMREL1	
7	DMIMi	I	DVMREL1 entry identification number
Word 7 repeats until -8000 occurs			
FLAG = 9000		DVPREL2	
7	DPI2Pi	I	DVPREL2 entry identification number
Word 7 repeats until -9000 occurs			
FLAG = 10000		DVCREL2	
7	DCI2Ci	I	DVCREL2 entry identification number
Word 7 repeats until -10000 occurs			
FLAG = 11000		DVMREL2	
7	DMI2Mi	I	DVMREL2 entry identification number
Word 7 repeats until -11000 occurs			
FLAG = 12000		DRESP2	
7	NRRi	I	Nested DRESP2 entry identification number
Word 7 repeats until -12000 occurs			
End entry when -1 occurs			

Record – DRESP3(6700,67,433)

Externally computed design responses.

Word	Name	Type	Description
1	ID	I	Unique entry identifier
2	LABEL(2)	CHAR4	User-defined label referenced by a constraint bulk entry or by the objective function
4	GROUP(2)	CHAR4	Selects a specific external response routine
6	TYPE(2)	CHAR4	Refers to a specific user-created response calculation type in the external function evaluator
8	REGION	I	Region identifier for constraint screening
9	FLAG	I	
FLAG = 1000		DESVAR	
10	DVIDi	I	DESVAR entry identification number
Word 10 repeats until -1000			
FLAG = 2000		DTABLE	
10	LABLi(2)	CHAR4	Label for a constant in the DTABLE entry
Word 10 repeats until -2000			
FLAG = 3000		DFRFNC	
10	DFRFIDi	I	Identification number for a DFRFNC record in the DTI, DFRFNC table
Word 10 repeats until -3000			
FLAG = 4000		DRESP1	
10	NRi	I	DRESP1 entry identification number

Word	Name	Type	Description
Word 10 repeats until -4000			
FLAG = 5000		DNODE	
10	Gi	I	Grid point identification number
11	Ci	I	Component number of grid point
Words 10 and 11 repeat until -5000			
FLAG = 6000		DVPREL1	
10	DPIPi	I	DVPREL1 entry identification number
Word 10 repeats until -6000			
FLAG = 7000		DVCREL1	
10	DCICi	I	DVCREL1 entry identification number
Word 10 repeats until -7000			
FLAG = 8000		DVMREL1	
10	DMIMi	I	DVMREL1 entry identification number
Word 10 repeats until -8000 occurs			
FLAG = 9000		DVPREL2	
10	DPI2Pi	I	DVPREL2 entry identification number
Word 10 repeats until -9000 occurs			
FLAG = 10000		DVCREL2	
10	DCI2Ci	I	DVCREL2 entry identification number
Word 10 repeats until -10000 occurs			
FLAG = 11000		DVMREL2	
10	DMI2Mi	I	DVMREL2 entry identification number

Word	Name	Type	Description
Word 10 repeats until -11000 occurs			
FLAG = 12000		DRESP2	
10	NRRi	I	Nested DRESP2 entry identification number
Word 10 repeats until -12000 occurs			
FLAG = 13000		USRDATA	
10	USRDATA(2)	CHAR4	Character strings in user-defined format for any use in the evaluator
Words 10 and 11 repeat until -13000 occurs			
End entry when -1 occurs			

Record – DSCREEN(4206,42,363)

Design constraint screening data.

Word	Name	Type	Description
1	RTYPE	I	Response type for which the screening criteria apply
2	TRS	RS	Truncation threshold
3	NSTR	I	Maximum number of constraints to be retained per region per load case

Record – DTABLE(3706,37,358)

Table constants.

Word	Name	Type	Description
1	LABLi(2)	CHAR4	Label for the constant
3	VALUi	RS	Value of the constant

Word	Name	Type	Description
Words 1 thru 3 repeat until -1 occurs			

Record – DVBSHAP(5806,58,474)

Design variable to boundary shapes.

Word	Name	Type	Description
1	DVID	I	Design variable identification number of a DESVAR entry
2	AUXMID	I	Auxiliary model identification number
3	COLi	I	Load sequence identification number from AUXMODEL command
4	SFi	RS	Scaling factor for load sequence identification number
Words 1 thru 4 repeat for i = 2 and 3			

Record – DVCREL1(6100,61,429)

Design variable to connectivity property relation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of an element connectivity entry
4	EID	I	Element identification number
5	FID	I	Entry is 0
6	CPMIN	RS	Minimum value allowed for this property
7	CPMAX	RS	Maximum value allowed for this property
8	C0	RS	Constant term of relation
9	CPNAME(2)	CHAR4	Name of connectivity property

Word	Name	Type	Description
11	DVIDi	I	DESVAR entry identification number
12	COEFi	RS	Coefficient of linear relation
Words 11 and 12 repeat until -1 occurs			

Record – DVCREL2(6200,62,430)

Design variable to connectivity property relation based on a user-supplied equation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of an element connectivity entry
4	EID	I	Element identification number
5	FID	I	Entry is 0
6	CPMIN	RS	Minimum value allowed for this property
7	CPMAX	RS	Maximum value allowed for this property
8	EQID	I	DEQATN entry identification number
9	CPNAME(2)	CHAR4	Name of connectivity property
11	FLAG	I	DESVAR/DTABLE
FLAG = 1000		DESVAR	
12	DVIDi	I	A DESVAR entry identification number
Word 12 repeats until -1000			
FLAG = 2000		DTABLE	
12	LABLi(2)	CHAR4	Label for a constant on the DTABLE entry
Words 12 and 13 repeat until -2000			
End flag when -1 occurs			

Record – DVGEOM(5906,59,356)

Design variable to geometry relation.

Word	Name	Type	Description
1	DVID	I	DESVAR entry identification number
2	ENTITY(2)	CHAR4	Designed entity
4	ID	I	Identification number of the parent entity specified by ENTITY
5	IDPRTB	I	Identification number of the perturbed entity specified by ENTITY

Record – DVGRID(4406,44,372)

Design variable to grid point relation.

Word	Name	Type	Description
1	DVID	I	DESVAR entry identification number
2	GID	I	Grid point or geometric point identification number
3	CID	I	Coordinate system identification number
4	COEFF	RS	Multiplier of the vector defined by N(3)
5	N1	RS	Component of the vector measured in the coordinate system defined by CID
6	N2	RS	Component of the vector measured in the coordinate system defined by CID
7	N3	RS	Component of the vector measured in the coordinate system defined by CID

Record – DVMREL1(6300,63,431)

Design variable to material relation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of a material property entry
4	MID	I	Material identification number
5	FID	I	Entry is 0
6	MPMIN	RS	Minimum value allowed for this property
7	MPMAX	RS	Maximum value allowed for this property
8	C0	RS	Constant term of relation
9	MPNAME(2)	CHAR4	Name of material property
11	DVIDi	I	DESVAR entry identification number
12	COEFi	RS	Coefficient of linear relation
Words 11 and 12 repeat until -1 occurs			

Record – DVMREL2(6400,64,432)

Design variable to material relation based on a user-supplied equation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of a material property entry
4	MID	I	Material identification number
5	FID	I	Entry is 0
6	MPMIN	RS	Minimum value allowed for this property
7	MPMAX	RS	Maximum value allowed for this property
8	EQID	I	DEQATN entry identification number
9	MPNAME(2)	CHAR4	Name of material property

Word	Name	Type	Description
11	FLAG	I	DESVAR/DTABLE
FLAG = 1000		DESVAR	
12	DVIDi	I	A DESVAR entry identification number
Word 12 repeats until -1000			
FLAG = 2000		DTABLE	
12	LABLi(2)	CHAR4	Label for a constant on the DTABLE entry
Words 12 and 13 repeat until -2000			
End flag when -1 occurs			

Record – DVPREL1(3306,33,354)

Design variable to property relation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of a property entry
4	PID	I	Property entry identification number
5	FID	I	FID number input. Otherwise, either 0 if property name is input, or frequency (RS) if entry is for frequency dependent property. (See Words 9 and 10)
6	PMIN	RS	Minimum value allowed for this property
7	PMAX	RS	Maximum value allowed for this property
8	C0	RS	Constant term of relation
9	PNAME1	CHAR4	First word of property name, if any, or blanks if FID number is nonzero in Word 5

Word	Name	Type	Description
10	PNAME2	CHAR4	Second word of property name, if any. Otherwise, either blanks if FID number is nonzero in Word 5, or frequency (RS) if entry is for frequency dependent property. (See Word 5)
11	DVIDi	I	DESVAR entry identification number
12	COEFi	RS	Coefficient of linear relation
Words 11 and 12 repeat until -1 occurs			

Record – DVPREL2(3406,34,355)

Design variable to property relation based on a user-supplied equation.

Word	Name	Type	Description
1	ID	I	Unique identification number
2	TYPE(2)	CHAR4	Name of a property entry
4	PID	I	Property entry identification number
5	FID	I	FID number input. Otherwise, either 0 if property name is input, or frequency (RS) if entry is for frequency dependent property. (See Words 9 and 10)
6	PMIN	RS	Minimum value allowed for this property
7	PMAX	RS	Maximum value allowed for this property
8	EQID	I	DEQATN entry identification number
9	PNAME1	CHAR4	First word of property name, if any, or blanks if FID number is nonzero in Word 5

Word	Name	Type	Description
10	PNAME2	CHAR4	Second word of property name, if any. Otherwise, either blanks if FID number is nonzero in Word 5, or frequency (RS) if entry is for frequency dependent property. (See Word 5)
11	FLAG	I	DESVAR/DTABLE
FLAG = 1000		DESVAR	
12	DVIDi	I	A DESVAR entry identification number
Word 12 repeats until -1000			
FLAG = 2000		DTABLE	
12	LABLi(2)	CHAR4	Label for a constant on the DTABLE entry
Words 12 and 13 repeat until -2000			
End flag when -1 occurs			

Record – DVSHAP(5006,50,470)

Design variable to basis vectors.

Word	Name	Type	Description
1	DVID	I	Design variable identification number on the DESVAR entry
2	COLi	I	Column number of the displacement matrix
3	SFi	RS	Scaling factor applied to the COL column of the displacement matrix
Words 1 thru 3 repeat until (-1,-1,-1) occurs			

Record – MAT1DOM(103,1,9944)

If one or more properties from a MAT1 entry are used as design variables, the MAT1DOM record is written to the EDOM data block. This is different than the MAT1 record in the MPT data block.

Word	Name	Type	Description
1	MID	I	MAT1 identification number
2	FTE	I	Format code for Young's modulus
3	FTG	I	Format code for shear modulus
4	FTNU	I	Format code for Poisson's ratio

Record – MAT10DOM(2801,28,9945)

If one or more properties from a MAT10 entry are used as design variables, the MAT10DOM record is written to the EDOM data block. This is different than the MAT10 record in the MPT data block.

Word	Name	Type	Description
1	MID	I	MAT10 identification number
2	FTBULK	I	Format code for bulk modulus
3	FTRHO	I	Format code for mass density
4	FTC	I	Format code for speed of sound

Record – MODTRAK(6006,60,477)

Mode tracking parameters.

Word	Name	Type	Description
1	SID	I	Set identification number that is selected with a MODTRAK command
2	LOWRNG	I	Lowest mode number in range to search
3	HIGHRNG	I	Highest mode number in range to search

Word	Name	Type	Description
4	MTFILTER	RS	Filtering parameter used in mode cross-orthogonality check

Record – EODB(65535,65535,65535)

End of data block.

Record – TRAILER

Word	Name	Type	Description
1	BIT(6)	I	Record presence trailer words

EDT

Aero and element deformations.

Record – HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Data block name

Record – ACMODL(5201,52,373)

Fluid-structure interface modeling parameters.

Word	Name	Type	Description
1	INTER(2)	CHAR4	IDENT or DIFF method specification
3	INFO(2)	CHAR4	Allowable values are ALL, ELEMENTS, PID, SET3, and NONE
5	FSET	I	Fluid set ID
6	SSET	I	Structure set ID
7	NORML	RS	Outward normal search distance to detect fluid-structure interface

Word	Name	Type	Description
8	METHOD(2)	CHAR4	Interface calculation method
10	OVPLANG	RS	Angular tolerance in degrees used to decide whether a fluid free face and a structural face are overlapping
11	SRCHUNIT(2)	CHAR4	Search unit
13	INTOL	RS	Inward normal search distance to detect fluid-structure interface
14	AREAOPT	I	Area option
15	SKNEPS	RS	SKNEPS option. Only used when AREAOPT = 0
16	INTORD	I	Integration order

Record – ADAPT(6301,63,397)

Version adaptivity control.

Word	Name	Type	Description
1	SID	I	Adapt entry identification number selected by ADAPT case control command
2	ADGEN	I	Identification number of the first PVAL entry generated in the adaptive process
3	MAXITER	I	Number of analyses performed before adaptive process is stopped
4	PSTRTID	I	Identification number of the PVAL entry describing the starting p-order distribution
5	PMINID	I	Identification number of the PVAL entry describing the minimum p-order distribution

Word	Name	Type	Description
6	PMAXID	I	Identification number of the PVAL entry describing the maximum p-order distribution
7	PART(2)	CHAR4	Part name of the elements defined in ELSET and controlled by TYPE, ERREST, ERRTOL, SIGTOL, and EPSTOL
9	ELSET	I	Identification number of the SET command under the SETS DEFINITION command
10	TYPE	CHAR4	p-order adjustment
11	ERREST	I	Error estimator activation flag
12	ERRTOL	RS	Error tolerance
13	SIGTOL	RS	Stress tolerance
14	EPSTOL	RS	Strain tolerance
15	UNDEF(4)	None	
Words 7 thru 18 repeat until -1 occurs			

Record – AECOMP(7801,78,582)

Component for an integrated load monitor point.

Word	Name	Type	Description
1	NAME(2)	CHAR4	Character string identifying the component
3	LISTTYPE(2)	CHAR4	CAERO or AELIST for aerodynamic components and SET1 for structural components
5	LISTID	I	CAERO, AELIST, or SET1 entry identification number

Word	Name	Type	Description
Word 5 repeats until -1 occurs			

Record – AECOMPL(7901,79,583)

Component for an integrated load monitor point.

Word	Name	Type	Description
1	NAME(2)	CHAR4	Character string identifying the component
3	LABEL(2)	CHAR4	Character string identifying other components defined by AECOMP or AECOMPL entries
Words 3 and 4 repeat until (-1,-1) occurs			

Record – AEDW(7301,73,574)

Parametric normal wash loading for aerodynamics.

Word	Name	Type	Description
1	MACH	RS	Mach number
2	SYMXZ(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
4	SYMXY(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
6	UXID	I	The identification number of a UXVEC entry
7	DMIJ(2)	CHAR4	The name of a DMI or DMIJ entry that defines the downwash
9	DMIJI(2)	CHAR4	The name of a DMIJI entry that defines the CAERO2 interference element downwashes

Record – AEFACT(4002,40,273)

Aerodynamic lists.

Word	Name	Type	Description
1	SID	I	Set identification number
2	D	RS	Number
Word 2 repeats until -1 occurs			

Record – AEFORCE(7501,75,576)

Parametric force for aerodynamics.

Word	Name	Type	Description
1	MACH	RS	Mach number
2	SYMXZ(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
4	SYMXY(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
6	UXID	I	The identification number of a UXVEC entry
7	MESH(2)	CHAR4	Character string identifying whether the force is defined on the aerodynamic mesh or structural mesh. Allowable values are AERO and STRUCT
9	FORCE	I	The identification number of a FORCE/MOMENT set
10	DMIK(2)	CHAR4	The name of a DMIK entry that defines the aerodynamic force vector

Record – AELINK(2602,26,386)

Links aeroelastic variables.

Word	Name	Type	Description
1	ID	I	Trim set identification number
2	LABLD(2)	CHAR4	Character string identifying the dependent aerodynamic variable
4	LABL(2)	CHAR4	Character string identifying the aerodynamic variable
6	C	RS	Linking coefficient for the aerodynamic variable
Words 4 thru 6 repeat until (-1,-1,-1) occurs			

Record – AELIST(2302,23,341)

Aerodynamic element list.

Word	Name	Type	Description
1	SID	I	Set identification number
2	E	I	List of aerodynamic boxes generated by CAERO1 entries to define a surface
Word 2 repeats until -1 occurs			

Record – AEPARM(7001,70,571)

General controller for use in trim.

Word	Name	Type	Description
1	ID	I	Controller identification number
2	LABEL(2)	CHAR4	Controller name
4	UNIT(2)	CHAR4	Label used to describe the units of the controller values

Record – AEPRESS(7401,74,575)

Parametric pressure loading for aerodynamics.

Word	Name	Type	Description
1	MACH	RS	Mach number
2	SYMXZ(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
4	SYMXY(2)	CHAR4	Character string for identifying symmetry of the force vector. Allowable values are SYMM, ASYMM, and ANTI
6	UXID	I	The identification number of a UXVEC entry
7	DMIJ(2)	CHAR4	The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure
9	DMIJI(2)	CHAR4	The name of a DMIJI entry that defines the CAERO2 interference element downwashes

Record – AERO(3202,32,265)

Aerodynamic physical data.

Word	Name	Type	Description
1	ACSID	I	Aerodynamic coordinate system identification number
2	VELOCITY	RS	Velocity for aerodynamic force data recovery and to calculate the BOV parameter
3	REFC	RS	Reference length for reduced frequency
4	RHOREF	RS	Reference density
5	SYMXZ	I	Symmetry key for the aero coordinate xz plane

Word	Name	Type	Description
6	SYMXY	I	Symmetry key for the aero coordinate xy plane that can be used to simulate ground effects

Record – AEROS(2202,22,340)

Static aeroelasticity physical data.

Word	Name	Type	Description
1	ACSID	I	Aerodynamic coordinate system identification number
2	RCSID	I	Reference coordinate system identification number for rigid body motions
3	REFC	RS	Reference chord length
4	REFB	RS	Reference span
5	REFS	RS	Reference wing area
6	SYMXZ	I	Symmetry key for the aero coordinate xz plane
7	SYMXY	I	Symmetry key for the aero coordinate xy plane that can be used to simulate ground effects

Record – AESTAT(2102,21,339)

Static aeroelasticity trim variables.

Word	Name	Type	Description
1	ID	I	Identification number of an aerodynamic trim variable dof
2	LABEL(2)	CHAR4	An alphanumeric string identifying the dof

Record – AESURF(2002,20,338)

Aerodynamic control surface.

Word	Name	Type	Description
1	ID	I	Controller identification number
2	LABEL(2)	CHAR4	Controller name
4	CID1	I	Identification number of a rectangular coordinate system with a y-axis defining the hinge line of the control surface component
5	ALID1	I	Identification number of an AELIST entry identifying all aerodynamic elements that make up the control surface component
6	CID2	I	Identification number of a rectangular coordinate system with a y-axis defining the hinge line of the control surface component
7	ALID2	I	Identification number of an AELIST entry identifying all aerodynamic elements that make up the control surface component
8	EFF	RS	Control surface effectiveness. Default = 1.0
9	LDW	I	Linear downwash flag
10	CREFC	RS	Reference chord length for the control surface
11	CREFS	RS	Reference surface area for the control surface
12	PLLIM	RS	Lower deflection limit for the control surface in rad
13	PULIM	RS	Upper deflection limit for the control surface in rad

Word	Name	Type	Description
14	HMLLIM	RS	Lower hinge moment limit for the control surface in force-length units
15	HMULIM	RS	Upper hinge moment limit for the control surface in force-length units
16	TQLLIM	RS	Set identification number of TABLED entry providing the lower deflection limit for the control surface as a function of dynamic pressure
17	TQULIM	RS	Set identification number of TABLED entry providing the upper deflection limit for the control surface as a function of dynamic pressure

Record – AESURFS(7701,77,581)

Aerodynamic control surface.

Word	Name	Type	Description
1	ID	I	Identification number of an aerodynamic trim variable DOF
2	LABEL(2)	CHAR4	Control surface name
4	LIST1	I	Identification number of a SET1 entry that contains the grid IDs associated with this control surface
5	LIST2	I	Identification number of a SET1 entry that contains the grid IDs associated with this control surface

Record – BOLT(7108,71,251)

Bolt definition.

Word	Name	Type	Description
1	BID	I	Bolt identification number

Word	Name	Type	Description
2	FLAG	I	
FLAG = 1			
3	BMID	I	Beam identification number
Word 3 repeats until -1			
FLAG = 2		Not supported – for future use	
3	CID	I	Bolt direction component number
4	G1	I	First grid point defining bolt direction
5	G2	I	Second grid point defining bolt direction
6	EID	I	Solid element identification number
Word 6 repeats until -1			

Record – CAERO1(3002,30,263)

Aerodynamic panel element connection.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number of a PAERO1 entry
3	CP	I	Coordinate system for locating points 1 and 4
4	NSPAN	I	Number of spanwise boxes
5	NCHORD	I	Number of chordwise boxes
6	LSPAN	I	Identification number of an AEFACT entry containing a list of division points for spanwise boxes

Word	Name	Type	Description
7	LCHORD	I	Identification number of an AEFACT entry containing a list of division points for chordwise boxes
8	IGID	I	Interference group identification number
9	X1	RS	X-coordinate of point 1 in coordinate system CP
10	Y1	RS	Y-coordinate of point 1 in coordinate system CP
11	Z1	RS	Z-coordinate of point 1 in coordinate system CP
12	X12	RS	Edge chord length in aerodynamic coordinate system
13	X4	RS	X-coordinate of point 4 in coordinate system CP
14	Y4	RS	Y-coordinate of point 4 in coordinate system CP
15	Z4	RS	Z-coordinate of point 4 in coordinate system CP
16	X43	RS	Edge chord length in aerodynamic coordinate system

Record – CAERO2(4301,43,167)

Aerodynamic body connection.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number of a PAERO2 entry
3	CP	I	Coordinate system for locating point 1

Word	Name	Type	Description
4	NSP	I	Number of slender body elements
5	NINT	I	Number of interference elements
6	LSP	I	Identification number of an AEFACT entry for slender body division points
7	LINT	I	Identification number of an AEFACT entry containing a list of division points for interference elements
8	IGID	I	Interference group identification number
9	X1	RS	X-coordinate of point 1 in coordinate system CP
10	Y1	RS	Y-coordinate of point 1 in coordinate system CP
11	Z1	RS	Z-coordinate of point 1 in coordinate system CP
12	X12	RS	Length of body in the x-direction of the aerodynamic coordinate system
13	UNDEF(4)	None	

Record – CAERO3(4401,44,168)

Aerodynamic panel element configuration.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number of a PAERO3 entry
3	CP	I	Coordinate system for locating points 1 and 4

Word	Name	Type	Description
4	LISTW	I	Identification number of an AEFACT entry that lists coordinate pairs for structural interpolation of the wing
5	LISTC1	I	Identification number of an AEFACT entry that lists coordinate pairs for control surfaces
6	LISTC2	I	Identification number of an AEFACT entry that lists coordinate pairs for control surfaces
7	UNDEF(2)	None	
9	X1	RS	X-coordinate of point 1 in coordinate system CP
10	Y1	RS	Y-coordinate of point 1 in coordinate system CP
11	Z1	RS	Z-coordinate of point 1 in coordinate system CP
12	X12	RS	Edge chord length in aerodynamic coordinate system
13	X4	RS	X-coordinate of point 4 in coordinate system CP
14	Y4	RS	Y-coordinate of point 4 in coordinate system CP
15	Z4	RS	Z-coordinate of point 4 in coordinate system CP
16	X43	RS	Edge chord length in aerodynamic coordinate system

Record – CAERO4(4501,45,169)

Aerodynamic macro-strip element connection.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number of a PAERO4 entry
3	CP	I	Coordinate system for locating points 1 and 4
4	NSPAN	I	Number of strips
5	LSPAN	I	Identification number of an AEFACT entry containing a list of division points for strips
6	UNDEF(3)	None	
9	X1	RS	X-coordinate of point 1 in coordinate system CP
10	Y1	RS	Y-coordinate of point 1 in coordinate system CP
11	Z1	RS	Z-coordinate of point 1 in coordinate system CP
12	X12	RS	Edge chord length in aerodynamic coordinate system
13	X4	RS	X-coordinate of point 4 in coordinate system CP
14	Y4	RS	Y-coordinate of point 4 in coordinate system CP
15	Z4	RS	Z-coordinate of point 4 in coordinate system CP
16	X43	RS	Edge chord length in aerodynamic coordinate system

Record – CAERO5(5001,50,175)

Aerodynamic panel element configuration.

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number of a PAERO5 entry
3	CP	I	Coordinate system for locating points 1 and 4
4	NSPAN	I	Number of strips
5	LSPAN	I	Identification number of an AEFACT entry containing a list of division points for strips
6	NTHRY	I	Parameter to select Piston or van Dyke's theory
7	NTHICK	I	Parameter to select thickness integrals input
8	UNDEF	None	
9	X1	RS	X-coordinate of point 1 in coordinate system CP
10	Y1	RS	Y-coordinate of point 1 in coordinate system CP
11	Z1	RS	Z-coordinate of point 1 in coordinate system CP
12	X12	RS	Edge chord length in aerodynamic coordinate system
13	X4	RS	X-coordinate of point 4 in coordinate system CP
14	Y4	RS	Y-coordinate of point 4 in coordinate system CP
15	Z4	RS	Z-coordinate of point 4 in coordinate system CP

Word	Name	Type	Description
16	X43	RS	Edge chord length in aerodynamic coordinate system

Record – CLOAD(6201,62,143)

Static load combination for superelement loads.

Word	Name	Type	Description
1	CID	I	Combination identification number
2	S	RS	Scale factor
3	SI	RS	Scale factors
4	IDVI	I	Identification numbers of load vectors
Words 3 and 4 repeat until (-1,-1) occurs			

Record – CSSCHD(6401,64,307)

Aerodynamic control surface schedule input.

Word	Name	Type	Description
1	SID	I	Set identification number
2	AESID	I	Identification number of an AESURF entry to which the schedule is attached
3	LALPHA	I	Identification number of an AEFACT entry containing a list of angles of attack at which schedule information is provided
4	LMACH	I	Identification number of an AEFACT entry containing a list of Mach numbers at which schedule information is provided

Word	Name	Type	Description
5	LSCHD	I	Identification number of an AEFACT entry which contains the scheduling information

Record – DEFORM(104,1,81)

Static element deformation.

Word	Name	Type	Description
1	SID	I	Deformation set identification number
2	EID	I	Element number
3	D	RS	Deformation

Record – DIVERG(2702,27,387)

Divergence analysis data.

Word	Name	Type	Description
1	SID	I	Unique set identification number
2	NROOT	I	Number of divergence roots to output
3	M	RS	Mach number
Word 3 repeats until -1 occurs			

Record – FLFACT(4102,41,274)

Aerodynamic physical data.

Word	Name	Type	Description
1	SID	I	Set identification number
2	F	RS	Aerodynamic factor
Word 2 repeats until -1 occurs			

Record – FLUTTER(3902,39,272)

Aerodynamic flutter data.

Word	Name	Type	Description
1	SID	I	Set identification number
2	METHOD(2)	CHAR4	Flutter analysis method
4	DENS	I	Identification number of an FLFACT entry specifying density ratios for flutter analysis
5	MACH	I	Identification number of an FLFACT entry specifying Mach numbers for flutter analysis
6	RFREQ	I	Identification number of an FLFACT entry specifying reduced frequencies for flutter analysis
7	IMETH(2)	CHAR4	Interpolation method for aerodynamic matrix interpolation
9	NEIGN	I	Number of eigenvalues beginning with the first eigenvalue
10	EPS	RS	Convergence parameter
Words 1 thru 10 repeat until -1 occurs			

Record – ITER(5808,58,220)

Iterative solver options.

Word	Name	Type	Description
1	SID	I	Set identification number
2	PRECOND	I	Preconditioner option for global and element iterative solution
3	CONV	I	Convergence criterion

Word	Name	Type	Description
4	MSGFLG	I	Message flag
5	ITSEPS	RS	User-given convergence parameter epsilon
6	ITSMAX	I	Maximum number of iterations
7	IPAD	I	Global iterative solution padding value
8	IEXT	I	Global iterative solution extraction level in reduced or block incomplete Cholesky factorization
9	PREFONLY	I	Specifies early termination of the global iterative solver
10	ZPIVOT	RS	Singularity tolerance adjustment during the preconditioner phase of an element iterative solution

Record – MKAERO1(3802,38,271)

Mach number and reduced frequency table.

Word	Name	Type	Description
1	M(16)	RS	List of from 1 to 8 Mach numbers and their corresponding 1 to 8 reduced frequencies

Record – MKAERO2(3702,37,270)

Mach number and reduced frequency table.

Word	Name	Type	Description
1	M1	RS	Mach number
2	K1	RS	Reduced frequency
3	M2	RS	Mach number

Word	Name	Type	Description
4	K2	RS	Reduced frequency
5	M3	RS	Mach number
6	K3	RS	Reduced frequency
7	M4	RS	Mach number
8	K4	RS	Reduced frequency

Record – MONPNT1(7601,76,577)

Integrated load monitor point.

Word	Name	Type	Description
1	NAME(2)	CHAR4	Character string identifying the monitor point
3	LABEL(14)	CHAR4	Character string identifying and labeling the monitor point
17	AXES	I	Component axes about which to sum
18	COMP(2)	CHAR4	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined
20	CID	I	Coordinate system identification number
21	X	RS	X-coordinate in the CID coordinate system
22	Y	RS	Y-coordinate in the CID coordinate system
23	Z	RS	Z-coordinate in the CID coordinate system

Record – PAERO1(3102,31,264)

Aerodynamic panel property.

Word	Name	Type	Description
1	PID	I	Property identification number referenced by a CAERO1 entry
2	B1	I	Identification number of CAERO2 entries for an associated body
3	B2	I	Identification number of CAERO2 entries for an associated body
4	B3	I	Identification number of CAERO2 entries for an associated body
5	B4	I	Identification number of CAERO2 entries for an associated body
6	B5	I	Identification number of CAERO2 entries for an associated body
7	B6	I	Identification number of CAERO2 entries for an associated body
8	UNDEF	None	

Record – PAERO2(4601,46,170)

Aerodynamic body properties.

Word	Name	Type	Description
1	PID	I	Property identification number
2	ORIENT	CHAR4	Orientation flag
3	UNDEF	None	
4	WIDTH	RS	Reference half-width of body and the width of the constant width interference tube
5	AR	RS	Aspect ratio of the interference tube (height/width)

Word	Name	Type	Description
6	LRSB	I	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the slender body elements
7	LRIB	I	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the interference elements
8	LTH1	I	Identification number of an AEFACT entry defining theta arrays for interference calculations
9	LTH2	I	Identification number of an AEFACT entry defining theta arrays for interference calculations
10	THI1	I	The first interference element of a body to use the theta1 array; others use the theta2 array
11	THN1	I	The last interference element of a body to use the theta1 array; others use the theta2 array
12	THI2	I	The first interference element of a body to use the theta1 array; others use the theta2 array
13	THN2	I	The last interference element of a body to use the theta1 array; others use the theta2 array
14	THI3	I	The first interference element of a body to use the theta1 array; others use the theta2 array
15	THN3	I	The last interference element of a body to use the theta1 array; others use the theta2 array

Record – PAERO3(4701,47,171)

Aerodynamic panel property.

Word	Name	Type	Description
1	PID	I	Property identification number
2	NBOX	I	Number of Mach boxes in the flow direction
3	FLAG	I	
FLAG = 0			
4	UNDEF	None	
5	X5	RS	X-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry
6	Y5	RS	Y-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry
7	X6	RS	X-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
8	Y6	RS	Y-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
FLAG = 1			
4	UNDEF	None	
5	X5	RS	X-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry

Word	Name	Type	Description
6	Y5	RS	Y-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry
7	X6	RS	X-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
8	Y6	RS	Y-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
9	X7	RS	X-coordinate of point 7 in the aerodynamic coordinate system defining the cranks and control surface geometry
10	Y7	RS	Y-coordinate of point 7 in the aerodynamic coordinate system defining the cranks and control surface geometry
11	X8	RS	X-coordinate of point 8 in the aerodynamic coordinate system defining the cranks and control surface geometry
12	Y8	RS	Y-coordinate of point 8 in the aerodynamic coordinate system defining the cranks and control surface geometry
13	X9	RS	X-coordinate of point 9 in the aerodynamic coordinate system defining the cranks and control surface geometry
14	Y9	RS	Y-coordinate of point 9 in the aerodynamic coordinate system defining the cranks and control surface geometry

Word	Name	Type	Description
15	X10	RS	X-coordinate of point 10 in the aerodynamic coordinate system defining the cranks and control surface geometry
16	Y10	RS	Y-coordinate of point 10 in the aerodynamic coordinate system defining the cranks and control surface geometry
FLAG = 2			
4	UNDEF	None	
5	X5	RS	X-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry
6	Y5	RS	Y-coordinate of point 5 in the aerodynamic coordinate system defining the cranks and control surface geometry
7	X6	RS	X-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
8	Y6	RS	Y-coordinate of point 6 in the aerodynamic coordinate system defining the cranks and control surface geometry
9	X7	RS	X-coordinate of point 7 in the aerodynamic coordinate system defining the cranks and control surface geometry
10	Y7	RS	Y-coordinate of point 7 in the aerodynamic coordinate system defining the cranks and control surface geometry
11	X8	RS	X-coordinate of point 8 in the aerodynamic coordinate system defining the cranks and control surface geometry

Word	Name	Type	Description
12	Y8	RS	Y-coordinate of point 8 in the aerodynamic coordinate system defining the cranks and control surface geometry
13	X9	RS	X-coordinate of point 9 in the aerodynamic coordinate system defining the cranks and control surface geometry
14	Y9	RS	Y-coordinate of point 9 in the aerodynamic coordinate system defining the cranks and control surface geometry
15	X10	RS	X-coordinate of point 10 in the aerodynamic coordinate system defining the cranks and control surface geometry
16	Y10	RS	Y-coordinate of point 10 in the aerodynamic coordinate system defining the cranks and control surface geometry
17	X11	RS	X-coordinate of point 11 in the aerodynamic coordinate system defining the cranks and control surface geometry
18	Y11	RS	Y-coordinate of point 11 in the aerodynamic coordinate system defining the cranks and control surface geometry
19	X12	RS	X-coordinate of point 12 in the aerodynamic coordinate system defining the cranks and control surface geometry
20	Y12	RS	Y-coordinate of point 12 in the aerodynamic coordinate system defining the cranks and control surface geometry

Record – PAERO4(4801,48,172)

Aerodynamic strip properties.

Word	Name	Type	Description
1	PID	I	Property identification number
2	CLA	I	Select Prandtl-Glauert correction
3	LCLA	I	Identification number of the AEFACT entry that lists the lift curve slope on all strips for each Mach number on the MKAERO entry
4	CIRC	I	Select Theodorsen's function C(k) or the number of exponential coefficients used to approximate C(k)
5	LCIRC	I	Identification number of the AEFACT entry that lists the b, beta values for each Mach number
6	DOC	RS	Distance of the control surface hinge aft of the quarter-chord divided by the strip chord
7	CAOC	RS	Control surface chord divided by the strip chord
8	GAPOC	RS	Control surface gap divided by the strip chord
Words 6 thru 8 repeat until -1 occurs			

Record – PAERO5(5101,51,176)

Aerodynamic panel property.

Word	Name	Type	Description
1	PID	I	Property identification number
2	NALPHA	I	Number of angle of attack values input for each Mach number on the MKAERO1 or MKAERO2 entry

Word	Name	Type	Description
3	LALPHA	I	Identification number of the AEFACT entry that lists the angle of attack values for the strips at each Mach number in the MKAERO1 or MKAERO2 entry
4	NXIS	I	Number of dimensionless chord coordinates to be input
5	LXIS	I	Identification number of the AEFACT entry that lists the dimensionless chord coordinates for the strip in order indicated by values of NXIS and NTHICK
6	NTAUS	I	Number of thickness ratio values to be input
7	LTAUS	I	Identification number of the AEFACT entry that lists the number of thickness ratio values
8	CAOC	RS	Control surface chord divided by the strip chord
Word 8 repeats until -1 occurs			

Record – PANEL(5301,53,378)

Panel definition for coupled fluid-structural analysis.

Word	Name	Type	Description
1	NAME(2)	CHAR4	Panel label
3	SETID	I	Identification number of a SET1 entry that lists the grid points of the panel
Words 1 thru 3 repeat until -1 occurs			

Record – SET1(3502,35,268)

Set definition.

Word	Name	Type	Description
1	SID	I	Unique identification number
2	G	I	Structural grid point identification number
Word 2 repeats until -1 occurs			

Record – SET2(3602,36,269)

Grid point list.

Word	Name	Type	Description
1	SID	I	Unique identification number
2	MACRO	I	Identification number of an aerodynamic macro element
3	SP1	RS	Lower span division point defining the prism containing the set
4	SP2	RS	Higher span division point defining the prism containing the set
5	CH1	RS	Lower chord division point defining the prism containing the set
6	CH2	RS	Higher chord division point defining the prism containing the set
7	ZMAX	RS	Z-coordinate of the top of the prism containing the set
8	ZMIN	RS	Z-coordinate of the bottom of the prism containing the set.

Record – SET3(4302,43,607)

Set definition.

Word	Name	Type	Description
1	SID	I	Set identification number
2	TYPE	I	Set type 1 = grid 2 = element 3 = property
3	ID	I	Identifying number for either structural grids, elements, or physical properties
Word 3 repeats until -1 occurs			

Record – SPLINE1(3302,33,266)

Surface spline methods.

Word	Name	Type	Description
1	EID	I	Element identification number
2	CAERO	I	CAERO entry defining the plane of the spline
3	BOX1	I	First box with motions interpolated using this spline
4	BOX2	I	Last box with motions interpolated using this spline
5	SETG	I	The SETi entry listing the structural grid points to which the spline is attached
6	DZ	RS	Linear attachment flexibility
7	METH(2)	CHAR4	Method for the spline fit: IPS, TPS, or FPS

Word	Name	Type	Description
9	USAGE(2)	CHAR4	Spline usage flag to determine whether this spline applies to force transformation, displacement transformation, or both: FORCE, DISP, or BOTH
11	NELEM	I	Number of elements on x-axis for FPS
12	MELEM	I	Number of elements on y-axis for FPS

Record – SPLINE2(3402,34,267)

Linear spline.

Word	Name	Type	Description
1	EID	I	Element identification number
2	CAERO	I	CAERO entry that is to be interpolated
3	ID1	I	First box or body element with motions interpolated using this spline
4	ID2	I	Last box or body element with motions interpolated using this spline
5	SETG	I	The SETi entry listing the structural grid points to which the spline is attached
6	DZ	RS	Linear attachment flexibility
7	DTOR	RS	Torsional flexibility ratio
8	CID	I	Rectangular coordinate system for which the y-axis defines the axis of the spline
9	DTHX	RS	Rotational attachment flexibility for rotation about the x-axis of the spline
10	DTHY	RS	Rotational attachment flexibility for rotation about the y-axis of the spline

Word	Name	Type	Description
11	USAGE(2)	CHAR4	Spline usage flag to determine whether this spline applies to force transformation, displacement transformation, or both: FORCE, DISP, or BOTH

Record – SPLINE3(4901,49,173)

Aeroelastic constraint equation.

Word	Name	Type	Description
1	EID	I	Element identification number
2	CAERO	I	Identification number of the macro-element on which the element to be interpolated lies
3	BOXID	I	Identification number of the aerodynamic element
4	COMP	I	The component of motion to be interpolated
5	USAGE(2)	CHAR4	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation, or both: FORCE, DISP, or BOTH
7	G	I	Identification number of the independent grid point
8	C	I	Component number in the displacement coordinate system
9	A	RS	Coefficient of the constraint relationship
Words 7 thru 9 repeat until -1 occurs			

Record – SPLINE4(6501,65,308)

Surface spline methods.

Word	Name	Type	Description
1	EID	I	Element identification number
2	CAERO	I	Aero panel identification number defining the interpolation surface
3	AELIST	I	Identification number of an AELIST entry listing boxes with motions that are interpolated using this spline
4	SETG	I	The SETi entry listing the structural grid points to which the spline is attached
5	DZ	RS	Linear attachment flexibility
6	METH(2)	CHAR4	Method for the spline fit: IPS, TPS, or FPS
8	USAGE(2)	CHAR4	Spline usage flag to determine whether this spline applies to force transformation, displacement transformation, or both: FORCE, DISP, or BOTH
10	NELEM	I	Number of elements along the local spline x-axis for FPS
11	MELEM	I	Number of elements along the local spline y-axis for FPS

Record – SPLINE5(6601,66,309)

Linear spline.

Word	Name	Type	Description
1	EID	I	Element identification number
2	CAERO	I	CAERO entry that is to be interpolated

Word	Name	Type	Description
3	AELIST	I	Identification number of an AELIST entry listing aerodynamic boxes with motions that are interpolated using this spline
4	SETG	I	The SETi entry listing the structural grid points to which the spline is attached
5	DZ	RS	Linear attachment flexibility
6	DTORXY	RS	Torsional flexibility ratio in xy-plane
7	CID	I	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending
8	DTHX	RS	Rotational attachment flexibility for rotation about the x-axis of the spline
9	DTHY	RS	Rotational attachment flexibility for rotation about the y-axis of the spline
10	DTHZ	RS	Set to 0.0
11	USAGE(2)	CHAR4	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation, or both: FORCE, DISP, or BOTH
13	METHOD(2)	CHAR4	Set to IPS
15	DTORZY	RS	Set to 1.0

Record – TRIM(2402,24,342)

Trim variable constraint.

Word	Name	Type	Description
1	SID	I	Trim set identification number

Word	Name	Type	Description
2	MACH	RS	Mach number
3	Q	RS	Dynamic pressure
4	AEQR	RS	Flag to request rigid trim analysis
5	LABEL(2)	CHAR4	AESTAT or AESURF entry identifying aerodynamic trim variables
7	UX	RS	Magnitude of aerodynamic extra point dof
Words 5 thru 7 repeat until (-1,-1,-1) occurs			

Record – UXVEC(7201,72,573)

Control parameter state.

Word	Name	Type	Description
1	ID	I	Control vector identification number
2	LABEL(2)	CHAR4	Controller name
4	UX	RS	Magnitude of aerodynamic extra point dof
Words 2 thru 4 repeat until (-1,-1) occurs			

Record – TRAILER

Word	Name	Type	Description
1	UNDEF(6)	None	

Updated modules

ADD5

Note

The changes to ADD5 are documentation only.

Matrix add

To compute [X] = a[A] + b[B] + I [C] + Δ[D] + ϵ[E], where a, b, I , Δ, and ϵ are scalar multipliers.

Updated Parameters:

GAMMA	Input-complex single precision-default = (1.0,0.0). This is I , the scalar multiplier for [C].
DELTAD	Input-complex double precision-default = (1.0D0,0.0D0). This is the scalar multiplier for [D].

BDRYINFO

Generate the geometry and connectivity information for an external superelement definition based on the ASETi and QSETi Bulk Data entries and requested by the EXTSEOUT Case Control command.

Updated Format:

```
BDRYINFO    CASECC,GEOM1,GEOM2,BGPDT,GPDT,USET/GEOM1EX,GEOM2EX,GEOM4EX,CASEX/
            NOMATK/NOMATM/NOMATB/NOMATKY/NOMATP/DMIGSFIX $
```

New Parameters:

NOMATK Input-integer-default = -1. Stiffness matrix existence flag.

-1	Does not exist
0	Empty
1	Exists

NOMATM Input-integer-default = -1. Mass matrix existence flag.

-1	Does not exist
0	Empty
1	Exists

NOMATB Input-integer-default = -1. Viscous damping matrix existence flag.

-1 Does not exist

0 Empty

1 Exists

NOMATKY Input-integer-default = -1. Hysteretic damping matrix existence flag.

-1 Does not exist

0 Empty

1 Exists

NOMATP Input-integer-default = -1. Load matrix existence flag.

-1 Does not exist

0 Empty

1 Exists

BOLTFOR

Calculates bolt forces from preload

Updated Format:

```
BOLTFOR     CASECC,BGPDT,CSTM,GEOM3,ECT,EDT,SIL/
            BTFG,EBOLT,BNFDAT/NSKIP/S,N,LUSET/S,N,NBOLTS $
```

New Output Data Block:

BNFDAT Bolt element grid point force data.

BOLTSF

Calculates bolt strain force

Updated Format:

```
BOLTSF        CASECC,BGPDT,CSTM,GEOM3,KDICT,KELM,EDT,UG,BNFDAT/
              BOLTFOR,BTSFG/NSKIP $
```

New Input Data Blocks:

KDICT	Element stiffness dictionary.
KELM	Element stiffness matrix.
BNFDAT	Bolt element grid point force data.

Note

The ECT, EPT, MPT, and SIL input data blocks have been removed from the module.

New Output Data Block:

BOLTFOR	List of element numbers and bolt force for SDRX.

Revised Parameter:

NSKIP	Input-integer-default=0. Number of cases to skip.

DOM11

Updates geometry and element properties in design optimization

Updated Format:

```
DOM11        EPT,EPTTAB*,PROPO*,XO,DESTAB,CSTM,BGPDT,
             DESGID,COORDO,CON,SHPVEC,GEOM1,GEOM2,MPT,DMATCK,
             DIT,MFRDEP/
             EPTN,COORDN,GEOM1N,GEOM2N,MPTN,DITN/
             DESCYCLE/PROTYP $
```

New Input Data Blocks:

DIT	Direct input tables containing also TABLEDi bulk data for use herein.
MFRDEP	Table for frequency dependent properties that can be designed.

New Output Data Block:

DITN	Direct input tables with modified TABLEDi for updated values of designed frequency dependent properties

Updated Example:

Excerpt from subDMAP PREDOM:

```
DOM11    EPT,EPTTAB,PROPI,XINIT,DESTAB,,,
  DESGID,COORDO,CON,SHPVEC,,,,,,/
  EPTNN,COORDN,JUNKL,,,/0/0 $
```

Initial:

```
DOM11    EPT,EPTTAB,PROPI,XINIT,DESTAB,CSTM,BGPDT,
  DESGID,COORDO,CON,SHPVEC,,,,,,/
  EPTNNX,COORDN,,,,/0 $
```

Final:

```
DOM11    EPT,EPTTAB,PROPO,XO,DESTAB,CSTM,BGPDT,
  DESGID,COORDO,CON,SHPVEC,GEOM1,,,,,/
  EPTN,COORDN,GEOM1N,,,/DESCYCLE $
```

DOM12

Performs soft and hard convergence checks in design optimization. Outputs updated optimization data into the punch file, and prints final optimization results.

Updated Format:

```
DOM12         XINIT,XO,CVAL,PROPI*,PROPO*,OPTPRM,HIS,
              DESTAB,GEOM1N,COORDO,EDOM,MTRAK,EPT,GEOM2,MPT,
              EPTTAB*,DVPTAB*,XVALP,GEOM1P,
              R1TABRG,R1VALRG,RSP2RG,R2VALRG,PCOMPT,OBJTBG,
              ALBULK,AMLIST,DIT/
              HISADD,OPTNEW,DBCOPT,DESNEW/
              DESCYCLE/OBJIN/OBJOUT/S,N,CNVFLG/CVTYP/OPTEXIT/
              DESMAX/MDTRKFLG/DESPCH/DESPCH1/MODETRAK/
              EIGNFREQ/DSAPRT/PROTYP/BADMESH/XYUNIT/FSDCYC $
```

New Input Data Block:

DIT	Direct input tables containing also TABLEDi bulk data for use herein

Updated Example:

1. Excerpt from subDMAP DESOPT following hard convergence:

```
DBVIEW XPREV=XINIT          (WHERE DESITER=DESCYCLP) $
DBVIEW PROPPV=PROPI         (WHERE DESITER=DESCYCLP AND DPTYPE=*) $
DBVIEW HISPV=HIS            (WHERE DESITER=DESCYCLP) $

DOM12 XPREV,XINIT,CVALRG,PROPPV,PROPIF,OPTPRMG,
HISPV,DESTAB,,,EDOM,MTRAK,EPT,GEOM2,MPT,
EPTTABF,DVPTABF,,,,,,,,,,,/
HISADD,NEWPRM,,NEWDES/
DESCYCLE/OBJPV/OBJIN/S,N,CNVFLG/2/OPTEXIT//
MDTRKFLG/DESPCH/DESPCH1/MODETRAK/EIGNFREQ/
DSAPRT/PROTYP $
APPEND HISADD,/HISX/2 $
EQUIVX HISX/HIS/-1 $
DBSTATUS NEWPRM,NEWDES//S,N,NONEWP/S,N,NONEWD $
IF ( NONEWP>0 ) EQUIVX   NEWPRM/OPTPRMG/-1 $
IF ( NONEWD>0 ) EQUIVX   NEWDES/DESTAB/-1 $
```

2. Excerpt from subDMAP DESOPT following soft convergence:

```
DBVIEW PROPIF  =PROPI      WHERE (DPTYPE=*) $
DBVIEW PROPOF=PROPO        (WHERE DPTYPE = *)
DBVIEW EPTTABF =EPTTAB     WHERE (DPTYPE=*) $
DBVIEW DVPTABF =DVPTAB     WHERE (DPTYPE=*) $

DOM12 XINIT,XO,CVALO,PROPIF,PROPOF,OPTPRMG,HIS,DESTAB,
GEOM1N,COORDO,,,EPT,GEOM2,MPT,EPTTABF,DVPTABF,,,,,,,,,,,/
HISADD,NEWPRM,,NEWDES/
DESCYCLE/OBJIN/OBJOUT/S,N,CNVFLG/1/OPTEXIT//
MDTRKFLG/DESPCH/DESPCH1/MODETRAK/EIGNFREQ//PROTYP $
APPEND HISADD,/HISX/2 $
EQUIVX HISX/HIS/-1 $
```

3. Excerpt from subDMAP EXITOPT for termination:

```
IF ( CNVFLG>0 OR DESCYCL1=DESMAX OR OPTEXIT>3 OR
  DSPRINT OR DSUNFORM OR DSEXPORT OR MODETRAK>0 OR
  BADMESH ) DOM12,
   ,XVAL,,,PROPOF,,HIS,DESTAB,GEOM1,COORDO,EDOM,MTRAK,
   EPT,GEOM2,MPT,EPTTABF,DVPTABF,XVALP,GEOM1P,,,,,,,,,/
   ,,DBCOPT,/
   DESCYCL1///CNVFLG/3/OPTEXIT/DESMAX/MDTRKFLG/
   DESPCH/DESPCH1/MODETRAK/EIGNFREQ/DSAPRT/PROTYP/
   BADMESH/XYUNIT $
```

DOPR1

Preprocesses design variables and designed property values

Updated Format:

```
DOPR1         EDOM,EPT,DEQATN,DEQIND,GEOM2,MPT,CASEXX,DIT/
              DESTAB,XZ,DXDXI,DTB,DVPTAB*,EPTTAB*,CONSBL*,
              DPLDXI*,PLIST2*,XINIT,PROPI*,DSCREN,DTOS2J*,
              OPTPRM,CONS1T,DBMLIB,BCON0,BCONXI,DMATCK,DISTAB,
              CASETM,SPAN23,MFRDEP/
              S,N,MODEPT/S,N,MODGEOM2/S,N,MODMPT/DPEPS/
              S,N,PROTYP/S,N,DISVAR $
```

New Input Data Block:

DIT	Direct input tables containing also TABLEDi bulk data for use herein.

New Output Data Block:

MFRDEP	Table for frequency dependent properties that can be designed.

DOPR3

Preprocesses DCONSTR, DRESP1, DRESP2, and DRESP3 Bulk Data entries per analysis type and superelement. Creates tables related to the design objective and a Case Control table for recovering design responses.

Updated Format:

```
DOPR3         CASE,EDOMS,DTB,ECT,EPT,DESTAB,EDT,FRLF,DEQIND,
              DEQATN,DESGID,DVPTAB,VIEWTB,OINT,PELSET,DFRFNC,
              TSPAN23,DIT,EDOM/
              OBJTAB,CONTAB,R1TAB,RESP12,RSP1CT,FRQRSP,CASEDS,
              OINTDS,PELSETDS,DESELM,RESP3,TNSPAN23,SPAN1RG,
              FRLTMP/
              DMRESD/S,N,DESGLB/S,N,DESOBJ/S,N,R1CNT/S,N,R2CNT/
              S,N,CNCNT/SOLAPP/SEID/S,N,EIGNFREQ/PROTYP/DSNOKD/
              SHAPES/S,N,R3CNT/PRESENS3 $
```

Revised Input Data Block:

FRLF input data block was previously TMPFRL.

FRLF	Complex or real eigenvalue summary table, transient response time output list, or frequency response frequency output list. Output by FRLG, TRLG, CEAD, and READ. However, with Version 6.0, for frequency response, this is no longer the same as the previous OL data block, but is now the full data block for the frequency sets data for the current cycle, together with a header record listing the frequency sets.

New Output Data Block:

FRLTMP	A composite FRL data block containing only the relevant records from each FRL data block in multiple subcase, multiple frequency set, frequency response analysis, relevant when the PRESENS3 parameter is TRUE. (See the parameter list.)

New Parameter:

PRESENS3	Logical parameter that is TRUE when one or more of FREQ3, FREQ4, or FREQ5 types of bulk data exist.

DOPR5

Updates design sensitivity tables

Updated Format:

```
DOPR5        XINIT,EPTTAB*,PROPI*,DESTAB,DTOS2K*,DTOS4K,
             TABDEQ,DELBSH,GEOM4,DESGID,DVPTAB*,DBMLIB/
             DTOS2*,DTOS4,DELBSX,DBMLIBU/
             STPSCL/S,N,RGSENS/PROTYP $
```

New Input Data Blocks:

DVPTAB*	Family of tables of attributes of the designed properties by internal property identification number order
DBMLIB	Table of designed beam library data

New Output Data Block:

DBMLIBU	Modified DBMLIB data block updated for current values of designed PBARL/PBEAML dimensions.

DSABO

Incorporates element property design variable perturbations

Incorporates element property design variable perturbations into tables required for stiffness, mass, damping, and load generation.

Updated Format:

```
DSABO         ECT,EPT,EST,DTOS2*,ETT,DIT,MPT,DMATCK,PCOMPT,
              DBMLIBU/
              ESTDVP,MPTX,EPTX,TABEVP,MIDLIS,ESTDVM,PCOMPTX,
              FRQDPV/
              S,N,PROPOPT/DELTAB/PROTYP/PEXIST $
```

New Input Data Block:

DBMLIBU	Modified DBMLIB data block updated for current values of designed PBARL/PBEAML dimensions

New Output Data Block:

FRQDPV	Table of frequencies matching the perturbed elements summary table (ESTDV) when frequency dependent properties are designed

Updated Remark 2:

If central difference is requested, DSABO must be executed for the backward tables. For example, in subDMAP PSLGDV, DSABO is used as follows:

```
DSABO    ECTS,EPTS,EST,DTOS2,ETT,DIT,MPTS,,,/
         ESTDVP,MPTX,EPTX,TABEVP,,,,/
         S,N,PROPOPT/DELTAB $
IF ( CDIFX='YES') THEN $
   DELTABX=-DELTAB $
   DSABO    ECTS,EPTS,EST,DTOS2,ETT,DIT,MPTS,,,/
            ESTDVPB,MPTXB,EPTX,TABEVP,,,,/
            S,N,PROPOPT/DELTABX $
ENDIF $ CDIFX='yes'
```

DSADJ

Creates sensitivity of grid responses

Creates sensitivity of grid responses with respect to design variables based on the combination of adjoint and analysis solution matrices and element sensitivity data. Applicable in frequency response or static analysis only.

Updated Format:

```
DSADJ   XDICTDS,XELMDS,BGPDT,CSTM,XDICTX,XELMX,UGX,ADJG,
        DRDUTB,DSPT1,FRQDPV/
        ADELX/
        NOK4GG/WTMASS/XTYPE/CDIF/COUPMASS/SHAPEOPT $
```

New Input Data Block:

FRQDPV	Table of frequencies matching the perturbed elements summary table (ESTDV) when frequency dependent properties are designed

DSVG1

Creates pseudo loads or scalar terms required in sensitivity analysis

Updated Format:

```
DSVG1       XDICTDS,XELMDS,BGPDT,SIL,CSTM,XDICT,XELM,
            {UGX or AGX},VG,LAMA,DSPT1,FRQDPV,FRQRPR/
            EGX/
            NOK4GG/WTMASS/IAPP/DSVGSF/NOPSLG $
```

New Input Data Blocks:

FRQDPV	Table of frequencies matching the perturbed elements summary table (ESTDV) when frequency dependent properties are designed
FRQRPR	Table containing the number of first level retained (DRESP1) responses per response type and per frequency or time step

EFFMAS

Computes modal effective mass.

Compute the modal effective mass based on the normal modes.

Updated Format:

```
EFFMAS      CASECC,MAA,PHA,LAMA,USET,BGPDT,UNUSED,CSTM,VGQ/
            TEMF,EMM,DMA,MEMF,MPFEM,MEM,MEW,MDLIST/
            SEID/WTMASS/S,N,CARDNO/SETNAM/IUNIT/S,N,IPLOT/EFOPT/S,N,NORBM $
```

New Parameters:

IPLOT	Output-integer-default=0. Set to 1 if PLOT option is requested.
NORBM	Output-integer-default=0. Set to 1 if no rigid body mass matrix was produced.

ELTPRT

Prints element summary information.

Updated Format:

```
ELTPRT      ECT,GPECT,BGPDT,NSMEST,EST,CSTM,MPT,DIT,CASECC,EPT/
            VELEM/
            PROUT/S,N,ERROR/WTMASS $
```

New Input Data Blocks:

NSMEST	Non-structural mass element summary table
EPT	Table of Bulk Data entry images related to element properties

EMG

Computes elemental matrices

Computes elemental matrices for stiffness, differential stiffness, mass, damping, heat conduction, or heat capacity.

Updated Format:

```
EMG         EST,CSTM,MPT,DIT,CASECC,UG,ETT,EDT,DEQATN,DEQIND,
            BGPDT,GPSNT,ECTA,EPTA,EHTA,DITID,EBOLT,COMPEST,EFILL/
            KELM,KDICT,MELM,MDICT,BELM,BDICT/
            S,N,NOKGG/S,N,NOMGG/S,N,NOBGG/S,N,NOK4GG/S,N,NONLHT/COUPMASS/
            TEMPSID/DEFRMSID/PENFAC/IGAPS/LUMPD/LUMPM/MATCPX/KDGEN/TABS/
            SIGMA/K6ROT/LANGLE/NOBKGG/ALTSHAPE/PEXIST/FREQTYP/FREQVAL/
            FREQWA/UNSYMF/S,N,BADMESH/DMGCHK/BOLTFACT/REDMAS/TORSIN/SHLDAMP/
            SHLDMP/BSHDMP/LMSTAT/LMDYN $
```

New Input Data Blocks:

COMPEST	Composite solid element summary table
EFILL	Element fill ratio (TBD)

New Parameters:

LMSTAT	Input-integer-default=6. Power of 10 used to reduce the stiffness in a static analysis.
LMDYN	Input-integer-default=-1. Power of 10 used to reduce the stiffness in a dynamic analysis.

FOCOST

Form contact status

Updated Format:

```
FOCOST      CNELM,ECSTAT,ELAMDA,DLAMDA,ECDISP/ECSTAT2/
            CITO/NOFAC/S,N,NCS0/S,N,NCS1/S,N,NCS2/S,N,NCS3/S,N,NCSC/INREL $
```

New Parameter:

INREL Input-integer. Inertia relief options.
= 0 No inertia relief (default)
= -1 Manual inertia relief
= -2 Automatic inertia relief

GPFDR

Computes grid point forces and element strain energy

Updated Format:

```
GPFDR         CASECC,UG,KELM,KDICT,ECT,EQEXIN,GPECT,PG,QG,
              BGPDT,{LAMA or FOL or TOL or OLF},CSTM,VELEM,PTELEM,
              QMG,NFDICT,FENL,MELM,MDICT,BELM,BDICT,MDLIST/
              ONRGY1,OGPFB1,OEKE1,OEDE1/
              APP/TINY/XFLAG/CYCLIC/WTMASS/S,N,NOSORT2 $
```

New Input Data Block:

MDLIST	Selected mode list from EFFMAS

MATMOD

Matrix modification.

Transforms matrix or table data blocks according to one of many options into output matrix or table data blocks.

Format:

```
MATMOD         I1,I2,I3,I4,I5,I6,I7,I8,I9,I10,I11,I12,I13,I14,I15/
               O1,O2/
               P1/P2/P3/P4/P5/P6/P7/P8/P9/P10/P11/P12/p13/p14/p15/p16/p17/p18/p19 $
```

New Option P1=55

Compute modal damping coefficients, as in the GKAM module.

Format:

```
MATMOD        LAMA,CASECC,DIT,,,,,,,,,,,,/ZETA,/55 $
```

Input Data Block:

LAMA	Normal modes eigenvalue summary table.
CASECC	Table of Case Control command images.

DIT Table of TABLEij Bulk Data entry images.

Output Data Block:

ZETA Diagonal matrix of modal damping coefficients as produced by the GKAM module.

New Option P1=56

Replace the value of, or add a value to, a single term in a matrix.

Format:

```
MATMOD     I1,,,,,,,,,,,,,,/
           O1,/
           56/ICOL/IROW/TYPE/REAL//NCOL/NROW/////////REALD/CMPLX/CMPLXD $
```

Input Data Block:

I1 A matrix. May be purged. See NCOL.

Output Data Block:

O1 Modified I1 matrix.

Parameters:

ICOL Column number.

IROW Row number.

TYPE	Type of value to be replaced or added. 1 Replace current value with REAL. 2 Replace current value with REALD. 3 Replace current value with CMPLX. 4 Replace current value with CMPLXD. -1 Add REAL to current value. -2 Add REALD to current value. -3 Add CMPLX to current value. -4 Add CMPLXD to current value.
REAL	Real single-precision value.
NCOL	Number of columns in O1 if I1 is purged. Default=ICOL.
NROW	Number of rows in O1 if I1 is purged. Default=IROW.
REALD	Real double-precision value.
CMPLX	Complex single-precision value.
CMPLXD	Complex double-precision value.

Example:

In matrix A, add 10.5 to the value of a_{32} (the element of the matrix at column 2 and row 3):

```
MATMOD A,,,,,,,,,,,,,/A1,/56/2/3/—1/10.5 $
```

New Option P1=57

Modify frequency list table TOL.

Format:

```
MATMOD      TOL,,,,,,,,,,,,,,/
            TOL1,/
            57/P2/P3 $
```

Input Data Block:

TOL	Table of frequencies in transient response.

Output Data Block:

TOL1	Truncated table of frequencies containing values ranging between P2 and P3.

Parameters:

P2	Input-integer-default = 0. First frequency table entry to copy from TOL to TOL1.
P3	Input-integer-default = 0. Last frequency table entry to copy from TOL to TOL1.

MODACC

Note

The changes to MODACC are documentation only.

OFREQ, OTIME, and OMODES command processor.

Removes columns in solution and load matrices based on the OTIME, OFREQ, and OMODES case control commands.

Updated Output Data Blocks:

OL1	OL truncated by the OFREQ or OTIME or OMODES command.
U1	U truncated by the OFREQ or OTIME or OMODES command.

Pi1	Pi truncated by the OFREQ or OTIME or OMODES command.

Updated Parameter:

IOPT Input-integer-default=0. Processing options:

0=process OFREQ or OTIME or OMODES.

1=process SETMC for MODCON.

2=process SETMC for PANCON.

Updated Remark:

MODACC selects vectors based on OTIME or OFREQ or OMODES commands in CASECC. If APP = 'CEIG', the selection is based on the imaginary part of the complex eigenvalue. If APP = 'REIG', the selection is based on the frequency ($f = w/2\pi$)

NXNMATLB

Creates an MATLAB .m script file for a superelement.

Creates an MATLAB .m script file for a superelement. The output is based on the MBDEXPORT case control command with the MATLAB describer specified.

Updated Format:

For the STANDARD case control option:

```
NXNMATLB        CASES,LAMA,PHIG,EMVD,U8DOF,MFORC,//
                SEID/WTMASS/FLXERR $
```

For the STATESPACE case control option:

```
NXNMATLB        CASES,AMAT,BMAT,CMAT,EMAT,U7DOF,U8DOF//
                SEID/WTMASS/FLXERR $
```

New Input Data Block:

EMAT	State-space [E] matrix for superelement.

OUTPRT

Constructs sparse load reduction and sparse data recovery partitioning vectors.

Updated Format:

```
OUTPRT      CASECC,ECT,BGPDT,SIL,XYCDB,DYNAMIC,MATPOOL,PG,VGFD,
            TABEVP,TABEVS,SETMC,TEXTSE/
            PVGRID,PVSPC,PVMPC,PVLOAD,PVCODES,TEXTOU/
            S,N,SDRMETH/NOSE/SDROVR/SDRDENS/MCSET/
            MCFLAG/IRTYPE/MATTYP/APP $
```

Updated Output Data Block:

PVGRID   Partitioning vector with ones at rows corresponding to degrees-of-freedom connected to elements or grids specified on the following Case Control commands:

- DISPLACEMENT
- VELOCITY
- ACCELERATION
- FORCE
- STRESS
- STRAIN
- SPCFORCE
- MPCFORCE
- MPRES
- GPFORCE
- ESE
- EKE
- EDE
- GPKE

- MODCON
- PANCON

New Parameter:

APP	Input-character-default=' '. Analysis type.

RBMG3

Computes rigid body information

Computes the rigid body transformation matrix, rigid body error ratio, and strain energy matrix.

Updated Format:

```
RBMG3       LLL,ULL,KLR,KRR/
            DM/CDITER $
```

New Parameter:

CDITER	Input-integer-default=0. Maximum number of iterations in a constrained displacement analysis.

RMAXMIN

Searches result tables during SOL 112.

Searches stress, force and displacement tables during SOLs 101 or 112 for extreme values.

Updated Format:

```
RMAXMIN     OUGV1,OEF1,OES1,PNLIN1,PNLIN2,PNLIN3/
            OUGV1MX,OEF1MX,OES1MX,PNLOUT1,PNLOUT2,PNLOUT3/
            IFABS/IAPPN/IDIAG/RMXTRAN/NPAVG/START/END/ISTART/IEND/RMXPAN  $
```

New Input Data Blocks:

PNLINi	Control information for successive calls when using the panel method and parameter RMXPAN > 0

New Output Data Blocks:

PNLOUTi	Updated control information for successive calls when using the panel method and parameter RMXPAN > 0

New Parameter:

RMXPAN	Input-integer-no default. Determines whether the parameter option is used for SOL 112 runs. = 0 No paneling is performed > 0 RMXPAN time steps are processed

SDR1

Computes solution and single-point forces

Computes and appends the solution (displacements, velocities, acceleration) and single-point forces of constraint at the g-set for each boundary condition. Also appends applied loads.

Updated Format:

```
SDR1         USET,PG,UL,UOO,YS,GOA,GM,PS,KFS,KSS,QR,RGPV,KOA/
             UG,PGT,QG/
             NSKIP/APP/NOQG $
```

New Input Data Block:

KOA	o-set and a-set matrix partition of stiffness matrix

SDR2

Creates output tables.

Creates tables based on output requests for forces of single-point and multipoint forces of constraint, applied loads, displacements, velocities, accelerations, element stresses, element strains, and element forces. These output tables are suitable for printing, plotting, and various other postprocessing.

Updated Format:

```
SDR2        CASECC,CSTM,MPT,DIT,EQEXIN,SILD,
            ETT,{OL or EDT},BGPDT,PG,QG,UG,EST,XYCDB,
            OINT,PELSET,VIEWTB,GPSNT,DEQATN,DEQIND,DITID,
            PCOMPT,GPKE,BOLTFOR,MDLIST,COMPEST/
            OPG1,OQG1,OUG1,OES1,OEF1,PUG,OGPKE1,OEFIIP,OEFIIS,OESRIP,OESRIS/
            APP/S,N,NOSORT2/NOCOMP/ACOUSTIC/METRIK/
            ISOFLG/GPF/ACOUT/PREFDB/TABS/
            SIGMA/ADPTINDX/ADPTEXIT/BSKIP/FREQW/
            BTBRS/LANGLE/OMID/SRCOMPS $
```

New Input Data Block:

COMPEST	Composite solid element summary table.

New Output Data Blocks:

OEFIIP	Data block for in-plane ply failure indices.
OEFIIS	Data block for inter-laminar shear failure indices.
OESRIP	Data block for in-plane ply strength ratios.
OESRIS	Data block for inter-laminar shear strength ratios.

New Parameter:

SRCOMPS	Input-character-default='NO'. Requests output of failure indices for composite solid elements. If SRCOMPS='YES", requests output of strength ratios for composite solid elements.

SSG1

Computes static load matrix.

Computes the static load matrix based on static loads, thermal loads, and enforced deformation loads or heat transfer loads. Also generates the acceleration matrix due to inertial loads for design sensitivity analysis.

Updated Format:

```
SSG1      SLT,BGPDT,CSTM,MEDGE,EST,MPT,ETT,EDT,MGG,CASECC,
          DIT,UG,DEQATN,DEQIND,GPSNT,CSTMO,SCSTM,GEOM4,
          EPT,PCOMPT,COMPEST/
          {PG or AG},PTELEM,SLTH/
          LUSET/NSKIP/DSENS/APP/ALTSHAPE/TABS/SEID/LMFACT $
```

New Input Data Block:

COMPEST	Composite solid element summary table

New Parameter:

LMFACT	Scale factor for the stiffness matrix of Lagrange rigid elements.

Note

The LMFACT parameter was introduced in NX Nastran 5, but was previously undocumented.

SSG2

Reduces static load and enforced displacement matrices

Updated Format:

```
SSG2      USET,GM,YS,KFS,GOA,DM,PG,KOA/
          QR,PO,PS,PA,PL $
```

New Input Data Block:

KOA	o-set and a-set matrix partition of stiffness matrix

TA1

Combines element data into tables.

Combines all of the element data (geometry, connection, and properties) into a table(s) convenient for generation of the element matrices (stiffness, mass, and so on) and output quantities (stress, force, and so on).

Updated Format:

TA1	MPT,ECT,EPT,BGPDT,SIL,ETT,CSTM,DIT,ECTA,EHT/ EST,ESTNL,GEI,GPECT,ESTL,VGFD,DITID,NFDICT,COMPEST,NSMEST/ LUSET/S,N,NOESTL/S,N,NOSIMP/NOSUP/S,N,NOGENL/SEID/ LGDISP/NLAYERS/S,N,FREQDEP/BSHDAMP/S,N,BSHDMP/NSMID $

New Output Data Blocks:

COMPEST	Composite solid element summary table
NSMEST	Non-structural mass element summary table

VDRMC

Creates modal contribution tables based on modal contribution output requests.

Updated Format:

VDRMC	CASEG,SETMC,AMC,NMC,MAG,APC,NPC,PNLLST,OL,MFRQ,ECT,BGPDT,TEXTSE/ OUTFLE/ APP/S,N,NOSORT2/S,N,NOSOUT/FMODE/IRTYPE/FSFLAG $

New Input Data Blocks:

APC	Absolute panel contributions results matrix.
NPC	Normalized panel contributions results matrix.
PNLLST	Panels list table.

New Remark:

APC and PNLLST are only valid for acoustic analysis in SOL 111. They are optional and may be purged.

VDRPC

Creates panel or grid contribution tables based on panel/grid contribution output requests.

Updated Parameter:

DATTYP Input-integer-no default. Data type.
1 = panel contributions
2 = grid contributions
3 = reciprocal panel contributions

XYTRAN

Creates table of plot instructions for x-y plots

Updated Format for SDR2 Outputs:

```
XYTRAN        XYCDB,OPG2,OQG2,OUG2,OES2,OEF2,OSTR2,OQMG2/
              XYPLOT/
              APP/XYSET/S,N,PLTNUM/S,N,CARDNO/S,N,NOXYPLOT/
              S,N,TABID/DFLAG/VFLAG/AFLAG $
```

New Parameters:

DFLAG	Input-integer-default=0. Displacement output flag.	
	0	Output displacement
	1	Do not output displacement
VFLAG	Input-integer-default=0. Velocity output flag.	
	0	Output velocity
	1	Do not output velocity
AFLAG	Input-integer-default=0. Acceleration output flag.	
	0	Output acceleration
	1	Do not output acceleration

New modules

DRMH2

Creates partitioning vector to properly align external superelement OTMs with system solution.

Format:

```
DRMH2        BGPDTS,USET/PARTV/S,N,NOPARTV $
```

Input Data Blocks:

BGPDTS	Basic grid point definitions table for external superelement.
USET	DOF set definition table for external superelements.

Output Data Blocks:

PARTV	Partitioning vector.

Parameters:

NOPARTV	Input-integer-no default. Partitioning vector existence flag. -1 = PARTV not created; external superelement OTMs are already correct 0 = PARTV created; external superelement OTMs need to be realigned

Remarks:

1. All data blocks are required.

Example:

Partition displacements OTM.

```
DRMH2    BGPDTS,USET/PARTV/S,N,NOPARTV $
IF (NOPARTV>-1) THEN $
   FILE      MUG1X=APPEND $
   PARTN     MUG1,PARTV,/MUG1G,,MUG1S,/1 $
   APPEND    MUG1G,MUG1S/MUG1X $
ELSE $
```

```
      EQUIVX    MUG1/MUG1X/ALWAYS $
   ENDIF $
```

In the example, MUG1X is then used in the call to DRMH3.

FOCOASET

ASET processing of contact DOF.

Format:

```
FOCOASET      CASECC,GEOM4,CNELM/GEOM4N/S,N,ICTASET $
```

Input Data Blocks:

CASECC	Table of Case Control command images
GEOM4	Table of Bulk Data entry images related to constraints, degree-of-freedom membership, and rigid element connectivity
CNELM	Contact element definition table

Output Data Blocks:

GEOM4N	Updated GEOM4 data block with the a-set data

Parameters:

ICTASET Output-integer-no default. Contact ASET status parameter:

1	Not all criteria for ASET processing with contact have been met
-1	All criteria for ASET processing with contact have been met

GLFONOTR

Glue normal and in-plane tractions.

Format:

```
GLFONOTR     GNELM,UGLCURB,ELGNST,CASECC/
             OQGGF1,OBG1/NROW/NVEC $
```

Input Data Blocks:

GNELM	Glue element definition table
UGLCURB	Global displacements in basic coordinate system
ELGNST	Table of glue element stiffness
CASECC	Table of Case Control command images

Output Data Blocks:

OQGGF1	Glue forces at grid point in basic coordinate system
OBG1	Glue normal and tangential tractions at grid point in basic coordinate system

Parameters:

NROW	Input-integer-no default. Number of rows in the glue force vector.
NVEC	Input-integer-no default. Number of solution results to process.

GP1EX

Modify the GEOM3 data block to convert external superelement loads via SELOAD bulk entries into FORCE and MOMENT bulk entries.

Format:

```
GP1EX     GEOM1,GEOM3,BGPDTS,EQEXINS,SELOAD,PG/
          GEOM3XSE/SEID/S,N,UPDATE $
```

Input Data Blocks:

GEOM1	Table of bulk entry images related to geometry for the entire model.
GEOM3	Table of bulk entry images related to static and thermal loads for the entire model.
BGPDTS	Basic grid point definition table for the external superelement.
EQEXINS	Table to equivalence external and internal grid/scalar numbers for the external superelement.
SELOAD	DTI table for SELOAD bulk entry images.
PG	External superelement load matrix in the g-set.

Output Data Blocks:

GEOM3XSE	Table of bulk data entry images related to static and thermal loads for the entire model that now contains FORCE and MOMENT bulk images to represent the external superelement applied loads.

Parameters:

SEID	Input-integer-no default. Superelement ID of the external superelement.
UPDATE	Output-integer-no default. GEOM3 update flag. 0 = GEOM3 not updated and GEOM3XSE not created 1 = GEOM3 updated and GEOM3XSE created

GPFTOTL

Recalculates the totals for grid point force output.

Format:

```
GPFTOTL       OGPFIN/OGPFOU $
```

Input Data Blocks:

OGPFIN	Grid point force balance data block (SORT1 format) for which grid point force balance totals are to be recalculated.

Output Data Blocks:

OGPFOU	Resulting grid point force balance data block (SORT1 format).

Parameters:

None

GPFXPND

Expands a grid point force output data block based on a template.

Format:

```
GPFXPND       OGPFTMP,OGPFIN/OGPFOU $
```

Input Data Blocks:

OGPFTMP	Grid point force balance data block to be used as a template (SORT1 format).
OGPFIN	Grid point force balance data block (SORT1 format) to be expanded.

Output Data Blocks:

OGPFOU	Resulting grid point force balance data block (SORT1 format).

Parameters:

None

XSELOADS

Generates DTI,SELOAD entries for an external superelement.

Format:

```
XSELOADS        CASECC,SELOAD,GEOM3,DYNAMIC,PA/XSELOADS $
```

Input Data Blocks:

CASECC	Case control data block.
SELOAD	Table of user-supplied DTI,SELOAD entries.
GEOM3	Table of applied load card images.
DYNAMIC	Table of dynamic data card images.
PA	Matrix of applied loads.

Output Data Block:

XSELOAD	Table of DTI,SELOAD definitions of the external superelement consistent with the applied loads.

Parameters:

None

Chapter

13 NX Nastran 8 Problem Report (PR) fixes

Problems identified in previous releases that have been fixed in NX Nastran 8 are listed below.

PR#	Problem Reported	Problem Originated	Problem Description
1710250	V6.1	V6.0	Models fail to solve when a rigid element is connected to the mid-edge of higher-order shell elements and the Lagrange option is selected.
1747955	V6.1	V6.1	Slow performance occurs for models containing glue.
1770754			Same as PR#1710250.
1771520			Same as PR#1781406.
1771717	V7.0	V1.0	A greater than expected difference is observed in the results of a SOL 108 analysis when compared to the results of a SOL 111 analysis.
1781406	V7.0	V6.0	A model containing RBE3 elements produces different results depending on whether the Lagrange option is used or not.
1802897	V7.0	V6.0	SOL 601,106 produces incorrect results when the model contains RBE3 elements and CONM2 elements.
1805750	V7.1	V2.0	A fatal error occurs when using the element iterative solver on a model containing rigid elements that has different boundary conditions in each subcase.
1807693	V7.1	V1.0	System cell 463 does not override the fatal error that arises when CBEAM elements with offset vectors are used in a SOL 106 large displacement analysis.
1808103			Same as PR#6412237.
1813802	V7.0	V7.0	A segmentation violation error occurs when computing deflections for CFAST elements in a SOL 111 analysis.

1816262	V7.1	V7.0	The contact pairing code does not always locate and selected the closest face, which can lead to unexpected contact results.
1817722	V7.1	V7.0	Specific models that contain surface-to-surface glue produce incorrect rigid body modes 4, 5 and 6.
1817838	V7.1	V7.0	When running a large model having more than ten subcases, incorrect imaginary part stress results are produced by SOL 108. The problem only occurs on Linux platforms.
1821507	V7.1	V7.0	In SOL 601, when GPFORCE output is requested with the Total Load Application (TLA) method, the run may terminate or not output the GPFORCE results. TLA is an NXSTRAT parameter with AUTO = 3.
1822144	V7.1	V4.0	For models with linear contact and multiple subcases, a fatal error is produced if the second or later subcase does not contain a load definition.
1824002			Same as PR#1805750.
1827062	V7.1	V7.0	When using NX Nastran 7.1, SYSTEM FATAL MESSAGE 4276 is issued for a model that contains CWELD elements. The same model runs without error when NX Nastran 7 is used.
1830290	V7.0	V1.0	An incorrect projection failure sometimes occurs when internally generating CWELD element construction points with the ELPAT option. A fatal error occurs for versions prior to NX Nastran 7.1. Using NX Nastran 7.1, the same job runs, but an unneccessary diameter reduction is performed.
1844934	V7.1	V7.0	When running a model with 8-digit grid ID numbers on 32-bit platforms, processing of the PANCON case control command fails.
1845762	V7.1	V7.1	A fatal error occurs when there is a projection failure generating CWELD element construction points. The element number for the failing element is incorrect.
2147657	V7.0	V7.0	The yield condition for the Mohr-Coulomb and the Drucker-Prager yield criteria in a SOL 106 plasticity analysis are incorrect.

2148477	V7.0	V6.0	ADAMS MNF file creation fails when modal damping and the HOU option for the EIGR bulk entry are specified.
2156140	V7.0	V5.1	A fatal error occurs when requesting more than 16GB of memory for a DMP static solution using the old GDSTAT partitioning method (GPART = 0 on the command line).
2160633	V6.1	V2.0	The results from a second load case are missing in the element iterative solver when each subcase has different specified displacements.
2161330	V7.1	V5.1	When the shell offset feature is used for shell elements like CQUAD4 and CQUAD8, valid results are only produced if all of the shell elements that have contact defined have the same offset value or if the ORDER of the shell elements that have contact defined does not change when the shell faces are sorted.
2166074	V4.1	V4.0	Coupled heat transfer and linear static analysis runs fail when contact is present.
2163394	V7.1	V4.0	A DMAP error causes a matrix partitioning operation to error out during sparse data recovery operations for modal transient enforced motion analysis (SOL 112 with SPCD using the default constraint mode method).
6195473	V7.0	V6.1	If all contact elements become inactive, a generic error message is output rather than a descriptive one that informs the user that all contact elements are inactive and NX Nastran will attempt to perform the solution without contact.
6298637			Same as PR#1710250.
6343086	V7.0	V5.0	When a load is specified on the independent grid of a rigid element and only translational DOF exist, the element iterative solver produces incorrect displacements for the independent grid and correct displacements for the other grids.
6355711	V7.0	V4.0	Erroneous external superelement data is created when the SPOINT IDs are less than the BSET grid IDs.

6359198	V7.1	V7.0	When a 2D contact surface is a closed loop defined on higher-order axisymmetric, plane stress, or plane strain elements, the contact traction results may be incorrect.
6366793	V7.1	V7.0	While performing an analysis on two parts, one with design body and the other with simplified body, results are not produced, but an error message is not issued and the error log does not appear when you try to access it.
6369808	V7.0	V5.0	Different MPC forces are produced when the Lagrange option is used as compared to when it is not used.
6382382	V7.0	V7.0	An incorrect singularity arises for the reference grid of RBE3 elements when the Lagrange option is selected and the analysis is performed on the Windows 64 bit platform.
6383070	V7.1	V7.0	SYSTEM FATAL MESSAGE 4276 (QOPEN) and ERROR CODE 3012 PID= 0 arise when SPC forces are requested, multiple subcases exist, the iterative solver is used, and contact is present.
6385222	V7.1	V7.0	Incorrect results are produced when the load set identification number of an ACSRCE bulk entry is greater than the set identification number of an RLOAD bulk entry.
6387264	V7.0	V7.0	In SOL 106 geometric nonlinear analysis only, the thermal strain component is not included in the linear strain output for CBEAM elements.
6390969	V7.1	V3.0	When using SOL 601, a GPFORCE output request causes severe performance degradation.
6392352	V7.1	V6.0	In SOL 601 with Tied Contact (a BCTPARA parameter, TIED = 1), only the 1st pair in BCTSET is recognized when MAXD1 is specified.
6393035	V7.1	V1.0	ECHO = UNSORT,PUNCH causes the wrong continuation characters to be written.

6393175	V7.0	V6.1	Eigenvalues are unnecessarily recomputed during a SOL 111 restart run of a SOL 103 analysis when non-structural mass is present.
6398742	V7.1	V1.0	When there are multiple random loadings, all the results are not provided and a low memory error message is not issued.
6398969	V7.1	V7.0	When using the ACSRCE bulk entry, the DPHASE bulk entry is ignored when the DAREA bulk entry references the lowest grid ID in model.
6410044	V7.1	V6.0	A fatal error occurs when there are more than 43 subcases with DRSPAN case control commands during a SOL 200 analysis.
6412237	V7.0	V7.0	A SOL 601 run may terminate unexpectedly when using LDC and contact.
6414861	V7.1	V1.0	In a SOL 106 run, the solution is considered converged by NX Nastran when user information message 6189.1 (BEST ATTAINABLE SOLUTION HAS BEEN FOUND) is issued. However, NX Nastran is inconsistent in how it handles the outputs, depending on whether or not intermediate output is requested. Furthermore, when no intermediate output is requested, there is an error in what is output.
6415843	V7.1	V7.0	For RDMODES, SOL 103 fails when running parallel and NRC > 128.
6419850	V7.1	V7.0	Errors in the mass occur when connector mass is specified on CFAST elements.
6424012	V7.1	V4.0	GRID bulk entries referenced by a CORD1x bulk entry are not written to the punch file.
6426592	V7.1	V7.0	An MPC bulk entry with only one grid per line and continuation lines is allowable for some platforms, but causes a fatal error on little endian ILP64 executables.
6427286	V7.1	V4.0	The external superelement DOF order is not interpreted correctly.
6429413	V7.1	V4.1	For SOL 153, glue element conductance is incorrectly calculated using a constant value for thermal conductivity even though a table that defines the temperature dependency of the thermal conductivity is supplied.

6430224	V7.1	V1.0	Auto correlation random output produces a fatal error when output for a nonexisting grid is requested.
6443407	V7.0	V1.0	DMIG loads are only applied to the first subcase.
6447021	V7.1	V4.0	A fatal error occurs when contact or glue are defined on solid elements that reference PLSOLID bulk entries.
6450805	V7.1	V7.0	MEFFMASS(PRINT,ALL,THRESH = 0.7) = YES does not work.
6451068	V6.1	V6.0	Composite strength ratios are not written to the .op2 file when running a SOL 106 analysis.
6451202	V7.0	V1.0	DEFORM bulk entries are only valid with SOL 101, 105, 114 and SOL 200 (statics only). When DEFORM entries are used in an unsupported solution sequence, a fatal error is not generated and the run is allowed to proceed and produce incorrect results.
6451967	V7.1	V7.0	For a rotor dynamic analysis using the default mode tracking method 2, the .csv file containing the Campbell diagram information is incorrectly generated. Additionally, the format for the mode tracking method 2 is not documented.
6452785	V7.1	V1.0	In SOL 106, the CTE for MID2 and MID4 is not computed correctly.
6461216	V7.1	V5.0	If glue is present in a contact analysis that only contains gap elements, a fatal error is produced.
6463361	V7.1	V7.0	A model containing a CWELD element defined using the GRIDID format and a CWELD element defined using the ELPAT format runs fine when using either NX Nastran 7 or NX Nastran 7.1. However, when a CWELD element defined using the ALIGN format is added to the input file, the run terminates when using NX Nastran 7.1. The model runs fine when NX Nastran 7 is used.

6466203	V7.1	V6.0	The grid point force balance OTM generated by the EXTSEOUT case control command for a model with very large grid point and element ID numbers causes a string of eight asterisks to be written to the DTI data in the punch file (DMIGPCH option).
6469944	V7.1	V6.0	A fatal error occurs when the loading contains both DAREA and PLOAD bulk entries.
6471311	V7.0	V6.0	The existence of PLOTEL elements in a run containing the EXTSEOUT case control command causes the OTM for grid point force balance to be created incorrectly.
6472651			Same as PR#6473807.
6473807	V7.1	V7.0	NX Nastran 7.1 crashes when processing a large number of BLSEG bulk entries in an edge-surface glue.
6474120	V7.1	V7.0	AUTOSPC incorrectly restrains dependent DOF of RBE3 elements when the Lagrange method is used as the rigid element processing method.
6475072	V7.1	V7.0	The utility nastran executable could not run (a) estimate executable, (b) nastran architecture dependent binary and analysis executable with Linux kernels 2.6.23 and higher.
6475884	V7.1	V7.1	Normal modes must be mass normalized for rotor dynamics when using SOL 110.
6482678	V7.1	V1.0	AUTOSPC is not supported for SOL 129, LGDISP=1. However, the code does not trap the use of AUTOSPC with SOL 129, LGDISP=1, as the QRG documentation indicates.
6482783	V7.1	V1.0	A matrix math failure occurs when generating residual vectors for a fluid/structure interaction analysis (e.g. acoustics).
6483070	V7.1	V7.0	When using the THRU option on the SET case control command and there is only one valid entry in the range, output will not be produced for this entry.
6483571	V7.1	V4.0	A fatal error occurs when the SPCD method is used with acoustic/structure coupling.

6493690	V7.1	V4.1	A fatal error occurs in conmods when running an acoustic model with enforced acceleration in SOL 111.
6499510	V6.1	V6.1	User fatal message 316 occurs if SHLTHK = 0 and ZOFFSET = 0 on a BCTPARM bulk entry.
6490190	V7.1	V1.0	Negative values for element strain energy are output when running a linear static analysis with thermal strains.
6512941	V7.1	V1.0	Attempts to use the AVGAMP keyword of the EDE/EKE/ESE case control command fails.
6513118	V7.1	V6.0	The TGPF table from an EXTSEOUT case control command does not output during a restart run.
6513125	V7.1	V7.0	The MGPF OTM from the EXTSEOUT case control command is incorrect when CONM2 or NSM bulk entries are used.
6524684	V7.1	V7.1	Calculating panel contributions using the PANCON case control command in a SOL 108 analysis generated incorrect results.

Chapter

14 System description summary

The list of supported systems is included in the **README.txt** file located in the NX Nastran installation under the *nxn8* directory.