NX Nastran 7.1 Release Guide

Proprietary & Restricted Rights Notice

TAUCS Copyright and License

Availability

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

Multi-body dynamics and control system software interfaces

The capability to export a state-space (first-order) representation of a dynamic system was added with NX Nastran 7. The state-space representation is generated from an NX Nastran normal modes solution and written to an OUTPUT4 file. Outside of NX Nastran, you can convert the OUTPUT4 file to a format readable by third-party control system software.

Beginning with NX Nastran 7.1, the capability to export dynamic system representations has been enhanced. Now you can also:

- Export a standard (second-order) representation of a dynamic system to an OUTPUT4 file for use with control system software.

- Export a standard or state-space representation of a dynamic system to a MATLAB script file.

You can access the new capabilities using the MBDEXPORT case control command. To request a standard representation written to an OUTPUT4 file, specify the following describers:

OP4=unit

STANDARD

FLEXBODY=YES

A new describer, MATLAB, has been created to request a MATLAB script file. To request a standard representation written to a MATLAB script file, specify the following describers:

MATLAB

STANDARD

FLEXBODY=YES

To obtain a state-space representation written to a MATLAB script file, specify the following describers:

MATLAB

STATESPACE

FLEXBODY=YES

For a mathematical description of the standard and state-space representations of a dynamic system, see “Multi-body Dynamics and Control System Interfaces” in the *Advanced Dynamic Analysis User’s Guide*.

See the updated MBDEXPORT case control command.

MBDEXPORT Multi-Body Dynamics Export

Generates interface file for third-party multi-body dynamics and control system software during a normal mode solution (SOL 103).

Format:

```
MBDEXPORT [ RECURDYN | ADAMS | OP4 = unit | MATLAB ] , [ STANDARD | STATESPACE ] , [ FLEXBODY = { NO | YES } ] , [ FLEXONLY = { YES | NO } ] ,

[ MINVAR = { PARTIAL | CONSTANT | FULL | NONE } ] , [ PSETID = { NONE | setid | ALL | sktunit } ] ,

[ OUTGSTRS = { NO | YES } ] , [ OUTGSTRN = { NO | YES } ] ,

[ RECVROP2 = { NO | YES } ] , [ CHECK = { NO | YES } ]
```

The general examples, describers, and remarks are an overview for all interface types. Below this are specific examples, describers, and remarks sections for each interface type.

General Examples:

```
MBDEXPORT ADAMS STANDARD FLEXBODY=YES FLEXONLY=NO
MBDEXPORT FLEXBODY=YES MINVAR=FULL
MBDEXPORT OP4=22 STANDARD FLEXBODY=YES
MBDEXPORT OP4=22 STATESPACE FLEXBODY=YES
MBDEXPORT MATLAB STANDARD FLEXBODY=YES
MBDEXPORT MATLAB STATESPACE FLEXBODY=YES
```

General Describers:

Describer	Meaning
ADAMS	Generate ADAMS Interface Modal Neutral File (MNF).
RECURDYN	Generate RecurDyn Flex Input (RFI) file. (default)

Describer	Meaning
OP4	Generate OP4 file.
MATLAB	Generate MATLAB script file.
STANDARD	Matrices are based on standard second order differential equations of motion. (default)
STATESPACE	Matrices are based on first order differential equations that represent the equations of motion, and are suitable for use with control system software.

General Remarks:

1. Only one choice of ADAMS, RECURDYN, OP4, or MATLAB is allowed and must immediately follow the MBDEXPORT command.
2. The describers can be truncated to the first 4 characters.
3. STATESPACE is not valid for ADAMS or RECURDYN.

The information from this point on is specific to each interface type.

RECURDYN STANDARD Describers:

Describer	Meaning
FLEXBODY	Requests the generation of RFI (required).
NO	Standard NX Nastran solution without RFI creation. (default)
YES	RFI generation requested.
FLEXONLY	Determines if DMAP solution and data recovery runs or not after RFI creation is complete.
YES	Only RFI creation occurs. (default)
NO	RFI file creation occurs along with standard DMAP solution and data recovery.
MINVAR	Determines how mass invariants are computed.

Describer	Meaning
PARTIAL	Mass invariants 6 and 8 are not computed. (default)
CONSTANT	Mass invariants 1,2,3 and 9 are computed.
FULL	All nine mass invariants are computed.
NONE	No mass invariants are computed.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) (including PLOTEL) whose connectivity is exported into the RFI. (See Remark 16)
NONE	No specific sets are selected, thus all grids, geometry and associated modal data are written to RFI. (default)
setid	The connectivity of a specific element set is used to export face geometry.
ALL	The connectivity of all element sets are used to export face geometry.
OUTGSTRS	Determines if grid point stress is written to RFI.
NO	Do not write grid point stress to RFI. (default)
YES	Write grid point stress to RFI.
OUTGSTRN	Determines if grid point strain is written to RFI.
NO	Do not write grid point strain to RFI. (default)
YES	Write grid point strain to RFI.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of RecurDyn/Flex results.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.

Describer		Meaning
CHECK		Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 20)
	NO	No debug output will be written. (default)
	YES	Debug output will be written.

RECURDYN STANDARD Remarks:

1. The creation of the RecurDyn Flex Input file is applicable in a non-restart SOL 103 analysis only. RFI files are named 'jid_seid.rfi', where seid is the integer number of the superelement (0 for residual-only run). These files are located in the same directory as the jid.f06 file.

2. The creation of the RecurDyn Flex Input file is initiated by MBDEXPORT RECURDYN FLEXBODY=YES (other describers are optional) and the inclusion of the bulk data entry DTI,UNITS.

3. The Data Table Input Bulk Data entry DTI,UNITS, which is required for an MBDEXPORT RECURDYN FLEXBODY=YES run, is used to specify the system of units for the data stored in the RFI (unlike NX Nastran, RecurDyn is not a unitless code). Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

   All entries are required. Acceptable character strings are listed below.

   Mass:

   KG - kilogram

   LBM – pound-mass (0.45359237 kg)

   SLUG – slug (14.5939029372 kg)

   GRAM – gram (1E-3 kg)

   OZM – ounce-mass (0.02834952 kg)

   KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

   MGG – megagram (1E3 kg)

   MG – milligram (1E-6 kg)

MCG – microgram (1E-9 kg)

NG – nanogram (1E-12 kg)

UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

CN – centinewton (1E–2 N)

P – poundal (0.138254954 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

4. Because DTI,UNITS determines all units for the RFI, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the RFI. For example, if the model mass is in kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1 ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the RFI would be: “DTI,UNITS,1,KG,N,M,S”.

5. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points; or for a residual-only type model, you can use NX Nastran ASET Bulk Data entries to define the attachment points.

6. The eight mass variants are:

$$\underset{1x1}{{}^{1}I} = \sum_{p=1}^{N} m_p$$

$$\underset{3x1}{{}^{2}I} = \sum_{p=1}^{N} m_p s_p$$

$$\underset{3x3}{{}^{3}I} = \sum_{p=1}^{N} (m_p \tilde{s}_p \tilde{s}_p - I_p)$$

$$\underset{3xM}{{}^{4}I_j} = \sum_{p=1}^{N} m_p \Phi_p \quad j = 1,\ldots,M$$

$$\underset{3x3}{{}^{5}I_j} = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj} \qquad j = 1,\ldots,M$$

$$\underset{3x3}{{}^{6}I_{jk}} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \phi_{pk} \qquad j,k = 1,\ldots,M$$

$$\underset{3xM}{{}^{7}I} = \sum_{p=1}^{N} m_p \tilde{s}_p \Phi_p + \sum_{p=1}^{N} I_p \Phi_p^*$$

$$\underset{3xM}{{}^{8}I_j} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \qquad j = 1,\ldots,M$$

$s_p = [xyz]^T$ are the coordinates of grid point p in the basic coordinate system.

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix}$$ = skew symmetric vector cross product operator.

f $_p$ = partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid p.

I_p = inertia tensor p.

f $_p{}^*$ = partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid p.

$\tilde{\phi}_{pf}$ = skew-symmetric matrix formed for each grid translational degree of freedom for each mode.

M = number of modes.

N = number of grids.

7. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTS (ns) should be at least ns=n+(6+p), assuming that residual flexibility is on. In the above equation for ns, the number of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL Bulk Data entries; the number of load cases is p. In general, you cannot have too many SPOINTs, as excess ones will be truncated with no performance penalty.

8. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in a more accurate representation of the component shapes with little additional computational effort.

9. OMIT or OMIT1 Bulk Data entries are not supported.

10. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.

11. P-elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

12. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an RFI. It must have consistent units with regard to the DTI,UNITS Bulk Data entry. Before generating the RFI, NX Nastran will appropriately scale the WTMASS from the physical mass matrix and mode shapes.

13. There is a distinction between how an MBDEXPORT RECURDYN FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

14. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran, the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with MBDEXPORT RECURDYN FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This caution only applies to a superelement run and not to a residual-only or a part superelement run.

15. OUTGSTRS and OUTGSTRN entries require the use of standard NX Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN= commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain print to the NX Nastran .f06 file.

16. To reduce the FE mesh detail for dynamic simulations, PSETID can include the ID of a SET entry. The SET entry lists PLOTEL or element IDs, whose connectivity is exported into the RFI to display the components in RecurDyn. This option can significantly reduce the size of the RFI without compromising accuracy in the FunctionBay simulation providing that the mass invariant computation is requested. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids are attached only to a-set or exterior grids), a SEELT Bulk Data entry must be specified to keep that display element with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

    If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements that are used to select grids to display the component in RecurDyn. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID then represents a list of OUTPUT(PLOT) defined elements' sets. The union of which will be used to define a set of PLOTELs or other elements used to select grids to display the component in RecurDyn. If you wish to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

    The following element types are not supported for writing to an RFI, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CFLUIDi, CMASS3, CMASS4, CRAC2D, CRAC3D, CTRMEM, CTWIST, CWEDGE, CWELD, and GENEL.

17. Typical NX Nastran data entry requirements are described below.

    Typical Parameters:

    - PARAM,RESVEC,character_value – controls calculation of residual vector modes.

    - PARAM,GRDPNT,value - mass invariants ${}^1I$, ${}^2I$, and ${}^3I$ will be computed using results of NX Nastran grid point weight generator execution in the basic coordinate system.

    Typical Case Control:

    - MBDEXPORT RECURDYN FLEXBODY=YES is required for RFI generation.

    - METHOD=n is required before or in the first subcase for modal solutions.

    - SUPORT1=seid is necessary to select a static support set for a residual only linear preload run.

- SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.

- OUTPUT(PLOT) is necessary to define elements used to select grids to display the component in RecurDyn when PSETID=ALL or setid.

  SET n=list of elements (including PLOTELs) is used to select grids to display the component.

- OUTPUT(POST) is necessary to define volume and surface for grid stress or strain shapes.

  SET n=list is a list of elements for surface definition for grid stress or strain shapes.

  Stress and strain data in the RFI is limited to the six components (that is, 3 normal and 3 shear) for a grid point for a given mode.

  SURFACE n SET n NORMAL z3 is used to define a surface for writing stress and strain data. Only one FIBER selection is allowed for each SURFACE, thus the use of the FIBER ALL keyword on the SURFACE case control command will write stresses to the RFI at the Z1 fiber location only.

  Since the FIBER keyword only applies to stresses, strain data will always be written to the RFI at the MID location.

  Stress and strain data at grid points can only be written to the RFI for surface and volume type elements (for example, CQUAD and CHEXA).

  VOLUME n SET n is a volume definition.

  The default SYSTEM BASIC is required with SURFACE or VOLUME.

- STRESS(PLOT) is necessary for stress shapes.

- STRAIN(PLOT) is necessary for strain shapes.

- GPSTRESS(PLOT) is necessary for grid point stress shapes to be included in the RFI.

- GPSTRAIN(PLOT) is necessary for grid point strain shapes to be included in the RFI.

  Typical Bulk Data:

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for RFI generation. For input files containing superelements, this command must reside in the main bulk data section.

- SPOINT,id_list defines and displays modal amplitude.
- SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.
- SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.
- RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.
- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).
- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main Bulk Data Section.
- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).
- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).
- SUPORT1,SID,IDi,Ci defines the static support for a preload condition with a residual-only run. This entry is case control selectable. Do not use SUPORT.
- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in RecurDyn.
- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

18. MBDEXPORT and ADAMSMNF case control entries cannot be used in the same analysis run. In other words, a RecurDyn RFI file or an ADAMS MNF file can be generated during a particular NX Nastran execution, but not both files at the same time. Attempting to generate both files in the same analysis will cause an error to be issued and the execution to be terminated.

19. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from an RecurDyn/Flex analysis. This option requires the following assignment command:

    ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20
    FORM=UNFORM

be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be used as input into an NX Nastran SOL 103 run using the MBDRECVR case control capability to perform results recovery from an RecurDyn/Flex analysis. FLEXBODY=YES is required with its use.

The data blocks output are:

MGGEW - physical mass external sort with weight mass removed
MAAEW - modal mass
KAAE - modal stiffness
CMODEXT - component modes.

This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

20. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06.

21. The MBDEXPORT data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using MBDEXPORT. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

ADAMS STANDARD Describers:

Describer		Meaning
FLEXBODY		Requests the generation of MNF.
	NO	Standard NX Nastran solution without MNF creation. (default)
	YES	MNF generation requested.
FLEXONLY		Determines if DMAP solution runs or not after MNF creation is complete.
	YES	Only MNF creation occurs. (default)
	NO	MNF file creation occurs along with standard DMAP solution.
MINVAR		Determines how mass invariants are computed.
	PARTIAL	Mass invariants 5 and 9 are not computed. (default)

Describer	Meaning
CONSTANT	Mass invariants 1,2,6 and 7 are computed.
FULL	All nine mass invariants are computed.
NONE	No mass invariants are computed.
PSETID	Selects a set of elements defined in the OUTPUT(PLOT) section (including PLOTEL) or on a sketch file whose connectivity is exported to face geometry to be used in ADAMS. (See Remark 15)
NONE	All grids, geometry and associated modal data is written to MNF. (default)
setid	The connectivity of a specific element set is used to export face geometry.
ALL	The connectivity of all element sets are used to export face geometry.
sktunit	The connectivity of element faces defined on a sketch file is used to export face geometry. Note that the value must be a negative number to distinguish it from a setid value.
OUTGSTRS	Determines if grid point stress is written to MNF.
NO	Do not write grid point stress to MNF. (default)
YES	Write grid point stress to MNF.
OUTGSTRN	Determines if grid point strain is written to MNF.
NO	Do not write grid point strain to MNF. (default)
YES	Write grid point strain to MNF.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post processing of ADAMS/Flex results.
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.

Describer		Meaning
CHECK		Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 18)
	NO	No debug output will be written. (default)
	YES	Debug output will be written.

ADAMS STANDARD Remarks:

1. The creation of the Adams MNF, which is applicable in a non-restart SOL 103 analysis only, is initiated by MBDEXPORT ADAMS FLEXBODY=YES (other describers are optional) and the inclusion of the bulk data entry DTI,UNITS. MNF files are named 'jid_seid.mnf', where seid is the integer number of the superelement (0 for residual-only run). The location of these files is the same directory as the jid.f06 file.

2. The Data Table Input Bulk Data entry DTI,UNITS, which is required for an MBDEXPORT ADAMS FLEXBODY=YES run, is used to specify the system of units for the data stored in the MNF. Unlike NX Nastran, ADAMS is not a unitless code. Once identified, the units will apply to all superelements in the model. The complete format is:

```
DTI     UNITS   1       MASS    FORCE   LENGTH  TIME
```

   All entries are required. Acceptable character strings are listed below.

   Mass:

   KG - kilogram

   LBM – pound-mass (0.45359237 kg)

   SLUG – slug (14.5939029372 kg)

   GRAM – gram (1E-3 kg)

   OZM – ounce-mass (0.02834952 kg)

   KLBM – kilo pound-mass (1000 lbm) (453.59237 kg)

   MGG – megagram (1E3 kg)

   MG – milligram (1E-6 kg)

   MCG – microgram (1E-9 kg)

   NG – nanogram (1E-12 kg)

   UTON – U.S. ton (907.18474 kg)

SLI – slinch (175.1268352 kg)

Force:

N – Newton

LBF – pound-force (4.44822161526 N)

KGF – kilograms-force (9.80665 N)

OZF – ounce-force (0.2780139 N)

DYNE – dyne (1E-5 N)

KN – kilonewton (1E3 N)

KLBF – kilo pound-force (1000 lbf) (4448.22161526 N)

MN – millinewton (1E-3 N)

MCN – micronewton (1E-6 N)

NN – nanonewton (1E-9 N)

Length:

M – meter

KM – kilometer (1E3 m)

CM – centimeter (1E-2 m)

MM – millimeter (1E-3 m)

MI – mile (1609.344 m)

FT – foot (0.3048 m)

IN – inch (25.4E-3 m)

MCM – micrometer (1E-6 m)

NM – nanometer (1E-9 m)

A – Angstrom (1E-10 m)

YD – yard (0.9144 m)

ML – mil (25.4E-6 m)

MCI – microinch (25.4E-9 m)

Time:

S – second

H – hour (3600.0 sec)

MIN-minute (60.0 sec)

MS – millisecond (1E-3 sec)

MCS – microsecond (1E-6 sec)

NS – nanosecond (1E-9 sec)

D – day (86.4E3 sec)

3. Since DTI,UNITS determines all units for the MNF, the units defined in WTMASS, which are important for units consistency in NX Nastran, are ignored in the output to the MNF. For example, if the model mass is in kilograms, force in Newtons, length in meters, and time in seconds, then WTMASS would equal 1, ensuring that NX Nastran works with the consistent set of kg, N, and m. The units written to the MNF would be: “DTI,UNITS,1,KG,N,M,S”.

4. You can create flexible body attachment points by defining the component as a superelement or part superelement, in which case the physical external (a-set) grids become the attachment points. For a residual-only type model, you can use standard NX Nastran ASET Bulk Data entries to define the attachment points.

5. The nine mass variants are:

$$\underset{1 \times 1}{{}^{1}I} = \sum_{p=1}^{N} m_p$$

$$\underset{3 \times 1}{{}^{2}I} = \sum_{p=1}^{N} m_p s_p$$

$$\underset{3 \times M}{{}^{3}I_j} = \sum_{p=1}^{N} m_p \Phi_p \quad j = 1, \ldots, M$$

$$\underset{3 \times M}{{}^{4}I} = \sum_{p=1}^{N} m_p \tilde{s}_p \underline{\bar{\Phi}}_p + I_p \Phi_p^{*}$$

$$\underset{3 \times M}{{}^{5}I_j} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1, \ldots, M$$

$$\underset{M \times M}{{}^{6}I} = \sum_{p=1}^{N} m_p \Phi_p^T \Phi_p + \Phi_p^{*T} I_p \Phi_p^{*}$$

$$\underset{3 \times 3}{{}^{7}I} = \sum_{p=1}^{N} m_p \tilde{s}_p^T \tilde{s}_p + I_p$$

$$\underset{3 \times 3}{{}^{8}I_j} = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj} \quad j = 1, \ldots, M$$

$$\underset{3 \times 3}{{}^{9}I_{jk}} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \phi_{pk} \quad j, k = 1, \ldots, M$$

$s_p = [xyz]^T$ are the coordinates of grid point p in the basic coordinate system.

$$\tilde{s}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix} = \text{skew symmetric vector cross product}$$

operator.

f $_p$ = partitioned orthogonal modal matrix that corresponds to the translational degrees of freedom of grid *p*.

I_p = inertia tensor *p*.

f $_p^*$ = partitioned orthogonal modal matrix that corresponds to the rotational degrees of freedom of grid *p*.

$\tilde{\phi}_{pf}$ = skew-symmetric matrix formed for each grid translational degree of freedom for each mode.

M = number of modes.

N = number of grids.

6. To accurately capture the mode shapes when supplying SPOINT/QSET combinations, the number of SPOINTS (ns) should be at least ns=n+(6+p), assuming that residual flexibility is on. In the above equation for ns, the number

of modes (n) is specified on the EIGR (METHOD=LAN) or EIGRL Bulk Data entries; the number of load cases is p. In general, you cannot have too many SPOINTs, as excess ones are truncated with no performance penalty.

7. For FLEXBODY=YES runs, residual vectors for the component should always be calculated as they result in a more accurate representation of the component shapes at little additional cost.

8. OMIT or OMIT1 Bulk Data entries are not supported.

9. Lumped mass formulation (default) is required. Either leave PARAM,COUPMASS out of the input file or supply PARAM,COUPMASS,-1 (default) to ensure lumped mass.

10. P-elements are not allowed because they always use a coupled mass formulation. Likewise, the MFLUID fluid structure interface is not allowed because the virtual mass matrix it generates is not diagonal.

11. PARAM,WTMASS,value with a value other than 1.0 may be used with an NX Nastran run generating an MNF. It must have consistent units with regard to the DTI,UNITS Bulk Data entry. Before generating the MNF, NX Nastran will appropriately scale the WTMASS from the physical mass matrix and mode shapes.

12. There is a distinction between how an MBDEXPORT ADAMS FLEXBODY=YES run handles element-specific loads (such as a PLOAD4 entry) versus those that are grid-specific (such as a FORCE entry), especially when superelements are used. The superelement sees the total element-specific applied load. For grid-specific loads, the loads attached to an external grid will move downstream with the grid. That is to say, it is part of the boundary and not part of the superelement. This distinction applies to a superelement run and not to a residual-only or parts superelement run.

13. The loads specified in NX Nastran generally fall into two categories: non-follower or fixed direction loads (non-circulatory) and follower loads (circulatory). The follower loads are nonconservative in nature. Examples of fixed direction loads are the FORCE entry or a PLOAD4 entry when its direction is specified via direction cosines. Examples of follower loads are the FORCE1 entry or the PLOAD4 entry when used to apply a normal pressure. By default in NX Nastran, the follower loads are always active in SOL 103 and will result in follower stiffness being added to the differential stiffness and elastic stiffness of the structure. In a run with MBDEXPORT ADAMS FLEXBODY=YES and superelements, if the follower force is associated with a grid description (such as a FORCE1) and the grid is external to the superelement, the follower load will move downstream with the grid. Thus, the downstream follower contribution to the component's stiffness will be lost, which could yield poor results. This

caution only applies to a superelement run and not to a residual-only or a part superelement run.

14. OUTGSTRS and OUTGSTRN entries require the use of standard NX Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN= commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain print to the NX Nastran .f06 file.

15. To reduce the FE mesh detail for dynamic simulations, PSETID (on the MBDEXPORT Case Control command) defined with a SET entry (i.e. setid) is used to define a set of PLOTELs or other elements used to select grids to display the components in ADAMS. This option can significantly reduce the size of the MNF without compromising accuracy in the ADAMS simulation providing that the mass invariant computation is requested. With superelement analysis, for any of these elements that lie entirely on the superelement boundary (all of the elements' grids attached only to a-set or exterior grids), a SEELT Bulk Data entry must be specified to keep that display element with the superelement component. This can also be accomplished using PARAM, AUTOSEEL,YES. The SEELT entry is not required with parts superelements, as boundary elements stay with their component.

    If the SET entry points to an existing set from the OUTPUT(PLOT) section, this single set is used explicitly to define elements used to select grids to display the component in ADAMS. If PSETID does not find the set ID in OUTPUT(PLOT), it will search sets in the case control for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements' sets, the union of which will be used to define a set of PLOTELs or other elements used to select grids to display the component in ADAMS. If the user wishes to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

    The following element types are not supported for writing to an MNF, nor are they supported as a 'type' entry in a set definition in OUTPUT(PLOT): CAABSF, CAEROi, CDUMi, CHACAB, CHACBR, CHBDYx, CDAMP3, CDAMP4, CELAS3, CELAS4, CFLUIDi, CMASS3, CMASS4, CRAC2D, CRAC3D, CTRMEM, CTWIST, CWEDGE, CWELD, and GENEL.

    PSETID can also point to a sketch file using PSETID= – sktunit, where sktunit references an ASSIGN statement of the form:

    ```
    ASSIGN SKT='sketch_file.dat',UNIT=sktunit.
    ```

    The grids defined for the elements' faces in the sketch file, along with *all* external (i.e. boundary) grids for the superelements, will be the only grids (and their associated data) written to the MNF.

    The format of the sketch file, which describes the mesh as a collection of faces, must be as follows:

```
face_count
face_1_node_count face_1_nodeid_1 face_1_nodeid_2 ...
face_2_node_count face_2_nodeid_1 face_2_nodeid_2 ...

<etc>
```

Faces must have a node count of at least two. For example, a mesh comprised of a single brick element might be described as follows:

```
6
4 1000 1001 1002 1003
4 1007 1006 1005 1004
4 1000 1004 1005 1001
4 1001 1005 1006 1002
4 1002 1006 1007 1003
4 1003 1007 1004 1000
```

Alternatively, the mesh might be described as a stick figure using a collection of lines (two node faces), as shown below:

```
8
2 101 102
2 102 103
2 103 104
2 104 105
2 105 106
2 106 107
2 107 108
2 108 109
```

16. Typical NX Nastran data entry requirements are described below.

Typical Parameters:

- PARAM,RESVEC,character_value – controls calculation of residual vector modes.
- PARAM,GRDPNT, value - mass invariants ^{1}I, ^{2}I, and ^{7}I will be computed using results of NX Nastran grid point weight generator execution in the basic coordinate system.

Typical Case Control:

- MBDEXPORT ADAMS FLEXBODY=YES is required for MNF generation.
- METHOD=n is required before or in the first subcase for modal solutions.
- SUPORT1=seid is necessary to select a static support set for a residual only linear preload run.

- SUPER=n,SEALL=n is useful with multiple superelement models to select an individual superelement as a flexible body. Cannot be used with a linear STATSUB(PRELOAD) run.

- OUTPUT(PLOT) is necessary to define elements used to select grids to display the component in ADAMS when PSETID=ALL or setid.

  SET n=list of elements (including PLOTELs) is used to select grids to display the component.

- OUTPUT(POST) is necessary to define volume and surface for grid stress or strain shapes.

  SET n=list is a list of elements for surface definition for grid stress or strain shapes.

  Stress and strain data in the MNF is limited to the six components (i.e. 3 normal and 3 shear) for a grid point for a given mode.

  SURFACE n SET n NORMAL z3 is used to define a surface for writing stress and strain data. Only one FIBER selection is allowed for each SURFACE, thus the use of the FIBRE ALL keyword on the SURFACE case control command will write stresses to the MNF at the Z1 fiber location only.

  Since the FIBRE keyword only applies to stresses, strain data will always be written to the MNF at the MID location.

  Stress and strain data at grid points can only be written to the MNF for surface and volume type elements (e.g. CQUAD and CHEXA).

  VOLUME n SET n is a volume definition.

  The default SYSTEM BASIC is required with SURFACE or VOLUME.

- STRESS(PLOT) is necessary for stress shapes.

- STRAIN(PLOT) is necessary for strain shapes.

- GPSTRESS(PLOT) is necessary for grid point stress shapes to be included in the MNF.

- GPSTRAIN(PLOT) is necessary for grid point strain shapes to be included in the MNF.

  Typical Bulk Data:

- DTI,UNITS,1,MASS,FORCE,LENGTH,TIME is required for MNF generation. For input files containing superelements, this command must reside in the main bulk data section.

- SPOINT,id_list defines and displays modal amplitude.SESET,SEID,grid_list defines a superelement (see GRID and BEGIN BULK SUPER=). The exterior grids will represent the attachment points along with the q-set.

- SEELT,SEID,element_list reassigns superelement boundary elements to an upstream superelement.

- RELEASE,SEID,C,Gi is an optional entry that removes DOFs from an attachment grid for which no constraint mode is desired. For example, this allows the removal of rotational degrees of freedom from an analysis where only translational degrees of freedom are required.

- SEQSET,SEID,spoint_list defines modal amplitudes of a superelement (see SEQSET1).

- SENQSET,SEID,N defines modal amplitudes of a part superelement. It must reside in the main Bulk Data Section.

- ASET,IDi,Ci defines attachment points for a residual-only run (see ASET1).

- QSET1,C,IDi defines modal amplitudes for the residual structure or modal amplitudes for a part superelement (see QSET).

- SUPORT1,SID,IDi,Ci defines the static support for a preload condition with a residual-only run. This entry is case control selectable. Do not use SUPORT.

- PLOTEL,EID,Gi can be used, along with existing model elements, to define elements used to select grids to display the components in ADAMS.

- EIGR,SID,METHOD,… obtains real eigenvalue extraction (see EIGRL).

17. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from an ADAMS/Flex analysis. This option requires the following assignment command:

    ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20
    FORM=UNFORM

    be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which then can be used as input into an NX Nastran SOL 103 run using the MBDRECVR case control capability to perform results recovery from an ADAMS/Flex analysis. FLEXBODY=YES is required with its use.

    The data blocks output are:

MGGEW - physical mass external sort with weight mass removed
MAAEW - modal mass
KAAE - modal stiffness
CMODEXT - component modes.

This capability is limited to no more than one superelement per NX Nastran model. Residual-only analyses are supported.

18. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

19. The MBDEXPORT data routines use the environment variable TMPDIR for temporary storage during the processing of mode shape data. As a result, TMPDIR must be defined when using MBDEXPORT. TMPDIR should equate to a directory string for temporary disk storage, preferably one with a large amount of free space.

20. If any damping is defined in the model, an equivalent modal viscous damping will be determined for each mode and written to the MNF. This equivalent modal viscous damping is defined as:

$$\begin{bmatrix} \ddots & & \\ & d & \\ & & \ddots \end{bmatrix} = y^T B_e y$$

where d = equivalent modal viscous damping
y = mode shapes
B_e = equivalent viscous damping defined by:

$$B_e = B^1_{AA} + B^2_{AA} + \frac{G}{W3} K^1_{AA} + \frac{1}{W4} K^4_{AA}$$

G, W3, and W4 are parameters described in the “Parameter Descriptions” section of this guide.

OP4 Describers:

Describer	Meaning
unit	The OP4 file is written to the specified logical unit number. The logical unit number must match the unit number on an ASSIGN statement.

Describer	Meaning
FLEXBODY	Requests the generation and writing of standard or state-space matrices to an OP4 file.
NO	NX Nastran solution without standard or state-space matrix generation. (default)
YES	Standard or state-space matrix generation requested.
FLEXONLY	Determines if DMAP solution runs or not after standard or state-space matrix generation is complete.
YES	Only standard or state-space matrix generation occurs. (default)
NO	Standard or state-space matrix generation occurs along with the standard DMAP solution.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post-processing of controls results. (See Remark 5)
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 6).
NO	No debug output will be written. (default)
YES	Debug output will be written.

OP4 Remarks:

1. The generation of standard or state-space matrices and the writing of them to an OP4 file via OUTPUT4, which is applicable in a non-restart SOL 103 analysis only, is initiated by MBDEXPORT OP4=unit STANDARD FLEXBODY=YES, or MBDEXPORT OP4=unit STATESPACE FLEXBODY=YES (other describers are optional) and the inclusion of the ASSIGN file management statement. This ASSIGN statement must be of the form:

   ASSIGN OUTPUT4=’filename’,UNIT=n,etc.

   where ‘n’ matches the value for unit on the MBDEXPORT OP4=unit case control command.

2. The parameters LFREQ/HFREQ or LMODES can be used to control which modes are used to derive the standard or state-space matrices.

3. For state-space matrices, user-defined set U7 is used for input DOF. User-defined set U8 is used for output DOF. Refer to the USET and USET1 bulk data entries.

4. For standard matrices, user-defined set U8 is used for output DOF. The mode shape output will be reduced to the DOF defined in DOF set U8. If DOF set U8 is not defined, the mode shape data for all DOF will be written. Refer to the USET and USET1 bulk data entries.

5. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from a system analysis. This option requires the following assignment command:

   ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

   be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which can then be used as an input into an NX Nastran SOL 103 run using the MBDRECVR case control command. FLEXBODY=YES is required when specifying RECVROP2=YES.

   The data blocks output are:

   MGGEW – physical mass external sort with weight mass removed

   MAAEW – modal mass

   KAAE – modal stiffness

   CMODEXT – component modes

   This capability is limited to one superelement per NX Nastran model. Residual-only analyses are supported.

6. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

MATLAB Describers:

Describer	Meaning
FLEXBODY	Requests the generation and writing of standard or state-space matrices to a MATLAB script file.

Describer	Meaning
NO	NX Nastran solution without standard or state-space matrix generation. (default)
YES	Standard or state-space matrix generation requested.
FLEXONLY	Determines if DMAP solution runs or not after standard or state-space matrix generation is complete.
YES	Only standard or state-space matrix generation occurs. (default)
NO	Standard or state-space matrix generation occurs along with the standard DMAP solution.
RECVROP2	Requests that the FLEXBODY run output an NX Nastran OP2 file for use in post-processing of controls results. (See Remark 7)
NO	OP2 file will not be generated. (default)
YES	OP2 file will be generated.
CHECK	Requests debug output be written to the f06 file when RECVROP2=YES. (See Remark 8).
NO	No debug output will be written. (default)
YES	Debug output will be written.

MATLAB Remarks:

1. The generation of standard or state-space matrices and the writing of them to a MATLAB script file, which is applicable in a non-restart SOL 103 analysis only, is initiated by MBDEXPORT MATLAB STANDARD FLEXBODY=YES, or MBDEXPORT MATLAB STATESPACE FLEXBODY=YES (other describers are optional). The MATLAB script files are named jid_seid.m where seid is the integer number of the superelement (0 for residual-only run). The location of the MATLAB script files is the same directory as the jid.f06 file.

2. The parameters LFREQ/HFREQ or LMODES can be used to control which modes are used to derive the standard or state-space matrices.

3. For state-space matrices, user-defined set U7 is used for input DOF. User-defined set U8 is used for output DOF. Refer to the USET and USET1 bulk data entries.

4. For standard matrices, user-defined set U8 is used for output DOF. The mode shape output will be reduced to the DOF defined in DOF set U8. If DOF set U8 is not defined, the mode shape data for all DOF will be written. Refer to the USET and USET1 bulk data entries.

5. For the state-space option, the MATLAB script file contains the [A], [B], and [C] state-space matrices. They are defined as AMAT, BMAT, and CMAT, respectively. The input and output DOF are defined as U7DOF and U8DOF, respectively with the first column being the grid ID and the second column being the direction code (1 through 6).

6. For the standard option, the MATLAB script file contains the modal mass, equivalent modal viscous damping, modal stiffness, mode shapes, and modal forces defined as MMASS, MDAMP, MSTIF, MSHAP, and MFORC, respectively. The physical DOF corresponding one-to-one with the rows of MSHAP are defined as U8DOF. The first column contains the grid ID and the second column contains the direction code (1 through 6).

7. The RECVROP2=YES option is used when you would like results recovery (using the MBDRECVR case control entry) from a system analysis. This option requires the following assignment command:

   ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20
   FORM=UNFORM

   be inserted into the file management section of the NX Nastran input file. It will cause an OP2 file with a .out extension to be generated, which can then be used as an input into an NX Nastran SOL 103 run using the MBDRECVR case control command. FLEXBODY=YES is required when specifying RECVROP2=YES.

   The data blocks output are:

   MGGEW – physical mass external sort with weight mass removed

   MAAEW – modal mass

   KAAE – modal stiffness

   CMODEXT – component modes

   This capability is limited to one superelement per NX Nastran model. Residual-only analyses are supported.

8. Setting CHECK=YES (which is only available when RECVROP2=YES) is *not* recommended for models of realistic size due to the amount of data that will be written to the f06 file.

Structural damping

In order to include the effects of structural damping in a modal transient analysis (SOL 112), you must convert structural damping to viscous damping.

Prior to NX Nastran 7.1, there was only one method available to convert structural damping to viscous damping for a modal transient analysis. When using this method, you must specify either one or two frequency factors using the W3 and W4 parameters. The software uses these frequency factors to convert the structural damping to viscous damping. Typically, you specify the frequency factors to account for the dominant frequency at which the damping is active. For more information regarding the W3 and W4 parameters, see the *Basic Dynamic Analysis User's Guide*.

Now a second method is available that you can use either independently or in conjunction with the original method. When you use it independently, you do not need to specify any frequency factors to use in the conversion. The software automatically uses the solved modal frequencies in the conversion. As a result, the conversion is valid over a much wider range of frequencies.

If you use the original structural-to-viscous conversion method, you must specify one or both of the W3 and W4 parameters. With these parameters specified, the structural damping is converted to viscous damping using:

$$\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]$$

where

$\left[K^1_{dd}\right]$ = assembled stiffness matrix.

$\left[K^4_{dd}\right]$ = assembled elemental damping matrix.

G = overall structural damping coefficient. (See the G parameter.)

$W3$ = user-defined conversion frequency for overall structural damping. (See the W3 parameter.)

$W4$ = user-defined conversion frequency for elemental structural damping. (See the W4 parameter.)

Then, the modal viscous damping matrix is calculated using:

$$\left[B_{qq}\right]=\left[\Phi^T_{dq}\right]\left(\left[B^1_{dd}\right]+\left[B^2_{dd}\right]+\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right]$$

where

$\left[B_{qq}\right]$ = modal viscous damping matrix.

$\left[\Phi_{dq}\right]$ = mode shape matrix.

$\left[B^1_{dd}\right]$ = assembled viscous damping matrix.

$\left[B^2_{dd}\right]$ = viscous damping matrix from DMIG input.

$\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]$ = converted structural damping matrix.

Beginning with NX Nastran 7.1, a new parameter named WMODAL is available that allows you to specify that structural damping be converted to viscous damping using:

$$\begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix} \left[\Phi^T_{dq}\right]\left(G\left[K^1_{dd}\right]+\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right] \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix}$$

where w_i are the modal frequencies in rad/sec.

For rigid body modes, the software partitions the diagonal entries, $1/\omega_i$, out of the calculation. The FZERO parameter is used to identify rigid body modes. (See the FZERO parameter.) Because all the modal frequencies of flexible body modes are used in the conversion, the resulting viscous damping is representative over a much wider range of frequencies.

Because the WMODAL parameter can be used in conjunction with the W3 and W4 parameters, you now have three options for converting structural damping to viscous damping.

- If you specify one or both of the W3 and W4 parameters, and either omit the WMODAL parameter from the input file or specify WMODAL = NO, the structural-to-viscous damping conversion is calculated using:

  $$\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]$$

  Therefore, the modal viscous damping matrix is calculated using:

  $$\left[B_{qq}\right]=\left[\Phi^T_{dq}\right]\left(\left[B^1_{dd}\right]+\left[B^2_{dd}\right]+\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right]$$

  This is identical to the method available prior to NX Nastran 7.1.

- If you specify WMODAL = YES, and either omit the W3 and W4 parameters from the input file or accept their default value of 0.0, the structural-to-viscous damping conversion is calculated using:

$$\begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix} \left[\Phi^T_{dq}\right]\left(G\left[K^1_{dd}\right]+\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right] \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix}$$

  Therefore, the modal viscous damping matrix is calculated using:

$$\left[B_{qq}\right]=\left[\Phi^T_{dq}\right]\left(\left[B^1_{dd}\right]+\left[B^2_{dd}\right]\right)\left[\Phi_{dq}\right] + \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix} \left[\Phi^T_{dq}\right]\left(G\left[K^1_{dd}\right]+\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right] \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix}$$

  You do not need to specify any frequency factors to use in the conversion if you use this option.

- If you specify WMODAL = YES and specify one or both of the W3 and W4 parameters, the contributions from both structural-to-viscous damping conversion methods are summed and the modal viscous damping matrix is calculated using:

$$\left[B_{qq}\right]=\left[\Phi^T_{dq}\right]\left(\left[B^1_{dd}\right]+\left[B^2_{dd}\right]+\frac{G}{W3}\left[K^1_{dd}\right]+\frac{1}{W4}\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right] + \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix} \left[\Phi^T_{dq}\right]\left(G\left[K^1_{dd}\right]+\left[K^4_{dd}\right]\right)\left[\Phi_{dq}\right] \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix}$$

An example of a complete input file containing the WMODAL parameter is *wmod_rbm.dat*. It can be found in *install_dir/nxn7p1/nast/tpl*.

WMODAL parameter

The WMODAL parameter is only applicable for SOL 112 and can be placed in either the bulk data or case control sections of the input file.

WMODAL Default = NO

WMODAL specifies a structural-to-viscous damping conversion method that uses the solved modal frequencies as conversion factors. WMODAL is only applicable to modal transient analysis (SOL 112). When one or both of the W3 and W4 parameters are specified (see the W3, W4, W3FL, W4FL parameter) in conjunction with WMODAL, the structural-to-viscous damping conversion is calculated using:

$$[B_{qq}] = [\Phi^T_{dq}]\left([B^1_{dd}] + [B^2_{dd}] + \frac{G}{W3}[K^1_{dd}] + \frac{1}{W4}[K^4_{dd}]\right)[\Phi_{dq}]$$
$$+ \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix} [\Phi^T_{dq}]\left(G[K^1_{dd}] + [K^4_{dd}]\right)[\Phi_{dq}] \begin{bmatrix} 1/\omega_1 & 0 & \cdots & 0 \\ 0 & 1/\omega_2 & \cdots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \cdots & 1/\omega_q \end{bmatrix}$$

where w_i are the modal frequencies in rad/sec.

For rigid body modes, the diagonal entries, $1/\omega_i$, are partitioned out of the calculation. The FZERO parameter is used to identify rigid body modes (see the FZERO parameter).

DDAM

NX Nastran 7.1 contains four enhancements to DDAM analysis.

- Results from a DDAM analysis can now be written to an .op2 file.
- Any consistent set of units can now be used in the NX Nastran model for a DDAM analysis.
- The formula for reference acceleration, A_o, has now been generalized to support all DDAM specifications including NRL-1396.
- The built-in weighting factors used to compute reference velocity and reference acceleration in the default user option are now set to values specified in NRL-1396.

DDAM results output

Beginning with NX Nastran 7.1, you can write results from standard case control requests to an .op2 file. Prior to NX Nastran 7.1, you could write the results from standard case control requests only to an .f06 file. The binary formatting of .op2 files is ideal for accessing the results with standard post processors. To write results to an .op2 file, you should include PARAM,POST,n<0 in either the case control or bulk entry sections of the input file.

DDAM units

Beginning with NX Nastran 7.1, you can use any consistent set of units in the NX Nastran model for a DDAM analysis. Prior to NX Nastran 7.1, you could run a DDAM analysis only if the model unit for force was lbf and the model unit for acceleration was in/sec^2.

In order to change the units from lbf and in/sec^2, you specify a force conversion factor c_f and an acceleration conversion factor c_a. The conversion factors c_f and c_a are defined in the DDAM input (.inp) file as lines 10 and 11, respectively.

First Line - spectrum control - format a1,1x,a1

- First Item - coefficients from external file or built-in source
  - T=use coefficients from external file
  - F=use built-in coefficients
- Second item - DDAM or general spectrum run flag
  - T=general non-DDAM spectrum run
  - F=DDAM

Second Line - file name (if needed) -format a80

- If 1st item on line 1 is T, name of external coefficient file
- If 2nd item on line 1 is T, name of external spectrum file
- If neither are T, line is not needed and should be omitted – do not leave a blank line

Third Line - location flags - format i1,1x,i1,1x,i1

- First Item - surface or submarine
  - 1=surface
  - 2=submarine
- Second Item - equipment location
  - 1=deck
  - 2=hull
  - 3=shell
- Third Item - coefficient type
  - 1=elastic
  - 2=elastic/plastic

4th Line - Weight cutoff percentage m_{cutoff} - format F8.3 (0. To 100.)

5th Line - Axis Orientation - format a1,1x,a1

First Item - F/A axis,X,Y, or Z

Second Item - vertical axis X,Y, or Z

6th Line - Input file name -format a80

7th Line - Output file name - format a80

8th Line - Verification file name - format a80

9th Line - Minimum acceleration value g_{min} (units of G's) - format F8.3 (must be >0.0)

10th Line - Force conversion factor c_f (converts lbf to model force units) - format F8.3 (must be >0.0)

11th Line - Acceleration conversion factor c_a (converts in/sec^2 to model acceleration units) - format F8.3 (must be >0.0)

Note

You should always include values for lines 9, 10, and 11. If the desired DDAM analysis units for force are lbf, enter 1.0 for line 10. If the desired DDAM analysis units for acceleration are in/sec^2, enter 1.0 for line 11. You can verify DDAM inputs by examining the verification file specified on line 8.

The three control file formats have changed to reflect the new unit conversion capability and are now:

Default user option:

F F

nsurf nstruc nplast

m_{cutoff} (user-supplied value)

f/a_axis vert_axis

.f11 filename

.f13 filename

.ver filename

g_{min} (user-supplied value)

c_f (user-supplied value)

c_a (user-supplied value)

User coefficient option:

T F
.dat filename
nsurf nstruc nplast
m_{cutoff} (user-supplied value)
f/a_axis vert_axis
.f11 filename
.f13 filename
.ver filename
g_{min} (user-supplied value)
c_f (user-supplied value)
c_a (user-supplied value)

User spectrum option:

F T
.dat filename
m_{cutoff} (user-supplied value)
f/a_axis vert_axis
.f11 filename
.f13 filename
.ver filename
g_{min} (user-supplied value)
c_f (user-supplied value)
c_a (user-supplied value)

An example DDAM input (.inp) file that uses N for the model force units and mm/sec^2 for the model acceleration units is:

```
F F
1 1 1
75.
X Z
navs1.f11
navs1.f13
navs1.ver
6.
4.448
25.4
```

Note

Because the line 1 specification is “F F”, this example is the default option and line 2 is not needed and is omitted from the file without leaving a blank line.

An example DDAM input (.inp) file that uses lbf for the model force units and ft/sec^2 for the model acceleration units is:

```
T F
navs1.dat
1 1 1
75.
X Z
navs1.f11
navs1.f13
navs1.ver
6.
1.
0.08333
```

Note

Because the line 1 specification is “T F”, this example is the user coefficient option and line 2 is required.

DDAM reference acceleration

Beginning with NX Nastran 7.1, the formula for reference acceleration, A_o, has been generalized to support all DDAM specifications including NRL-1396. The reference acceleration is now calculated using:

$$A_o = AF \frac{AA(AB + M)(AC + M)}{(AD + M)^2}$$

where M is modal weight in kips and *AA*, *AB*, *AC*, *AD*, and *AF* are weighting factors. The built-in weighting factors are used for the default user option. For the user coefficient option, you must specify the weighting factors in an external coefficient file.

You can optionally specify the alternative form of the reference acceleration equation by setting *AD* = *AC*. By doing so, the reference acceleration equation simplifies to:

$$A_o = AF \frac{AA(AB + M)}{(AC + M)}$$

DDAM weighting factors

NAVSHOCK has built-in weighting factors that are used to calculate the reference velocity and reference acceleration for the default user option. A default value for the weight cutoff percentage of 80.0 is also included with the built-in weighting factors in NAVSHOCK.

Beginning with NX Nastran 7.1, the built-in weighting factors are set to values specified in NRL-1396. The following tables list the values specified in NRL-1396. In both tables:

- nsurf – Refers to the ship type. Allowable values are SUB (submerged) and SURF (surfaceship).
- nstruc – Refers to the mounting location. Allowable values are DECK, HULL, and SHELL.
- nplast – Refers to elastic or elastic-plastic factors. Allowable values are ELASTIC and ELPL.

In the following table, (1), (2), and (3) after VF and AF refer to directions:

- (1) = fore/aft
- (2) = athwartship
- (3) = vertical

nsurf/nstruc/nplast	VF(1)	VF(2)	VF(3)	AF(1)	AF(2)	AF(3)
SURF/DECK/ELASTIC	0.4	0.4	1.0	0.4	0.4	1.0
SURF/HULL/ELASTIC	0.2	0.4	1.0	0.2	0.4	1.0
SURF/SHELL/ELASTIC	0.1	0.2	1.0	0.1	0.2	1.0
SURF/DECK/ELPL	0.2	0.2	0.5	0.4	0.4	1.0
SURF/HULL/ELPL	0.1	0.2	0.5	0.2	0.4	1.0
SUB/DECK/ELASTIC	0.8	2.0	1.0	0.8	2.0	1.0
SUB/HULL/ELASTIC	0.4	1.0	1.0	0.4	1.0	1.0
SUB/SHELL/ELASTIC	0.08	0.2	1.0	0.08	0.2	1.0
SUB/DECK/ELPL	0.4	1.0	0.5	0.8	2.0	1.0
SUB/HULL/ELPL	0.2	0.5	0.5	0.4	1.0	1.0

nsurf/nstruc/nplast	VA	VB	VC	AA	AB	AC	AD
SURF/DECK/ELASTIC	30.0	12.0	6.0	10.0	37.5	12.0	6.0
SURF/HULL/ELASTIC	60.0	12.0	6.0	20.0	37.5	12.0	6.0
SURF/SHELL/ELASTIC	120.0	12.0	6.0	40.0	37.5	12.0	6.0
SURF/DECK/ELPL	30.0	12.0	6.0	10.0	37.5	12.0	6.0
SURF/HULL/ELPL	60.0	12.0	6.0	20.0	37.5	12.0	6.0
SUB/DECK/ELASTIC	10.0	480.0	100.0	5.2	480.0	20.0	20.0
SUB/HULL/ELASTIC	20.0	480.0	100.0	10.4	480.0	20.0	20.0
SUB/SHELL/ELASTIC	100.0	480.0	100.0	52.0	480.0	20.0	20.0
SUB/DECK/ELPL	10.0	480.0	100.0	5.2	480.0	20.0	20.0
SUB/HULL/ELPL	20.0	480.0	100.0	10.4	480.0	20.0	20.0

For information on formatting an external coefficient file, see “DDAM Process Automation” in the “Dynamic Solution Techniques” chapter of the *Advanced Dynamic Analysis User’s Guide*.

DDAM units mathematical basis

The conversion factors c_f and c_a specified in the DDAM input (.inp) file are used to convert the modal effective mass M_{eff} calculated by NX Nastran to modal weight M in kips using:

$$M = \frac{386 c_a M_{eff}}{1000 c_f}$$

The modal weight is subsequently used by NAVSHOCK to calculate the reference velocity and reference acceleration. The reference velocity is calculated using:

$$V_o = VF \frac{VA(VB+M)}{(VC+M)}$$

where *VA*, *VB*, *VC*, and *VF* are built-in or user-specified weighting factors. The reference acceleration is calculated using:

$$A_o = AF \frac{AA(AB+M)(AC+M)}{(AD+M)^2}$$

where *AA*, *AB*, *AC*, *AD*, and *AF* are built-in or user-specified weighting factors.

The acceleration loading calculated by NAVSHOCK for use in the response simulation requires conversion of the acceleration to the NX Nastran model units. This conversion is accomplished using:

$$A = c_a \max\left[386 g_{\min}, \min\left(386 A_o, \omega V_o\right)\right]$$

where W is the modal frequency in rad/sec. For clarity, this equation can also be written as:

$$A = \begin{cases} 386 c_a A_o & \omega > \dfrac{386 A_o}{V_o} \\ c_a V_o \omega & \dfrac{386 g_{\min}}{V_o} \le \omega \le \dfrac{386 A_o}{V_o} \\ 386 c_a g_{\min} & \omega < \dfrac{386 g_{\min}}{V_o} \end{cases}$$

DDAM documentation updates

The following updates are applicable to the "DDAM Process Automation" section of the "Dynamic Solution Techniques" chapter of the *NX Nastran 7 Advanced Dynamic Analysis User's Guide*.

Input file unit assignment

The unit to assign the DDAM input file is correct in the example, but is incorrect in the text. In both instances, you must assign unit 21 to the INPUT4 file that stores the input control options for the FORTRAN program. An example of an ASSIGN statement assigning unit 21 to the ddam.inp file is:

```
ASSIGN INPUTT4='ddam.inp', UNIT=21, FORM=FORMATTED
```

User coefficient option

When using the user coefficient option for the DDAM input (.inp) file, the documentation states that you must include a complete set of weighting factors in the DDAM coefficient file. In actuality, if you provide a blank DDAM coefficient file, the software will use the built-in default values in NAVSHOCK.

Note

As a best practice, you should specify a complete set of weighting factors in the DDAM coefficient file when using the user coefficient option for the DDAM input (.inp) file.

Weight cutoff percentage

You can specify a weight cutoff percentage in either the control file or the coefficient file. A default value for the weight cutoff percentage of 80.0 is included with the built-in weighting factors in NAVSHOCK. The weight cutoff percentage value that is used by the software in the DDAM calculations depends on the weight cutoff percentage value specified in the control file.

- If the weight cutoff percentage value specified in the control file is greater than 1.0 x 10^{-3}, it is used by the software in the DDAM calculations.

- If the weight cutoff percentage value specified in the control file is less than or equal to 1.0 x 10^{-3} and a value for the weight cutoff percentage is specified in the coefficient file, the value specified in the coefficient file is used by the software in the DDAM calculations.

- If the weight cutoff percentage value specified in the control file is less than or equal to 1.0 x 10^{-3} and a coefficient file either does not exist or does not include a value for the weight cutoff percentage, the default value of 80.0 included with the built-in weighting factors in NAVSHOCK is used by the software in the DDAM calculations.

  Note

  As a best practice, you should specify the weight cutoff percentage in the control file.

Optionally defined shock spectra

Information on how to define a shock spectra is included in the documentation, but details on how to format the DATATYP entry are omitted. You should be aware that for the DATATYP entry:

- Each field has eight spaces.

- The fields cannot be comma separated.

- Entries in each field must be left justified.

Modal effective mass

The MEFFMASS case control command requests modal effective mass, participation factors, and modal effective mass fractions in a SOL 103, normal modes analysis.

In addition to these requests, the new THRESH describer has been created to optionally limit mode output by the effective mass fractions. Modes which have an effective mass fraction greater than the value of THRESH in at least one translational or rotational direction, are output to the .f06 and .op2 files.

For example, if you define THRESH=0.1 on the MEFFMASS case control command:

```
MEFFMASS(PRINT,PLOT,FRACSUM,THRESH=0.1) = YES
```

only modes which have a modal effective mass fraction greater than 0.1 in at least one direction are written to the .op2 file.

The highlighted modal effective mass fraction values are greater than 0.1 in the example below. The result is that only modes 1, 2, 3, and 10 are output to the .f06 and .op2 files.

MODAL EFFECTIVE MASS FRACTION

MODE NO.	FREQUENCY	T1 FRACTION	T2 FRACTION	T3 FRACTION
1	1.168325E+01	3.967454E-01	5.778298E-01	2.201405E-08
2	1.170527E+01	5.778004E-01	3.967312E-01	2.689021E-07
3	8.607883E+01	4.693496E-08	5.450135E-08	9.836653E-01
4	2.862540E+02	3.634175E-05	2.135268E-04	3.179199E-08
5	2.908367E+02	2.402732E-04	4.129519E-05	4.061894E-07
6	2.914032E+02	4.727352E-06	2.782838E-05	1.604274E-09
7	2.915664E+02	1.946457E-12	1.922122E-11	3.420981E-06
8	9.267581E+02	1.951350E-03	1.816837E-02	1.968711E-06
9	9.365037E+02	1.831924E-02	2.131173E-03	1.559326E-07
10	1.293207E+03	1.409690E-07	4.893393E-09	4.310920E-07

MODE NO.	FREQUENCY	R1 FRACTION	R2 FRACTION	R3 FRACTION
1	1.168325E+01	5.901106E-01	4.051844E-01	1.785694E-05
2	1.170527E+01	4.051759E-01	5.900953E-01	1.732233E-06
3	8.607883E+01	9.956436E-08	4.067034E-08	8.259511E-08
4	2.862540E+02	4.898476E-05	8.325061E-06	1.945107E-02
5	2.908367E+02	9.466828E-06	5.514712E-05	2.253730E-09
6	2.914032E+02	6.356951E-06	1.079133E-06	4.836033E-05
7	2.915664E+02	1.836027E-11	2.891883E-11	1.007765E-11
8	9.267581E+02	3.773368E-03	4.238621E-04	3.774173E-06
9	9.365037E+02	4.277494E-04	3.773427E-03	3.724737E-05
10	1.293207E+03	4.922870E-11	9.597709E-08	2.375482E-01

The .f06 file includes a statement similar to the following to indicate the modes that are written to the .f06 and .op2 files:

```
MODES THAT EXCEED THE EFFECTIVE MASS THRESHOLD AND STORED ARE:
           1      2      3     10
```

Note that THRESH does not limit mode storage for consecutive dynamic response solutions (SOL 111 or 112). You can use the MODSEL case control command to select/deselect modes in these solutions.

See the updated MEFFMASS case control command.

44 MEFFMASS

Modal effective mass output request. Optionally limits mode output by effective mass.

MEFFMASS Modal effective mass output request. Optionally limits mode output by effective mass.

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis. Optionally can limit mode output by effective mass fraction.

Format:

```
MEFFMASS [ ( [ PRINT ], [GRID = grid], [THRESH= f ], [ SUMMARY, PARTFAC, ] ) ] = { YES }
             [ PUNCH ]                                [ MEFFM, MEFFW,     ]       { NO  }
             [ PLOT  ]                                [ FRACSUM, ALL      ]
```

Examples:

```
MEFFMASS
MEFFMASS(GRID=12, SUMMARY, PARTFAC)
MEFFMASS (PLOT, ALL, THRESH=0.001)=YES
```

Describers:

Describer	Meaning
PRINT	Writes output to the print file. (Default)
PUNCH	Writes output to the punch file.
PLOT	Writes output to the output2 file.
GRID	Reference grid point for the calculation of the Rigid Body Mass Matrix. The default is the origin of the basic coordinate system.
SUMMARY	Requests calculation of the Total Effective Mass Fraction, Modal Effective Mass Matrix, and the Rigid Body Mass Matrix. (Default)
PARTFAC	Requests calculation of Modal Participation Factors.
MEFFM	Requests calculation of the Modal Effective Mass in units of mass.
MEFFW	Requests calculation of the Modal Effective Mass in units of weight.
FRACSUM	Requests calculation of the Modal Effective Mass Fraction.

Describer	Meaning
f	Threshold value of effective mass fraction for mode output. (Real; 0.0 < THRESH < 1.0; Default = all modes are stored). See Remark 8.

Remarks:

1. The SUMMARY describer produces three outputs:

   Modal Effective Mass Matrix $[\epsilon^T][m][\epsilon]$ where

ϵ	=	Modal Participation Factors
	=	$[m]^{-1}[f]^T[M_{aa}][D_{ar}]$
m	=	Generalized mass matrix
f	=	Eigenvectors
M_{aa}	=	Mass matrix reduced to the a-set (g-set for superelements)
D_{ar}	=	Rigid body transformation matrix with respect to the a-set

   A-set Rigid Body Mass Matrix: $[D^T_{ar}][M_{aa}][D_{ar}]$. For a superelement this is computed at the g-set.

   Total Effective Mass Fraction: i.e., diagonal elements of the Modal Effective Mass Matrix divided by the Rigid Body Mass Matrix.

2. The PARTFAC describer outputs the Modal Participation Factors table: ϵ

3. The MEFFM describer outputs the Modal Effective Mass table: ϵ^2, the term-wise square of the Modal Participation Factors table.

4. The MEFFW describer outputs the Modal Effective Weight table; i.e., the Modal Effective mass multiplied by user PARAMeter WTMASS.

5. The FRACSUM describer outputs the Modal Effective Mass Fraction table; i.e., the Generalized Mass Matrix (diagonal term) multiplied by the Modal Effective Mass and then divided by the Rigid Body Mass Matrix (diagonal term).

6. For superelements the MEFFMASS command uses the residual structure eigenvalues and eigenvectors, by default. If however, PARAM, FIXEDB, -1 is specified then MEFFMASS command uses the component mode eigenvalues and eigenvectors.

7. Effective mass is computed in the basic coordinate system.

8. Modes which have an effective mass fraction greater than the value of THRESH in at least one translational or rotational direction are output in the .f06 and .op2 files. THRESH does not limit modes for consecutive dynamic response solutions (SOL 111 or 112). Use the MODSEL case control command to select/deselect modes in these solutions.

Selection of fluid modes

Prior to NX Nastran 7.1, the MODSEL case control command could only be used to select a subset of structural modes for use in a response solution. For acoustic or coupled acoustic/structural models, you could not exclude any of the fluid modes from the response solution.

The functionality of the MODSEL case control command has been expanded to include fluid modes. Now you can use the MODSEL case control command to select either structural or fluid modes to use in a response solution. If you include a MODSEL case control command for both structural modes and fluid modes in the same input file, subsets of both the structural and fluid modes can be used in the response solution. If you do not include a MODSEL case control command in the input file, all the structural and fluid modes are used in the response solution.

The MODSEL case control command is used in conjunction with the SET case control command to define which structural and fluid modes to include in the response solution. An example is provided below.

MODSEL example

```
SOL 111
...
...
...
$ Specify that SET 12 includes the structural modes to use in the response solution
$
MODSEL(STRUCTURAL)=12
$
$ Specify that SET 110 includes the fluid modes to use in the response solution
$
MODSEL(FLUID)=110
$
$ SET 12 includes the first two structural modes
$
SET 12 = 1,2
$
$ SET 110 includes the first 10 fluid modes
$
SET 110 = 1 THRU 10
...
...
...
BEGIN BULK
...
...
...
ENDDATA
```

Note

MODSEL case control commands are typically located outside of subcases. If you include a MODSEL case control command in a subcase, it is ignored unless the modes are recalculated for the subcase.

An example of a complete input file using the MODSEL case control command is *ac1102s1.dat*. It can be found in *install_dir/nxn7p1/nast/tpl*.

See the updated MODSEL case control command.

MODSEL Selects mode numbers

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

Format:

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

Examples:

```
MODSEL=3
MODSEL(STRUCTURAL)=4
MODSEL(FLUID)=5
```

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 either structural or fluid modes. The mode numbers included in the SET case control command are used in the response solution. The mode numbers not included in the SET case control command are removed from the modal space. Mode numbers larger than the number of eigenvalues computed are ignored. (Integer>0)

Remarks:

1. All structural and fluid modes are used in the response solution if a MODSEL entry is not included in the input file.
2. Multiple MODSEL entries can be included in a single input file.
3. The use of MODSEL in a subcase is only effective if the modes are recalculated for the subcase.

Acoustic/fluid-structure coupling matrices

In a coupled acoustic/fluid-structure analysis, the coupling between the fluid and structure for the stiffness and mass is computed by default. Prior to NX Nastran 7.1, you had to either use the computed coupling matrices or ignore the coupling by preventing the coupling matrices from being computed.

Beginning with NX Nastran 7.1, you can also do the following:

- Select direct input acoustic/fluid-structure coupling matrices to use in the analysis. These matrices can be used to represent the acoustic/fluid-structure coupling either independently or in conjunction with the computed acoustic/fluid-structure coupling matrices.

- Scale the direct input and/or the computed acoustic/fluid-structure coupling matrices.

You control the expanded functionality using a new case control command named A2GG, an enhanced ASCOUP parameter, and new parameters named CA1 and CA2.

When you specify both ASCOUP and A2GG, the software either adds the direct input acoustic/fluid-structure coupling matrices to the computed coupling matrix or the computed coupling matrix is ignored altogether in favor of direct input matrices. In either case, you can scale the coupling matrices in one of these ways:

- Specify one or both of the CA1 and CA2 parameters.

- Specify in-line scale factors using the A2GG case control command.

- Specify both in-line scale factors using the A2GG case control command and one or both of the CA1 and CA2 parameters.

If PARAM,ASCOUP,YES and A2GG are both specified, the total acoustic/fluid-structure coupling matrix is:

$$\left[A_{jj}\right] = CA1 \cdot \left[A^{x}_{jj}\right] + CA2 \cdot \left[A^{2}_{jj}\right]$$

where $[A^{x}_{jj}]$ is the computed acoustic/fluid-structure coupling matrix and $[A^{2}_{jj}]$ is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

If PARAM,ASCOUP,NO and A2GG are both specified, the computed coupling matrix is ignored and the total acoustic/fluid coupling matrix is:

$$\left[A_{jj}\right] = CA2 \cdot \left[A^{2}_{jj}\right]$$

where $[A^{2}_{jj}]$ is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

An example of a complete input file containing the A2GG case control command, the ASCOUP parameter, and the CA2 parameter is *ac11201a.dat*. It can be found in *install_dir*/*nxn7p1*/*nast*/*tpl*.

See the new A2GG case control command.

A2GG Direct Input Acoustic/Fluid-Structure Matrix Selection

Selects direct input acoustic/fluid-structure coupling matrix or matrices.

Format:

A2GG=name

Examples:

```
A2GG=ADMIG
A2GG=ADMIG1, ADMIG2, ADMIG3
A2GG=1.25*ADMIG1, 1.0*ADMIG2, 0.75*ADMIG3
SET 100=A1, A2
A2GG=100
```

Describers:

Describer	Meaning
name	Name of direct input acoustic/fluid-structure coupling matrix [A^2_{gg}] defined using DMIG bulk entries. (Character) Scale factors may be included (see Remarks 4 and 5).

Remarks:

1. DMIG matrices are not used unless selected using A2GG.
2. If PARAM,ASCOUP,YES, the direct input acoustic/fluid-structure coupling matrix selected using A2GG is added to the computed acoustic/fluid-structure coupling matrix. If PARAM,ASCOUP,NO, the direct input acoustic/fluid-structure coupling matrix selected using A2GG replaces the computed acoustic/fluid-structure coupling matrix.
3. The matrix must be square and field 4 of the DMIG bulk entry must contain the integer 1. When filling out the DMIG bulk entries, the GJ column index corresponds to fluid points, CJ is zero, the Gi row index corresponds to structural points, Ci corresponds to DOF, and Ai are the area values.
4. The associated DMIG matrices can be scaled using either in-line scale factors entered on A2GG (for example, A2GG=1.25*ADMIG1), or the parameter CA2 (for example, PARAM,CA2,1.25), or both. For information regarding the CA2 parameter, see “Parameter Descriptions”.
5. Multiple matrices separated by a comma or a blank are additive. When multiple matrices and in-line scale factors are used together, each matrix name in the list must include a scale factor. 1.0 should be entered for matrices in the list that are

not scaled. For example, if A2GG=1.25*ADMIG1,1.0*ADMIG2,0.75*ADMIG3 is specified, the result is A2GG=1.25*ADMIG1 + ADMIG2 + 0.75*ADMIG3.

Specifying the CA2 parameter scales all the A2GG. For example, if both PARAM,CA2,1.30 and A2GG=1.25*ADMIG1,1.0*ADMIG2,0.75*ADMIG3 are specified, the result is A2GG=1.30(1.25*ADMIG1 + ADMIG2 + 0.75*ADMIG3).

6. A2GG is supported in dynamic solutions with acoustic/fluid-structure coupling.

7. Only one A2GG case control command should be used in an input file and it should appear above any subcases.

ASCOUP parameter

The ASCOUP parameter is applicable to all solutions except SOL 601/701 and must be placed in the bulk entry section of the input file.

ASCOUP Default = YES

In coupled fluid-structure analysis, if PARAM,ASCOUP,YES is specified in or omitted from the input file, and the A2GG case control command is also omitted from the input file, coupling for the stiffness and mass is computed. If PARAM,ASCOUP,NO is specified and A2GG is omitted, the coupling is not computed.

When ASCOUP and A2GG are both specified, direct input acoustic/fluid-structure coupling matrices can be added to the computed coupling matrix or the computed coupling matrix can be ignored altogether in favor of direct input matrices. In either case, the coupling matrices can be scaled by:

- Specifying one or both of the CA1 and CA2 parameters.
- Specifying in-line scale factors using the A2GG case control command.
- Specifying both in-line scale factors using the A2GG case control command and one or both of the CA1 and CA2 parameters.

If PARAM,ASCOUP,YES and A2GG are both specified, the total acoustic/fluid-structure coupling matrix is:

$$\left[A_{jj}\right] = CA1 \cdot \left[A^{x}_{jj}\right] + CA2 \cdot \left[A^{2}_{jj}\right]$$

where $[A^{x}_{jj}]$ is the computed acoustic/fluid-structure coupling matrix and $[A^{2}_{jj}]$ is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

If PARAM,ASCOUP,NO and A2GG are both specified, the computed coupling matrix is ignored and the total acoustic/fluid coupling matrix is:

$$\left[A_{jj}\right] = CA2 \cdot \left[A_{jj}^{2}\right]$$

where [A^2_{jj}] is the direct input acoustic/fluid-structure coupling matrix specified using A2GG.

CA1, CA2 parameters

The CA1, CA2 parameter is applicable to all solutions except SOL 601/701 and can be placed in either the bulk entry or case control sections of the input file.

CA1, CA2 Default = (1.0, 0.0)

CA1 and CA2 specify factors for scaling the total acoustic/fluid-structure coupling matrix. The total acoustic/fluid-structure coupling matrix is:

$$\left[A_{jj}\right] = CA1 \cdot \left[A_{jj}^{x}\right] + CA2 \cdot \left[A_{jj}^{2}\right]$$

where [A^x_{jj}] is the computed acoustic/fluid-structure coupling matrix and [A^2_{jj}] is the direct input acoustic/fluid-structure coupling matrix specified using the A2GG case control command. CA1 and CA2 are only effective if A2GG is specified in the case control section.

Defining panels

Panels are collections of structural grid points, and are used to evaluate the transfer path of structural vibration into dynamic acoustic pressure. Panels are defined using the PANEL bulk entry. Prior to NX Nastran 7.1, PANEL bulk entries referenced only SET1 bulk entries. SET1 bulk entries list structural grid points. Now PANEL bulk entries can also reference the new SET3 bulk entries. The SET3 bulk entry can list identification numbers for structural grid points, elements, or physical properties.

- When a PANEL bulk entry references SET3 bulk entries that have the GRID field, the panels will consist of the structural grid points listed on the SET3 bulk entries.

- When a PANEL bulk entry references SET3 bulk entries that have the ELEM field, the panels will consist of the grid points that are connection points for the structural elements listed on the SET3 bulk entries.

- When a PANEL bulk entry references SET3 bulk entries that have the PROP field, the panels will consist of the grid points that are connection points for the structural elements that reference the physical properties listed on the SET3 bulk entries.

An example of a complete input file using the SET3 bulk entry with the GRID field is *ac11102.dat*. An example of a complete input file using the SET3 bulk entry with the ELEM field is *ac11102el.dat*. An example of a complete input file using the SET3 bulk entry with the PROP field is *ac11102pr.dat*. All three files can be found in *install_dir/nxn7p1/nast/tpl*.

See the updated PANEL bulk entry.

See the updated SET1 bulk entry.

See the new SET3 bulk entry.

PANEL Panel Definition for Coupled Fluid-Structural Analysis

Selects sets of structural grid points, elements, or physical properties that define one or more panels.

Format:

1	2	3	4	5	6	7	8	9	10
PANEL	NAME1	SID1	NAME2	SID2	NAME3	SID3	NAME4	SID4	

Example:

PANEL	BKDOOR	103							

Fields:

Field	Contents
NAMEi	Panel label. (Character)
SIDi	Identification number of a SET1 or SET3 bulk entry that lists the structural grid points, elements, or physical properties of the panel. (Integer > 0)

Remarks:

1. Panels are groups of structural grid points.
   - If a set of grid points is referenced, the set must include only structural grid points. The panel will consist of all the grid points in the referenced set. SET1 and SET3 bulk entries are used to define sets of grid points.
   - If a set of elements is referenced, the set must include only structural elements. The panel will consist of all the grid points that are connection points for these elements. SET3 bulk entries are used to define sets of elements.
   - If a set of physical property identifiers is referenced, the physical properties must be referenced by structural elements. The panel will consist of all grid points that are connection points for the structural elements referencing the physical properties included in the set. SET3 bulk entries are used to define sets of physical property identifiers.
2. If the referenced SET1 or SET3 bulk entries include structural grid points, the sets must include at least four grid points for quadrilateral faces and three grid points for triangular faces.

3. It is recommended that all of the connection points for a given element belong to the same panel.

4. NAMEi is used only for labeling the output of the panel modal participation factors (refer to the MODCON and PANCON case control commands). See “Performing a Coupled Fluid-Structural Analysis” in the *NX Nastran User’s Guide*.

SET1 Set Definition

Defines a list of structural grid points for aerodynamic analysis, for XY-plots with SORT1 output, and for the PANEL bulk data entry. Also defines a list of DRESPi (i=1,2,3) response IDs for the P2RSET option on the DOPTPRM bulk entry.

Grid ID Format:

1	2	3	4	5	6	7	8	9	10
SET1	SID	G1	G2	G3	G4	G5	G6	G7	
	G8	-etc.-							

Response ID Format:

1	2	3	4	5	6	7	8	9	10
SET1	SID	R1	R2	R3	R4	R5	R6	R7	
	R8	-etc.-							

Example:

SET1	3	31	62	93	124	16	17	18	
	19								

Alternate Formats and Example:

SET1	SID	G1	"THRU"	G2					
– or –									
SET1	SID	R1	"THRU"	R2					
SET1	6	32	THRU	50					

Fields:

Field	Contents
SID	Unique identification number. (Integer > 0)
Gi	List of structural grid point identification numbers. (Integer > 0 or "THRU"; for the "THRU" option, G1 < G2.)
Ri	List of DRESPi (i=1,2,3) response IDs for the P2RSET option on the DOPTPRM bulk entry. (Integer > 0 or "THRU"; for the "THRU" option, R1 < R2.)

Remarks:

1. SET1 entries may be referenced by the SPLINEi entries, PANEL entries, XYOUTPUT, and the P2RSET option on the DOPTPRM bulk entry.

2. When using the “THRU” option for SPLINEi or PANEL data entries, all intermediate grid points must exist.

3. When using the “THRU” option for XYOUTPUT requests, missing grid points are ignored.

4. When using the “THRU” option for DRESPi requests, missing response IDs are ignored.

5. The SID must be unique from other SET1 and SET3 SIDs.

SET3 Set Definition

Defines a list of structural grid points, elements, or physical properties.

Grid ID Format:

1	2	3	4	5	6	7	8	9	10
SET3	SID	TYPE	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	-etc.-							

Example:

SET3	1	GRID	11	13	14	15	20	22	
	34	41							

Alternate Formats and Example:

SET3	SID	TYPE	ID1	"THRU"	ID2				
SET3	2	ELEM	20	THRU	33				

Fields:

Field	Contents
SID	Unique identification number. (Integer > 0)
TYPE	Set type. (Character: "GRID", "ELEM", "PROP")
IDi	Identifying numbers of structural grids, elements, or physical properties. (Integer > 0)

Remarks:

1. The SID must be unique from other SET1 and SET3 SIDs.

2. By specifying the GRID field, the PANEL bulk entry referencing the SET3 bulk entry interprets the IDi as structural grid point IDs.

3. By specifying the ELEM field, the PANEL bulk entry referencing the SET3 bulk entry interprets the IDi as structural element IDs.

4. By specifying the PROP field, the PANEL bulk entry referencing the SET3 bulk entry interprets the IDi as physical property IDs for structural elements.

Modal and panel contributions

Prior to NX Nastran 7.1, you could request output of modal contributions to structural response. Now the modal contributions capability has been expanded to support acoustic responses. In addition, the current release includes panel participation capability that allows you to examine acoustic response contributions by panel.

- Use the enhanced MODCON case control command to request structural mode and fluid mode contributions using the TOPS and TOPF describers, respectively.

- Use the new PANCON case control command to request structural panel and structural grid contributions using the TOPP and TOPG describers, respectively. Use the PANEL and GRID describers to specify the panels and grids to output panel contributions.

The PANCON capabilities are only supported for SOL 108 and 111.

An example of a complete input file containing the MODCON and PANCON case control commands is *ac11104.dat*. It can be found in *install_dir/nxn7p1/nast/tpl*. In the example, the MODCON case control command is used with the TOP describer specified. The TOP describer is a legacy describer for the MODCON case control command and is equivalent to and can be used interchangeably with the new TOPS describer. However, the use of the TOPS describer is recommended for clarity.

See the updated MODCON case control command.

See the new PANCON case control command.

See the updated SET case control command.

Modal and panel contributions mathematical basis

The structural modal contribution from structural modes is given by:

$$[P_{ss}] = [\Phi_s][\xi_s]$$

where [Φ_s] are the uncoupled, undamped structural modes and [x_s] are the structural modal amplitudes.

The fluid modal contribution from fluid modes is given by:

$$[P_{ff}] = [\Phi_f][\xi_f]$$

where [Φ_f] are the uncoupled, undamped, rigid-wall acoustic modes and [x_f] are the fluid modal amplitudes.

The fluid modal participation from structural modes is given by:

$$[P_{fs}] = -\omega^2 [\Phi_f][Z_2][a]^T [\xi_s]$$

where W is the excitation frequency in rad/sec. The matrix [Z_2] is given by:

$$[Z_2] = \left[-\omega^2 [m_f] + i\omega [b_f] + [k_f]\right]^{-1}$$

where [m_f] is the fluid modal mass, [b_f] is the fluid modal damping, and [k_f] is the fluid modal stiffness.

The matrix [a] is a modal representation of the acoustic coupling matrix given by:

$$[a] = [\Phi_f]^T [A][\Phi_f]$$

where [A] is the acoustic coupling matrix.

The fluid-structure panel participation is given by:

$$[P_p] = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T [A]^T_{panel} [\Phi_s][\xi_s]$$

where [A]$_{panel}$ is a reduced form of the acoustic coupling matrix for specific panels.

The fluid-structure panel-grid participation is given by:

$$[P_g] = -\omega^2 [\Phi_f][Z_2][\Phi_f]^T [\{A_b^T\}][\lfloor \Phi_s \rfloor][\xi_s]$$

Matrix [{ $A^T{}_b$ }] is formed from columns extracted from the bth boundary panel for panel grid i. Matrix [$\lfloor \Phi_s \rfloor$] is formed from rows of the structural modal matrix corresponding to panel grid i.

MODCON Modal Contribution Request

Requests the output of modal contribution results.

Format:

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

$$\begin{bmatrix}\mathrm{ABS}\\ \mathrm{NORM}\\ \mathrm{BOTH}\end{bmatrix},\mathrm{TOPS}=\mathrm{ps},\mathrm{TOPF}=\mathrm{pf},\mathrm{SOLUTION}=\left.\begin{Bmatrix}\mathrm{ALL}\\ \mathrm{setout}\end{Bmatrix}\right]=\begin{Bmatrix}\mathrm{n}\\ \mathrm{ALL}\\ \mathrm{NONE}\end{Bmatrix}$$

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.
NOPRINT	Generates, but does not print, modal contribution results.
ABS	Output modal contributions in absolute terms. (Default)
NORM	Output modal contributions in normalized terms.

Describer	Meaning
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*.
n	Calculate modal contributions for the list defined in SETMC n.
ALL	Calculate modal contributions for the lists defined in all SETMC sets defined in and above the current subcase.
NONE	Do not calculate modal contributions. This is useful to turn off modal contribution output for a specific subcase.

Remarks:

1. Both PRINT and PUNCH may be requested.
2. MODCON = NONE overrides an overall output request.
3. SOL 110, 111, 112, and 146 are supported. For SOL 110, modal contributions for superelements are not supported. The TOPF keyword is 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 keyword can be abbreviated to SOLU.

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.

Format:

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

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

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

Examples:

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

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of panels or grids for each frequency. (Default)
SORT2	Output will be presented as a tabular listing of frequency for each panel or grid.
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 in the bulk entry section 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 = NONE (default) requests that no structural grid contributions be output. GRID = *setg* requests contributions from structural grids included in the SET case control command having the identification number of *setg*. GRID = ALL requests contributions from all structural grids that are part of the acoustic coupling matrix.
n	Calculate panel and/or grid contributions for the list defined in SETMC n. Any response defined in SETMC n that is not an acoustic response will be ignored.

Describer	Meaning
ALL	Calculate panel and/or grid contributions for the lists defined in all SETMC sets defined in and above the current subcase. Any response defined in SETMC sets that is not an acoustic response will be ignored.
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 or times. 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.

SET Set Definition, General Form

Sets are used to define the following lists:

Lists

1. Identification numbers (point, element, or superelement) for processing and output requests.

2. Times for which output will be printed in transient response problems using the OTIME case control command or the MODCON case control command with the SOLUTION = *setout* describer specified.

3. Frequencies for which output will be printed in frequency response problems using the OFREQ case control command or the MODCON case control command with the SOLUTION = *setout* describer specified or the PANCON case control command with the SOLUTION = *setf* describer specified.

4. Panels for which output will be printed in frequency response problems using the PANCON case control command with the PANEL describer specified.

5. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD case control commands.

6. With SOL 200, DRESP1 design responses that are assigned to a specific subcase via a DRSPAN case control command which refers to a particular SET.

Formats:

SET n = {i_1[,i_2, i_3, THRU i_4, EXCEPT i_5, i_6, i_7, i_8, THRU i_9]}

SET n = {r_1, [r_2, r_3, r_4]}

SET n = {$name_1$,[$name_2$,$name_3$,$name_4$]}

SET n = ALL

Examples:

```
SET 77=5
SET 88=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU 300
SET 99=1 THRU 100000
SET 101=1.0, 2.0, 3.0
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
SET 5=PANEL1,PANEL3,PANEL4
```

Describers:

Describer	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only. (Integer>0)
i_1, i_2, etc.	Identification numbers. If no such identification number exists, the request is ignored. (Integer $\geq$ 0)
i_3 THRU i_4	Identification numbers (i_4>i_3). (Integer>0)
EXCEPT	Set identification numbers following EXCEPT will be deleted from the output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.
r_1, r_2, etc.	Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ, OTIME, MODCON (with SOLUTION = *setout* describer specified), or PANCON (with SOLUTION = *setf* describer specified) case control command references the set, then the values must be listed in ascending sequences, r_1>r_2>r_3>r_4...etc., otherwise some output may be missing. (Real>0.0)
$name_1$, $name_2$, etc.	Alphanumeric names of panels defined by PANEL bulk entries. (Character)
ALL	All members of the set will be processed. This option may not be used in a DRSPAN referenced set.

Remarks:

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.

2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.

3. In SET 88 in the example section above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.

Support for damping element forces

Prior to NX Nastran 7.1, force output on damping elements was not available for transient response analysis. Force output on damping elements was available for modal frequency response analysis, but you were required to select the mode displacement method (PARAM,DDRMM,-1).

Force output on damping elements is now available for:

- Transient response analysis.
- Modal frequency response analysis without having to select the mode displacement method.

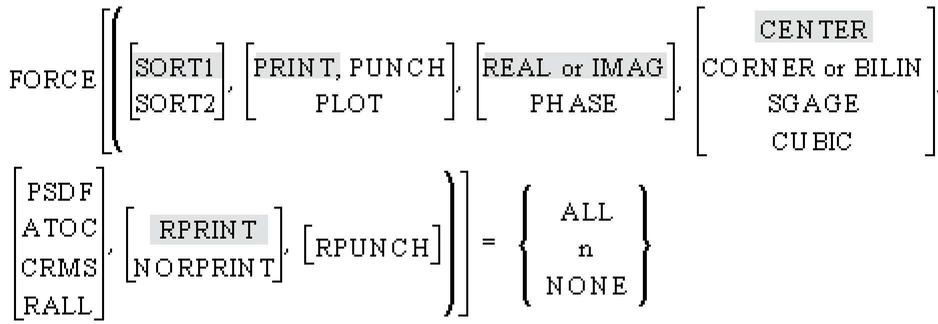
For both of these analysis types, you can output the forces on damping elements by simply including the damping elements in the force request.

See the updated FORCE case control command.

FORCE — Element Force Output or Particle Velocity Request

Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.

Format:



Examples:

```
FORCE=ALL
FORCE(REAL, PUNCH, PRINT)=17
FORCE=25
```

Describers:

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PLOT	Generates force output for requested set but no printed output.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 8.
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 8.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be made in the case control. See Remark 8.
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 8.
RPRINT	Writes random analysis results to the print file. (Default) See Remark 8.
NORPRINT	Disables the writing of random analysis results to the print file. See Remark 8.
RPUNCH	Writes random analysis results to the punch file. See Remark 8.
CENTER	Output CQUAD4, CQUADR, CTRIAR element forces at the center only.
CORNER or BILIN	Output CQUAD4, CQUADR, CTRIAR element forces at the center and grid points using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element forces at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element forces at center and grid points using cubic bending correction.
ALL	Forces for all elements will be output.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only forces of elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	Forces for no elements will be output.

Remarks:

1. ALL should not be used in a transient problem.

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

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

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

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

3. ELFORCE is an alternate form and is entirely equivalent to FORCE.

4. FORCE=NONE overrides an overall request.

5. For composite stress and/or failure index output, a FORCE request is required for the desired elements.

6. In nonlinear transient analysis, this request is ignored for nonlinear elements.

7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. Consequently, options specified in subcases other than the first subcase will be ignored.

- If the STRESS command is specified in the first subcase then the option on the STRESS command is used in all subcases with STRESS, STRAIN, and FORCE commands.

- If the STRAIN command and no STRESS command is specified in the first subcase, then the option on the STRAIN command is used in all subcases containing STRESS, STRAIN, and FORCE commands.

- If the FORCE command and no STRESS or STRAIN command is specified in the first subcase, then the option on the FORCE command is used in all subcases containing STRESS, STRAIN, and FORCE commands.

- If STRESS, STRAIN, and FORCE commands are not specified in the first subcase, then the CENTER option is used in all subcases containing STRESS, STRAIN, and FORCE commands.

8. 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.

Beta capability: Initial conditions for modal 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 the initial condition for both types of analysis.

For direct transient analysis (SOL 109) using the IC case control command:

- If you specify the PHYSICAL (default) describer, TIC bulk entries that contain the initial condition in physical space are referenced.

- If you specify the STATSUB describer, the results of a static subcase are used as the initial condition.

- If you specify the DIFFK describer in conjunction with the STATSUB describer, the results of a static subcase are used as the initial condition and the effects of differential stiffness are accounted for during the analysis.

For modal transient analysis (SOL 112) using the IC case control command:

- If you specify the PHYSICAL (default) describer, TIC bulk entries that contain the initial condition in physical space are referenced. The initial conditions in physical space are converted to modal space during the solve.

- If you specify the MODAL describer, TIC bulk entries that contain the initial condition in modal space are referenced.

- If you specify the STATSUB describer, the results of a static subcase are converted to modal space during the solve and used as the initial condition.

- If you specify the DIFFK describer in conjunction with the STATSUB describer, the results of a static subcase are converted to modal space during the solve and used as the initial condition. The effects of differential stiffness are accounted for during the analysis.

Beginning with the NX Nastran 7.1 release, as a beta capability, you can optionally use the new TZERO describer with the IC case control command. When you specify the TZERO describer, the static deflection in modal space resulting from the loading at time = 0 is used as the initial condition for a modal transient analysis (SOL 112).

> **Note**
>
> At present, there is no way to account for the effects of differential stiffness in a modal transient analysis (SOL 112) when you specify the TZERO describer.

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/nxn7p1/nast/tpl*.

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:

$$\mathrm{IC}\left[\begin{array}{c} \mathrm{PHYSICAL} \\ \mathrm{MODAL} \\ \mathrm{STATSUB}[,\mathrm{DIFFK}] \\ \mathrm{TZERO} \end{array}\right] = n$$

Examples:

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

Describers:

Describer	Meaning
PHYSICAL	The TIC bulk entries selected by set *n* define initial conditions for coordinates involving grid and scalar points (default). See Remark 6.
MODAL	The TIC bulk entries selected by set *n* define initial conditions for modal coordinates. See Remarks 3 and 6.
STATSUB	Use the solution of the static analysis subcase *n* as the initial conditions. See Remark 4.
DIFFK	Include the effects of differential stiffness in the solution. See Remarks 4 and 5.
TZERO	Use the static deflection resulting from the loading at time = 0 as the initial condition for a modal transient analysis (SOL 112). For this option, *n* is not needed. If *n* is specified, it will be ignored.

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). (Integer>0)

Remarks:

1. For structural analysis, TIC bulk entries will not be used 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) may not both be specified in the same execution. They are only applicable to direct transient analysis (SOL 109) and modal transient analysis (SOL 112), but not in a DMP solution.

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

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

7. IC(TZERO) may be specified only in modal transient analysis (SOL112).

8. The IC case control command is not supported in multiple subcases.

Sparse data recovery

An improved sparse data recovery option became available in NX Nastran 5 for modal frequency response analysis (SOL 111), modal transient response analysis (SOL 112), and optimization (SOL 200; ANALYSIS=MFREQ or MTRAN).

This sparse data recovery option is now also supported for direct frequency response (SOL 108), and is used by default. This change can significantly reduce your run times when large amounts of data are recovered.

The sparse data recovery option is on by default, but you can deactivate it with system cell 421:

NASTRAN SYSTEM(421) = 0

or by using the keyword SPARSEDR:

NASTRAN SPARSEDR = 0.

Unsymmetric A-set reduction for rotor dynamics

Rotor dynamic effects produce unsymmetric stiffness matrices. NX Nastran can now perform an A-set reduction of these matrices to increase solution efficiency. This method reduces problem sizes for dynamic analysis by extracting modes from the O-set, and preserving the degrees of freedom in the A-set. It is efficient for SOL 107 jobs with rotor dynamics.

Inputs

You must specify the desired A-set degrees of freedom with ASET or ASET1 bulk entries. For best accuracy and performance, select the A-set carefully. For a solid rotor example, it is sufficient if you specify points on the center line as A-set grid points; for turbines, you must also specify the grid points on the blades. For other structures, make a reasonable choice.

Remark

You must specify the EIGRL bulk entry in addition to the EIGC bulk entry to extract modes from the O-set. An appropriate EIGRL setting is problem dependent; more modes improve accuracy, but also increase run time.

Example

- Turbine model
- 45,350 grid points, 272,100 degrees of freedom
- Machine: 1.9 GHz Xeon
- 2 GB memory
- 10 steps in ROTORD

Method	Direct Sol 107	Unsym A-set Sol 107
Elapsed (min:sec)	3636:47	131:10
CPU (seconds)	139897	6784
Disk IO (GB)	2575	1120

Chapter

2 Superelements

External superelements

A more efficient external superelement procedure was introduced with NX Nastran 6 and enhanced in subsequent releases. The capabilities of external superelements are further enhanced in this release.

Prior to NX Nastran 7.1, the assembly solution required you to include external superelement output data in the assembly input file for each external superelement. It could be obtained as punch output in the external superelement solution by requesting EXTBULK output on the EXTSEOUT case control.

This data however is redundant because it is also available in the external superelement files containing the matrix data. Therefore, beginning in NX Nastran 7.1, the external superelement data is not required in the assembly input file and will be retrieved from the external superelement matrix files if not present in the input file.

EXTSEOUT External Superelement Creation Specification

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

Format:

```
EXTSEOUT [ ( [ STIFFNESS, MASS, DAMPING, K4DAMP, LOADS, ASMBULK, EXTBULK,

EXTID = seid, DMIGSFIX = { cccccc } ] [      MATDB(orMATRIXDB)      ] ) ]
                          { EXTID  }   [          DMIGDB             ]
                                       [       DMIGOP2=unit          ]
                                       [          DMIGPCH            ]
                                       [ MATOP4(orMATRIXOP4)=unit    ]
```

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)
```

See Remarks 10, 11, and 12.

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 boundary static loads matrix. See Remarks 1 and 2.

Describer	Meaning
ASMBULK	Generate bulk data 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 data portion of the subsequent assembly solution. See Remarks 4 and 13.
EXTBULK	Generate and store bulk data 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 data 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 data entries stored on the assembly punch file (.asm) if ASMBULK is specified and in the BEGIN SUPER bulk data 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 – 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 – 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 data entries on the database.
DMIGOP2=*unit*	Store the boundary matrices as DMIG bulk data 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 data entries on the standard punch file (.pch). See Remarks 6 – 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, 8, and 10.

Remarks:

1. If none of the describers STIFFNESS, MASS, DAMPING, K4DAMP, and LOADS are specified, then all matrices 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 data 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 data 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 data 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 entry.

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

    - If the creation involves component mode reduction, the required generalized coordinates are specified using QSET/QSET1 bulk data entries. The boundary data for the component mode reduction may be specified using the BNDFIX/BNDFIX1 and BNDFREE/BNDFREE1 bulk data entries (or their equivalent BSET/BSET1 and CSET/CSET1 bulk data 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 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 Data entries, then the following points should be noted:

        - PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the external superelement*.

        - PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for an external superelement apply *only to the superelement*.

- 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 Data. 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 exit, they will be ignored. The existence of the EXTSEOUT case control entry 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. Superelement results can be recovered in the second step (i.e. superelement assembly, analysis, and data recovery) for SOLs 101, 103, 105-112, 129, 144-146, 153, and 159.

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.).

Chapter 3 Advanced nonlinear

Advanced nonlinear

Element enhancements

- Four new plane stress elements CPLSTS3, CPLSTS4, CPLSTS6 and CPLSTS8 and four new plane strain elements CPLSTN3, CPLSTN4, CPLSTN6 and CPLSTN8 are available for SOL 601. The new PPLANE bulk entry defines the properties for the eight new elements.

  The examples of complete input files using the new plane stress elements are *anlplsts*.dat*. The examples of complete input files using the new plane strain elements are *anlplstn*.dat*. All of these files can be found in *install_dir/nxn7p1/nast/tpl/*.

  See chapter 2 of the *NX Nastran 7.1 Advanced Nonlinear Theory and Modeling Guide* and the *NX Nastran 7.1 Quick Reference Guide* for input descriptions.

Contact and glue enhancements

- The 2D edge-to-edge contact inputs used with axisymmetric elements are supported with the new plane stress and plane strain elements. You can use the existing BLSEG bulk entry for the 2D contact region definition. The 2D contact definition uses the bulk entries BCRPARA, BCTSET, BCTPARA, which are also used to define 3D contact. All three of these element types (axisymmetric, plane stress, plane strain) including 2D contact conditions can exist in the same model.

  An example of a complete input file using the new plane stress elements with contact conditions is *anlplsts07.dat*. An example of a complete input file using the new plane strain elements with contact conditions is *anlplstn11.dat*. Both of these files can be found in *install_dir/nxn7p1/nast/tpl/*.

- You can use the new BCPROPS bulk entry in SOL 601 to create solid element contact/glue regions by property ID. The BCPROPS bulk entry works similar to the BCPROP bulk entry that is used to create shell element regions by property ID. Only solid element free faces are considered in the contact and glue algorithms. The free faces are automatically determined by the software.

- Beginning in version 7.0, variable grid CTETRA and CPYRAM elements became available for SOL 601. They now can be included in contact or glue regions.

  Mixed order CPYRAM and CTETRA elements are useful to transition from a linear CHEXA mesh to a parabolic CTETRA mesh.

See the *NX Nastran 7.1 Advanced Nonlinear Theory and Modeling Guide* and the *NX Nastran 7.1 Quick Reference Guide* for input descriptions.

Results recovery restart

You can use the new SOL 601 restart option to recover results that you did not request in the original solution. Use the new MODEX parameter option “2” on the NXSTRAT bulk entry to request the results recovery solution.

See section 10.2 in the *NX Nastran 7.1 Advanced Nonlinear Theory and Modeling Guide* and the NXSTRAT bulk entry in the *NX Nastran 7.1 Quick Reference Guide*.

PUNCH output

SOL 601 now supports PUNCH output requests. The grid and element sets are also supported by the punch output request.

Chapter 4 Contact for linear solutions

Solid element property regions

The new BCPROPS bulk entry is available to create solid element contact and glue regions by property ID (PSOLID bulk entry ID). The BCPROPS entry works similar to the supported BCPROP bulk entry. The BCPROP bulk entry is used to create shell element contact and glue regions by property ID (PSHELL bulk entry ID). Only the solid element free faces are considered in the contact and/or glue algorithm. The free faces of the solid elements selected with a property ID are automatically determined by the software.

See the BCPROPS bulk entry.

BCPROPS Contact or Glue Region Definition by PSOLID Property ID

Defines a surface-to-surface contact or glue region by PSOLID Property ID.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROPS	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	IP10	-etc-					

Example:

BCPROPS	3	1	5	8	3	22			

Fields:

Field	Contents
ID	Identification number of a contact or glue region. See Remark 2 and 3. (Integer > 0)
IPi	PSOLID Property IDs. (Integer > 0)

Remarks:

1. The continuation field is optional.
2. BCPROPS is a collection of one or more solid property IDs. BCPROPS defines a contact or glue region formed by the free faces of the solid elements and may act as a source or target.
3. The ID must be unique with respect to all other BLSEG, BSURFS, BSURF, BCPROP, and BCPROPS entries.
4. Only the solid element free faces within the given set of PIDs are considered in the contact and/or glue algorithm. These free faces are automatically determined by the software.

Contact conditions with linear buckling

Contact conditions can be included in a linear buckling solution (SOL 105), although there are important considerations. The initial linear statics solution determines the differential stiffness (Ks) and the final contact stiffness (Kcontact). The final contact stiffness is added to the stiffness matrix for the buckling subcase:

$$([K + Kcontact] + l\ [Ks])\{D\}=0$$

Normally the buckling load is determined by multiplying l from the first buckling mode by the applied load. When contact conditions are included, this is valid only when the first l is close to "1.0". Although contact conditions can be included in linear solutions, they are iterative and nonlinear conditions. Because contact conditions are nonlinear, the contact stiffness from the initial statics solution is nonlinearly dependent on the applied load. As a result, the contact stiffness result which was valid for the original loading, may not be valid for the scaled load condition.

The buckling solution will report a warning if the lowest l is not within 10% of 1.0 (0.9 - 1.1). When the warning is issued, you can scale your load by l , and then rerun both the linear statics solution with contact conditions, and the buckling solution again. You will need to repeat this process until l is close 1.0.

The inputs for a linear buckling solution with contact conditions require a subcase for the linear statics subcase and the buckling solution.

In addition, the linear statics subcase must include the BCSET case control command. If the linear statics subcase is the first subcase, then a STATSUB bulk entry is not needed. If it is not the first subcase, a STATSUB bulk entry is needed in the buckling subcase to reference the linear statics subcase ID.

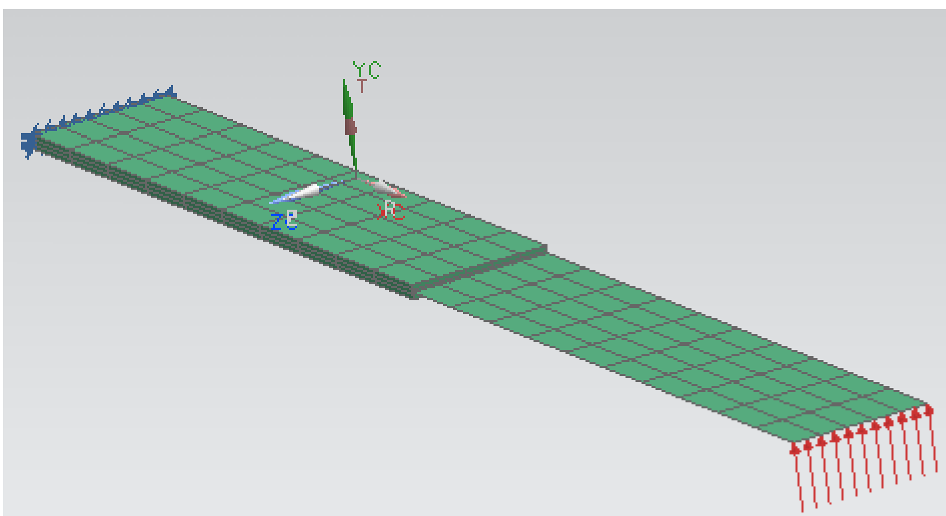
Chapter

5 Glue enhancements

Edge-to-surface glue

An edge-to-surface glue capability is now available. You can glue the edges of shell elements to the faces of a solid or shell elements.

- The existing BLSEG bulk entry defines the shell element edge regions.
- You can use any of the face region inputs to define the solid and/or shell face regions. Face region inputs include the BSURF, BSURFS, BCPROP, and BCPROPS bulk entries.
- The BGSET bulk entry defines the connection between the edge region ID and the face region ID.
- A weld like formulation is used to make the connection (the software always uses the GLUETYPE=2 option on the BGPARM bulk entry).



Edge-to-surface gluing remarks:

- A glue edge region defined with the BLSEG entry consists of one or more line segments defined between consecutive grid points. You must enter the grid points that define the edge region in a continuous topological order on the BLSEG entry. If an edge region or curve forms a closed loop, for example, the grid points around the perimeter of a cylinder edge, the last grid point identification number should be the same as the first grid point number.

- The grid point IDs on the BLSEG entry used to define a glue edge region can only be part of the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR and CTRIA6 element connectivity.

- On the BGSET bulk entry, you must enter the BLSEG ID as the source region ID, and a shell or solid element face region ID as the target region ID.

- The grid points on glued edges and surfaces do not need to be coincident.

- Edge-to-surface glue definitions are supported in all solution sequences except solutions 144 -146, 153, 159, 601 and 701. Edge-to-surface glue definitions cannot be used to represent acoustic glue connections.

- Refinement is ignored for edge-to-surface glue pairs.

- Shell offsets are not accounted for in any glue definitions. The gluing occurs at the grid locations and not at the offset location.

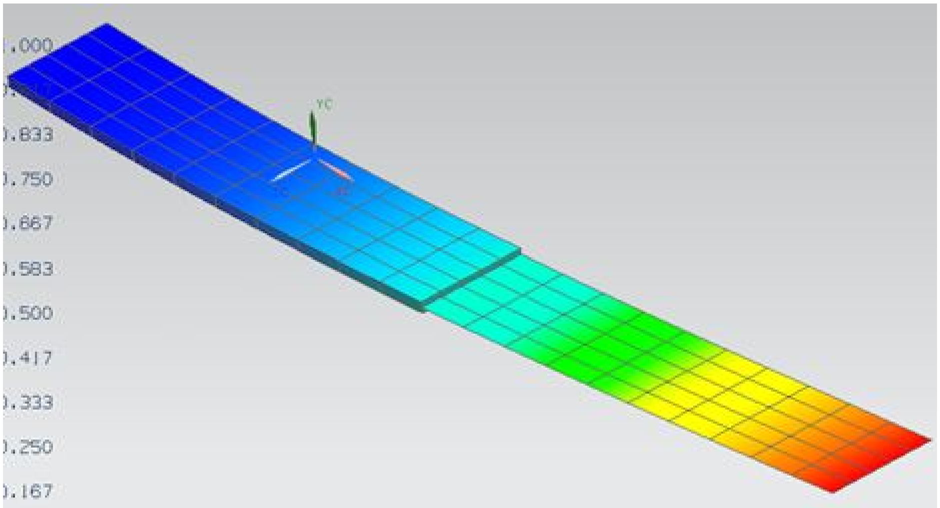
- Only the PENGLUE and PENTYP parameters on the BGPARM bulk entry are applicable to the edge-to-surface glue. All other parameters are ignored.

Input example:

```
...
Case Control

BGSET = 114
...
Bulk Data
...
$Grid points entered on a BLSEG entry define the edge region
BLSEG          2    1534    1699    1535    1697    1536    1695    1537+
+           1693    1538    1690    1539


$BSURFS defines the surface region in this example
BSURFS         1                            220     442     543     620+
+            210     441     620     619     200     440     619     477+
+            380     614     593     594     370     613     594     473
...
$The BLSEG ID must be entered as the source region on the BGSET entry
BGSET        114       2       1.0100504
```



The input file *gluedgsurf01.dat* is included in the *installation_path\nxn7p1\nast\tpl* directory to demonstrate the enhancement.

BLSEG Defines a glue or contact edge region, or a curve for slideline contact.

Defines a glue or contact edge region or a curve for slideline contact via grid numbers.

Format 1: (Formats 1 and 2 cannot be combined on the same line)

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	

Format 2:

BLSEG	ID	G1	"THRU"	G2	"BY"	INC			

Continuation Format 1: (Continuation formats 1 and 2 cannot be combined on a single continuation line)

	G8	G9	G10	G11	–etc.–				

Continuation Format 2:

	G8	"THRU"	G9	"BY"	INC				

Example:

BLSEG	14	101	THRU	190	BY	5			
	46	23	57	82	9	16			
	201	THRU	255						
	93	94	95	97					

Fields:

Field	Contents
ID	Edge region or curve identification number. See Remark 1 (Integer > 0).
Gi	Grid point identification numbers in a continuous topological order. See Remark 2 (Integer > 0).
INC	Grid point identification number increment. See Remark 3 for default (Integer or blank).

Remarks:

1. The ID must be unique with respect to all other BLSEG entries and all surface region IDs defined with the BSURF, BSURFS, BCPROP, and BCPROPS entries.

2. An edge region or curve consists of one or more line segments defined between consecutive grid points. The grid points defining the edge region or curve must be entered in a continuous topological order. If an edge region or curve forms a closed loop, for example, the grids around the perimeter of a cylinder edge, the last grid point identification number should be the same as the first grid point number.

3. When selecting grid points in a range using “THRU”, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing.

4. With edge-to-surface gluing:

   - The BLSEG entry is used to define an edge region that can be glued to the face of solid or shell elements. The BLSEG ID must be entered as the source region ID on the BGSET bulk entry, and a shell or solid element face region ID (BSURF, BSURFS, BCPROP, and BCPROPS) is entered as the target region ID.

   - The grid point IDs on the BLSEG entry used to define a glue edge region can only be part of the CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR and CTRIA6 element connectivity.

   - The BWIDTH bulk entry is ignored.

5. With slideline contact:

   - A corresponding BWlDTH Bulk Data entry may be required to define the width/thickness of each line segment. If the corresponding BWlDTH is not present, the width/thickness for each line segment is assumed to be unity.

   - Each line segment has a width in a 3-D slideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width/thickness of each line segment is defined via the BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWIDTH. That is, there must be a one to one correspondence between BLSEG and BWIDTH. BWIDTH Bulk Data entry may be omitted only if the width/thickness of each segment is unity.

   - The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the ‘ClD’ field of the BCONP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.

Remarks related to SOL 601 edge contact:

1. BLSEG defines a flexible or rigid 2D contact region on axisymmetric elements CQUADX4, CQUADX8, CTRAX3 and CTRAX6, plane stress elements CPLSTS3, CPLSTS4, CPLSTS6 and CPLSTS8, plane strain elements CPLSTN3, CPLSTN4, CPLSTN6 and CPLSTN8, or a rigid 2D contact target region when the grid points are not attached to any elements.

2. The grid points in a BLSEG entry must either be all attached to elements or all not attached to elements.

3. For a rigid target region, it is important to note that the top surface is on the left side of the line from Gi to Gi+1. By default, contact is expected to occur from the top surface. SURF='BOT' in BCRPARA entry may be used to change the contact side.

4. Grid points in BLSEG entry must lie in the basic XZ plane.

5. The BWIDTH bulk entry is ignored.

6. Contact set pairs are defined by BCTSET entry instead of BCONP entry.

7. Contact region properties are defined by BCRPARA entry and contact set properties are defined by BCTPARA entry in a similar way as for 3-D contact. In addition, global contact settings may be specified in the NXSTRAT entry.

Solid element property regions

The new BCPROPS bulk entry is available to create solid element contact and glue regions by property ID (PSOLID bulk entry ID). The BCPROPS entry works similar to the supported BCPROP bulk entry which is used to create shell element contact and glue regions by property ID (PSHELL bulk entry ID). Only the solid element free faces are considered in the contact and glue algorithm. The free faces are automatically determined by the software.

See the BCPROPS bulk entry.

Chapter 6 Bolt Preload

Combining and scaling bolt preload forces

NX Nastran allows for direct entry of bolt preload forces. When you use this functionality, the input file must contain:

- BOLT bulk entries to select CBEAM and CBAR elements to treat as bolts.
- BOLTFOR bulk entries to define bolt preload forces and assign them to the bolts defined by BOLT bulk entries.
- BOLTLD case control commands to select BOLTFOR bulk entries to include in either the global case or subcases.

Beginning with NX Nastran 7.1, the new BOLTLD bulk entry is available for use in conjunction with the existing bolt preload procedure. The BOLTLD bulk entry provides flexibility in defining combinations of bolts and bolt preload forces to include in the global case or subcases. It also provides a means to scale bolt preload forces.

When you use the BOLTLD bulk entry, the input file must contain:

- BOLT bulk entries to select CBEAM and CBAR elements to treat as bolts.
- BOLTFOR bulk entries to define bolt preload forces and assign them to the bolts defined by BOLT bulk entries.
- BOLTLD bulk entries to define combinations of BOLTFOR bulk entries and scale the corresponding bolt preload forces.
- BOLTLD case control commands to select BOLTLD bulk entries to include in either the global case or subcases.

You can also have an input file where some BOLTLD case control commands select BOLTLD bulk entries and some BOLTLD case control commands select BOLTFOR bulk entries. In this case, if the set identification number (SID) referenced by a BOLTLD case control command exists on both a BOLTLD bulk entry and BOLTFOR bulk entries, the BOLTLD bulk entry is selected.

The use of the BOLTLD bulk entry is optional. In the absence of BOLTLD bulk entries, the pre-NX Nastran 7.1 bolt preload procedure is still valid.

For more information regarding bolt preload, see the Applied Loads chapter of the *NX Nastran 7.0 User's Guide*.

Bolt preload example

A structure contains two bolts. One bolt has a M10 x 1.5 thread and the other has a M20 x 2.5 thread. Both bolts are modeled using CBAR elements. Four subcases are considered:

- Bolt preload forces at 70% of proof strength and the service loads excluded.
- Bolt preload forces at 70% of proof strength and the service loads included.
- Bolt preload forces at 80% of proof strength and the service loads excluded.
- Bolt preload forces at 80% of proof strength and the service loads included.

The following table contains bolt and proof strength data for use in the model. The equivalent radius is calculated from the tensile stress area.

Bolt size	Grade	Tensile stress area (mm^2)	Equivalent radius (mm)	Proof strength (MPa)	Preload force at 70% of proof strength (kN)	Preload force at 80% of proof strength (kN)	Preload force at 100% of proof strength (kN)
M10 x 1.5	5.8	58	4.30	380	15.4	17.6	22.0
M20 x 2.5	5.8	245	8.83	380	65.2	74.5	93.1

Three input files are provided that model this situation:

- The first input file does not include the new BOLTLD bulk entry. In this case, the BOLTLD case control commands select the BOLTFOR bulk entries having the same SID.
- The second input file includes BOLTLD bulk entries. In this case, the BOLTLD case control commands select the BOLTLD bulk entries having the same SID.
- The third input file demonstrates how both BOLTLD bulk entries and BOLTFOR bulk entries can be selected by BOLTLD case control commands in the same input file. In this case, the BOLTLD case control commands in two subcases select a BOLTLD bulk entry and the BOLTLD case control commands in the other two subcases select BOLTFOR bulk entries.

Input file without BOLTLD bulk entries

```
SOL 101


$   SUBCASES
$
SUBCASE = 1
  SUBTITLE    = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE    = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE    = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE    = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK


CBAR        2237       2    8001    8000     1.0     0.0     0.0
CBAR        2238       3    8003    8002     1.0     0.0     0.0
$
$   BOLT ENTRIES
$
BOLT           1       1    2237
BOLT           2       1    2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR        1  15.4+3       1
BOLTFOR        1  65.2+3       2
BOLTFOR        2  17.6+3       1
BOLTFOR        2  74.5+3       2


ENDDATA
```

Input file with BOLTLD bulk entries

```
SOL 101


$   SUBCASES
$
SUBCASE = 1
  SUBTITLE    = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE    = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE    = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE    = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK
........
........
CBAR        2237       2    8001    8000     1.0     0.0     0.0
CBAR        2238       3    8003    8002     1.0     0.0     0.0
$
$   BOLT ENTRIES
$
BOLT           1       1    2237
BOLT           2       1    2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR        1  22.0+3       1
BOLTFOR        2  93.1+3       2
$
$   SCALE BOLT PRELOAD ENTRIES
$
BOLTLD         1    0.70     1.0       1     1.0       2
BOLTLD         2    0.80     1.0       1     1.0       2


ENDDATA
```

Input file with BOLTLD case control commands selecting both BOLTLD and BOLTFOR bulk entries

```
SOL 101


$   SUBCASES
$
SUBCASE = 1
  SUBTITLE    = Bolt preload force at 70% of proof strength without service loads
  BOLTLD = 1
$
SUBCASE = 2
  SUBTITLE    = Bolt preload force at 70% of proof strength with service loads
  BOLTLD = 1
  LOAD = 1
$
SUBCASE = 3
  SUBTITLE    = Bolt preload force at 80% of proof strength without service loads
  BOLTLD = 2
$
SUBCASE = 4
  SUBTITLE    = Bolt preload force at 80% of proof strength with service loads
  BOLTLD = 2
  LOAD = 1
$
BEGIN BULK


CBAR        2237       2    8001    8000     1.0     0.0     0.0
CBAR        2238       3    8003    8002     1.0     0.0     0.0
$
$   BOLT ENTRIES
$
BOLT           1       1    2237
BOLT           2       1    2238
$
$   BOLT PRELOAD ENTRIES
$
BOLTFOR        2  17.6+3       1
BOLTFOR        2  74.5+3       2
BOLTFOR        3  22.0+3       1
BOLTFOR        3  93.1+3       2
$
$   SCALE BOLT PRELOAD ENTRIES
$
BOLTLD         1    0.70     1.0       3


ENDDATA
```

BOLTFOR Preload Force on Set of Bolts

Defines preload force for a set of bolts.

Format:

1	2	3	4	5	6	7	8	9	10
BOLTFOR	SID	LOAD	B1	B2	B3	B4	B5	B6	
	B7	THRU	B8						
	B9	B10	-etc-						

Example:

BOLTFOR	4	1000.0	12	THRU	21				
	1	4	6	9	10	26	32	34	
	37	43	51						

Fields:

Field	Contents
SID	Set identification number (SID) of BOLTFOR bulk entry. (Integer>0)
LOAD	Magnitude of the preload force. (Real)
Bi	Bolt identification numbers (BID) defined by BOLT bulk entries. (Integer>0 or use “THRU” option. For “THRU” option, B7<B8)

Remarks:

1. Multiple BOLTFOR entries having the same SID can be used and the data will be combined.

2. If the SID referenced by a BOLTLD case control command is present on both a BOLTLD bulk entry and BOLTFOR bulk entries, the BOLTLD bulk entry takes precedence and is selected. The BOLTFOR bulk entries that are not selected can optionally be included in the subcase by listing their SID in one of the Li fields on the BOLTLD bulk entry.

3. Repeated BID in a subcase is not allowed. A repeated BID occurs when:

   - A BID is included more than once on a BOLTFOR bulk entry.
   - A BID is included on multiple BOLTFOR bulk entries having the same SID.

- A BID is included on multiple BOLTFOR bulk entries having different SID, but referenced by the same BOLTLD bulk entry.

BOLTLD (bulk entry)

Bolt preload combining and scaling

Combines sets of bolts defined by BOLTFOR bulk entries and optionally scales the corresponding bolt preload forces.

Format:

1	2	3	4	5	6	7	8	9	10
BOLTLD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc-						

Example:

BOLTLD	101	0.5	1.0	3	6.2	4			

Fields:

Field	Contents
SID	Set identification number (SID) of BOLTLD bulk entry. (Integer>0)
S	Overall scale factor. (Real)
Si	Scale factor for individual Li. (Real)
Li	SID of BOLTFOR bulk entries. (Integer>0)

Remarks:

1. The applied preload value P_i for each BOLT bulk entry referenced by BOLTFOR bulk entry Li is given by:

   $P_i = S * Si * P_{Li}$

   where P_{Li} is the preload value defined for BOLTFOR bulk entry Li, Si is the individual scale factor corresponding to Li, and S is the overall scale factor.

2. Li must be unique. However, a single Li on a BOLTLD bulk entry can reference multiple BOLTFOR bulk entries. When this occurs, all the preload forces defined by the BOLTFOR bulk entries sharing the same SID are scaled by the corresponding individual scale factor Si and the overall scale factor S.

3. The bolt identification numbers (BID) in all BOLTFOR bulk entries selected by a single BOLTLD bulk entry must be unique.

4. The SID of each BOLTLD bulk entry must be unique.

BOLTLD (case control) Bolt Preload Set Selection

Selects either a BOLTLD bulk entry or BOLTFOR bulk entries for bolt preload processing.

Format:

BOLTLD=n

Examples:

```
BOLTLD=5
```

Describers:

Describer	Meaning
n	Set identification number (SID) of a BOLTLD bulk entry or BOLTFOR bulk entries. See Remark 2. (Integer>0)

Remarks:

1. Bolt preload is supported in SOLs 101, 103, 105, 107 through 112, and 601.

2. If the SID referenced by a BOLTLD case control command exists on both a BOLTLD bulk entry and BOLTFOR bulk entries, the BOLTLD bulk entry is selected.

3. Subcases without BOLTLD case control commands should be placed at either the beginning or end of the subcases.

4. For the special case where the sparse solver is used, the model does not contain contact, and the keyword scratch = yes, BOLTLD case control commands in the subcases must be ordered. For example, if a model contains three subcases with BOLTLD = 1 used twice and BOLTLD = 2 used once, the BOLTLD case control commands should be ordered with:

   - The first and second subcases containing BOLTLD = 1 and the third subcase containing BOLTLD = 2.

   - The first subcase containing BOLTLD = 2 and the second and third subcases containing BOLTLD = 1.

   If the iterative solver is used or contact is used or the keyword scratch = no, ordering the subcases in this way is not necessary, but is recommended. Doing so minimizes the number of matrix decompositions required during the solution.

5. Superelements with preloaded bolts are allowed. However, the elements used to define the bolts along with the bolt preload forces must be in the residual.

6. For dynamic solution sequences, a static subcase containing the BOLTLD case control command must be referenced by a STATSUB case control command.

Bolt preload with contact definitions

When a bolt preload definition exists, a two solution process occurs:

1. Preload Solution

   The elements representing the bolts are reduced in stiffness, the bolt preloads are applied, and a linear statics solution runs.

2. Final Solution

   The bolt strains determined from the preload solution plus the service loads are applied, and the final linear statics solution runs.

When a contact definition exists in the bolt preload process, the contact conditions are included in both the preload and final solutions. To decrease the solution time, now the final solution begins with the contact status from the end of the preload solution. As a result, the contact element creation steps and some of the initial contact iterations are avoided in the final solution.

Models that include bolt preload and contact definitions could produce slightly different results relative to previous releases, especially if the number of contact changes at convergence is not close to zero.

Bolt stiffness update

During the bolt preload solution process, the elements representing the bolts are reduced in stiffness and the bolt preloads are applied. Previously the bolt stiffness was decreased by BOLTFACT in all degrees-of-freedom (DOF). A problem when reducing the stiffness in all DOF is that if sliding occurs, the preload results are unpredictable. Beginning in NX Nastran 7.1, only the axial stiffness is decreased and the other five DOF remain unchanged.

Chapter 7 Optimization

Best design cycle

The SOL 200 optimization process adjusts design variables as it searches for the optimum conditions while it works in the domain of the constraints and the objective. At specific times in this solution process, the FE model is updated with the current values of the design variables, and a finite element analysis is requested. Such analysis results are used to update the optimization algorithm with current design responses and their gradients. Each loop through a finite element analysis, a sensitivity analysis, a solution of the generated optimization problem, and design updating in the SOL 200 process is considered a design cycle.

The .f06 file reports the design variables, constraints, responses, and the objective for each design cycle in which output is requested. By default, the initial and final design cycles are reported, but you can request output for all using PARAM,NASPRT,1.

The design variables, the value of the objective, and the most critical constraint value for the final design cycle do not necessarily reflect the best design, while often this may be the case. In previous releases, finding the design cycle which best satisfies the constraints and objective was a manual process. You had to consider the design responses from many design cycles.

Now the best design cycle is automatically determined by the SOL 200 process. An arrow points to the best design in the **Summary of Design Cycle History** table in the .f06 output file. In addition, the best design cycle is indicated under the table heading.

The following example .f06 output for an objective function maximization job demonstrates this enhancement.

Note that the best design in this example is consistent with Remark 9 on the DESOBJ case control command, except that the highest value among the feasible objectives has been selected because this is a maximization problem.

```
*************************************************************
S U M M A R Y   O F   D E S I G N   C Y C L E   H I S T O R Y
*************************************************************

     NUMBER OF FINITE ELEMENT ANALYSES COMPLETED         11
     NUMBER OF OPTIMIZATIONS W.R.T. APPROXIMATE MODELS   10

           OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY
           ***(Best Design Found At Cycle    6)***
```

CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT	
INITIAL		1.519072E+05		1.236165E+01	
1	1.461574E+05	1.295791E+05	1.279397E-01	4.037928E-01	
2	1.212510E+05	1.205930E+05	5.455981E-03	2.678087E-02	
3	1.146383E+05	1.123581E+05	2.029436E-02	7.671907E-01	
4	1.123708E+05	1.118530E+05	4.629403E-03	-4.006884E-02	
5	1.103805E+05	1.103248E+05	5.053970E-04	8.832471E-01	
6	1.123653E+05	1.123211E+05	3.942382E-04	-4.216779E-02	**<<== BEST DESIGN**
7	1.119073E+05	1.117819E+05	1.122373E-03	-5.640221E-02	
8	1.085288E+05	1.084036E+05	1.154684E-03	-1.635617E-02	
9	1.081977E+05	1.081934E+05	3.928151E-05	-4.287751E-02	
10	1.076919E+05	1.076685E+05	2.173193E-04	-4.717002E-02	

Updated bulk data output

The optimization algorithm adjusts your design variables while searching for optimum values. At the end of a successful optimization solution, you may want to update the complete bulk data with the new design variables as well as with any other design variable dependent data that will have changed in order to run some relevant analysis or to continue with the optimization. NX Nastran now automates this step for you.

You can now request the complete, updated bulk data output. The new bulk data, which is written to the punch file, will match your original except that it will contain updated design variables, related properties, grid point data for shape optimization, and when mode tracking is requested, eigenvalue DRESP1 data for the design cycle for which the bulk data is written. You can merge this entire new bulk data containing the updated bulk data with the other components of a relevant input file for any consecutive solution.

The existing ECHO case control command is used to make the request. The new option ECHO=PUNCH(BSTBULK) is used to output the bulk data for the best design cycle, or ECHO=PUNCH(NEWBULK) to output the bulk data for the last design cycle. Often the last design cycle and the best design cycle coincide. However, BSTBULK is useful for cases where the best design cycle is actually an earlier cycle. In contrast, NEWBULK may be useful when you know that the job will be continued later starting from the last design cycle.

ECHO Bulk Data Echo Request

Controls echo (i.e., printout) of the Bulk Data.

Format:

$$\text{ECHO}=\left\{\begin{bmatrix}\text{UNSORT}\begin{bmatrix}\text{SORT}[([\text{EXCEPT}]\text{cdni},...)]\\ \text{NOSORT}\end{bmatrix}\\ \text{BOTH}\\ \text{NONE}\end{bmatrix},\ \text{PUNCH}\left[\begin{Bmatrix}\text{BSTBULK}\\ \text{NEWBULK}\end{Bmatrix}\right]\right\}$$

Examples:

```
ECHO=NOSORT
ECHO=BOTH
ECHO=PUNCH,SORT(MAT1,PARAM)
ECHO=SORT(EXCEPT DMI,DMIG)
ECHO=PUNCH(BSTBULK)
ECHO=PUNCH(NEWBULK)
ECHO=SORT,PUNCH(BSTBULK)
```

Describers:

Describer	Meaning
UNSORT	The unsorted Bulk Data will be printed. If SORT is also not specified then the sorted Bulk Data will not be printed.
SORT	The sorted (arranged in alphanumeric order) Bulk Data will be printed.
NOSORT	The sorted Bulk Data will not be printed. If UNSORT is also not specified then the unsorted Bulk Data will not be printed.
cdni,...	Bulk Data entry names to be included or excluded by EXCEPT, in the sorted echo printout. The PUNCH file is not affected by cdni.
EXCEPT	Exclude cdni Bulk Data entries from sorted echo printout. See Remark 6.
BOTH	Both sorted and unsorted Bulk Data will be printed. This is equivalent to ECHO=SORT, UNSORT.
NONE	Neither sorted nor unsorted Bulk Data will be printed.

Describer	Meaning
PUNCH	The entire Bulk Data will be written to the punch file in sorted form.
BSTBULK	For SOL 200 design optimization runs, an unsorted updated bulk data file will be written to the punch file, using the updated design variables and properties as well as any mode-tracked eigenvalue DRESP1 (i.e. the updated design data) for the BEST design cycle obtained and indicated in the SOL 200 run. See Remark 7.
NEWBULK	For SOL 200 design optimization runs, an unsorted updated bulk data file will be written to the punch file, using the updated design variables and properties as well as any mode-tracked eigenvalue DRESP1 (i.e. the updated design data) for the LAST design cycle obtained and indicated in the SOL 200 run. See Remark 7.

Remarks:

1. If no ECHO command appears, a sorted Bulk Data will be printed.

2. Comments will appear at the front of the sorted file if ECHO=PUNCH.

3. Portions of the unsorted Bulk Data can be selectively echoed by including the commands ECHOON and ECHOOFF at various places within the Bulk Data. ECHOOFF stops the unsorted echo until an ECHOON command is encountered. Many such pairs of commands may be used. The ECHOON and ECHOOFF commands may be used in the Executive and Case Control Sections; however, ECHOOF should not be the first entry as continuation entries will not be handled correctly.

4. If the SORT (cdni,...) is specified in a restart in SOLs 101 through 200, then the continuation entries will not be printed.

5. If the SORT (cdni,...) describer is used, then it must appear as the last describer, as in the example above.

6. If EXCEPT is specified then it must be specified before all cdni. All Bulk Data entry types will be listed except those given for cdn1, cdn2, etc. If EXCEPT is not specified, then only those Bulk Data entry types listed under cdn1, cdn2, etc. will be listed.

7. For SOL 200, often the last design cycle and the best design cycle coincide. However, BSTBULK is useful for cases where the best design cycle is actually an earlier cycle. In contrast, NEWBULK may be useful when the user knows that the job will be continued later starting from the last design cycle.

PCOMPG support

The PCOMPG bulk entry was introduced in NX Nastran 7 to define a different number of shell composite layers and a different ply order on adjacent elements.

The composite properties defined with the PCOMPG entry can now be included as design variables in a SOL 200 optimization or sensitivity analysis.

The design variable inputs for the PCOMPG entry, which are the same as for the PCOMP entry, are defined with the DVPREL1 or DVPREL2 bulk entries.

The following tables list the DVPREL1 and PCOMPG bulk entries input formats:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYID1	MID1	T1	THETA1	SOUT1				
	GPLYID2	MID2	T2	THETA2	SOUT2				

To define a DVPREL1 property relation input for the PCOMPG entry, TYPE should be “PCOMPG”, PID should be assigned to the PCOMPG property ID, and PNAME/FID can refer to any of the PCOMPG field names or numbers. For example, entering either “THETA1” or “15” will both define the same PCOMPG field position.

The input example below represents four PCOMPG design variable/relations: THETA1 (field number “15”), THETA2 (field number “25”), THETA3 (field number “35”), and THETA4 (field number “40”).

```
....
PCOMPG         1             0.   8.E+7    HOFF                        +
+            101       1   1.E-4      0.     YES                       +
+            102       1   1.E-4     90.     YES                       +
+            103       1   1.E-4     45.     YES                       +
+            104       1   1.E-4     45.     YES                       +
+            105       1   1.E-4     45.     YES                       +
+            106       1   1.E-4     45.     YES                       +
+            107       1   1.E-4     90.     YES                       +
+            108       1   1.E-4      0.     YES

....
DVPREL1,5,PCOMPG,1,15,,,,,+
+,5,1.0
DVPREL1,6,PCOMPG,1,25,,,,,+
+,6,1.0
DVPREL1,7,PCOMPG,1,35,,,,,+
+,7,1.0
DVPREL1,8,PCOMPG,1,40,,,,,+
+,8,1.0
....
DESVAR,5,THETA1,1.0,1.0,90.0
DESVAR,6,THETA2,1.0,1.0,90.0
DESVAR,7,THETA3,1.0,1.0,90.0
DESVAR,8,THETA4,1.0,1.0,90.0
```

PSHELL structural damping support

You can select a different material on the PSHELL bulk entry for membrane (MID1), bending (MID2), transverse shear (MID3), and membrane-bending coupling (MID4). You can now also include the structure damping value from each of these materials as design variables in a SOL 200 optimization or sensitivity analysis. The DVMREL1 or DVMREL2 bulk entries define the material/variable relations.

The parameter input PARAM,SHLDAMP,DIFF is required to use the capability:

When PARAM,SHLDAMP = SAME (default), the structural damping coefficient (GE) defined on a PSHELL MID1 material will be used by all MIDi for that PSHELL.
When PARAM,SHLDAMP = DIFF, the structural damping coefficient (GE) defined on each PSHELL MIDi will be used. Any structural damping coefficient (GE) values which are blank default to zero.

PBUSH structural damping support

The PBUSH bulk entry includes the fields GE1, GE2, GE3, GE4, GE5, and GE6 to define structural damping for each degree-of-freedom.

You can now include these fields as design variables in a SOL 200 optimization or sensitivity analysis. The DVPREL1 or DVPREL2 bulk entries define the property/variable relations.

The parameter input PARAM,BSHDAMP is *not* required to turn on the capability since the default value "DIFF" includes the fields GE2-GE6. When PARAM,BSHDAMP,SAME is defined, the fields GE2-GE6 are ignored, and only fields GE1 are considered.

Note: When you use any of the GE2-GE6 fields, it is important to fully define all degrees-of-freedom deemed critical to the result, because a blank field will default to a zero value.

Additional enhancements

Alternate optimizer update

The alternate optimizer developed by Siemens PLM has been updated with several improvements. The enhancements include better quality of results in general due to algorithmic improvements and some improvement in the use of memory.

A major modification involves the normalization of constraints for purposes of this optimizer. Now when you select the alternate optimizer, a true normalization is performed, and the normalized constraint values always vary between -1.0 and +1.0 in a symmetric manner. Due to the method of normalization employed with the use of this optimizer, you do not need to account for a small denominator. Note that the "Maximum Value of Constraint" column in the "Summary of Design Cycle History" table in the f06 file will also contain different values based on which optimizer is selected.

The normalization for the alternate optimizer follows the expressions below.

If a bound is sandwiched between zero and r:

g=(LALLOW–r) / |r| or   g=(r–UALLOW) / |r|

Otherwise:

g=(LALLOW–r) / |2*LALLOW–r| or   g=(r–UALLOW) / |2*UALLOW–r|

The only special case gives a constraint value of 0.0, thus no difficulties arise with a zero denominator.

The constraint normalization for the DOT optimizer remains the same as before.

The alternate optimizer is selected with the system cell 425 (ADSOPT):
NASTRAN SYSTEM(425) = 1 -or- NASTRAN ADSOPT = 1

The DOT optimizer remains the default for NX Nastran 7.1:
NASTRAN SYSTEM(425) = 0 (default)

DRSPAN update

When the DRSPAN case control command was introduced with NX Nastran 6.0, it was noted that it was not available for use with superelements (see the *NX Nastran 6.0 Release Guide*). With NX Nastran 7.1, the DRSPAN command now can be used with superelements, as long as all DRESP1 responses in a DRSPAN related DRESP2 or DRESP3 are from the same superelement.

Chapter

8 New materials

3-D orthotropic materials

The new MAT11 and MATT11 bulk entries are available to define a 3-D orthotropic material definition. The new materials can be used by the CHEXA, CPENTA, CPYRAM, and CTETRA element types, and are selected with the MID field on the PSOLID bulk entry. They are supported in all solutions except SOL 601 and 701. Although you can include the MAT11 and MATT11 bulk entries in SOL 200, they cannot be used as design variables by the DVMREL1 and DVMREL2 bulk entries.

MAT11 Solid Element Orthotropic Material Property Definition

Defines the material properties for a 3-D orthotropic material for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT11	MID	E1	E2	E3	NU12	NU13	NU23	G12	
	G13	G23	RHO	A1	A2	A3	TREF	GE	

Example:

MAT11	101	2.1E7	2.2E7	2.3E7	0.31	0.29	0.33	2.1E6	
	2.2E6	2.3E6	0.34	0.35	0.36	0.37	0.38	0.39	

Fields:

Field	Contents
MID	Material identification number. (Integer > 0)
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Real > 0.0; no default, must be defined)
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Real > 0.0; no default, must be defined)
E3	Modulus of elasticity in ply layup direction, also defined as the thickness direction or 3-direction. (Real > 0.0; no default, must be defined)
NU12	Poisson's ratio (ϵ_2/ϵ_1 for uniaxial loading in 1-direction). Note that $NU_{21}=\epsilon_1/\epsilon_2$ for uniaxial loading in 2-direction is related to NU_{12}, E_1, and E_2 by the relation $NU_{12}*E_2=NU_{21}*E_1$. (Real > 0.0; no default, must be defined)
NU13	Poisson's ratio (ϵ_3/ϵ_1 for uniaxial loading in 1-direction). Note that $NU_{31}=\epsilon_1/\epsilon_3$ for uniaxial loading in 3-direction is related to NU_{13}, E_3, and E_1 by the relation $NU_{31}*E_1=NU_{13}*E_3$. (Real > 0.0; no default, must be defined)

Field	Contents
NU23	Poisson's ratio (ϵ_3/ϵ_2 for uniaxial loading in 2-direction). Note that $NU_{32}=\epsilon_2/\epsilon_3$ for uniaxial loading in 3-direction is related to NU_{23}, E_2, and E_3 by the relation $NU_{23}*E_3=NU_{32}*E_2$. (Real > 0.0; no default, must be defined)
G12	Shear modulus in plane 1-2. (Real > 0.0; no default, must be defined)
G13	Transverse shear modulus in shear in 1-3 plane. (Real > 0.0; no default, must be defined)
G23	Transverse shear modulus in shear in 2-3 plane. (Real > 0.0; no default, must be defined)
RHO	Mass density. (Real or blank; Default = 0.0)
A1	Thermal expansion coefficient in longitudinal direction. (Real or blank; Default = 0.0)
A2	Thermal expansion coefficient in lateral direction. (Real or blank; Default = 0.0)
A3	Thermal expansion coefficient in thickness direction. (Real or blank; Default = 0.0)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. (Real or blank; Default = 0.0)
GE	Structural damping coefficient. (Real or blank; Default = 0.0)

Remarks:

1. In general, NU_{12} is not the same as NU_{21}, but they are related by $NU_{ij}/E_i = NU_{ji}/E_j$. Furthermore, material stability requires that $E_i > NU_{ij}^2 E_j$ and $1-NU_{12}NU_{21}-NU_{23}NU_{32}-NU_{31}NU_{13}-2NU_{21}NU_{32}NU_{13} > 0.0$.

2. MAT11 materials may be made temperature dependent by use of the MATT11 entry.

3. MAT11 entries cannot be used as design variables in SOL 200 (via the DVMREL1 and DVMREL2 bulk entries).

MATT11 Solid Orthotropic Material Temperature Dependence

Defines the temperature dependent material property for a 3-D orthotropic material for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATT11	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU13)	T(NU23)	T(G12)	
	T(G13)	T(G23)	T(RHO)	T(A1)	T(A2)	T(A3)		T(GE)	

Example:

MATT11	20	10							
			11						

Fields:

Field	Contents
MID	Material property identification number that matches the identification number on MAT11 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for Young's modulus 1. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for Young's modulus 2. (Integer ≥ 0 or blank)
T(E3)	Identification number of a TABLEMi entry for Young's modulus 3. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)
T(NU13)	Identification number of a TABLEMi entry for Poisson's ratio 13. (Integer ≥ 0 or blank)
T(NU23)	Identification number of a TABLEMi entry for Poisson's ratio 23. (Integer ≥ 0 or blank)
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)

Field	Contents
T(G13)	Identification number of a TABLEMi entry for shear modulus 13. (Integer ≥ 0 or blank)
T(G23)	Identification number of a TABLEMi entry for shear modulus 23. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(A1)	Identification number of a TABLEMi entry for thermal expansion coefficient 1 in 1-direction. (Integer ≥ 0 or blank)
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient 2 in 2-direction. (Integer ≥ 0 or blank)
T(A3)	Identification number of a TABLEMi entry for thermal expansion coefficient 2 in 3-direction. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer ≥ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT11 entry referenced in field 2. The value in a particular field of the MAT11 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 10 and A1 is modified by TABLEMi 11. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT11 entry.

2. Any quantity modified by this entry must have a value on the MAT11 entry.

Chapter

9 Element enhancements

Solid element support in solutions 106 and 129

The CPYRAM solid pyramid element now supports geometry nonlinear conditions in solutions 106 and 129. Previously, the CPYRAM element behaved linearly in these solutions.

In addition, the variable grid CHEXA, CPENTA elements now support geometry nonlinear conditions in solutions 106 and 129. These previously behaved linearly in these solutions.

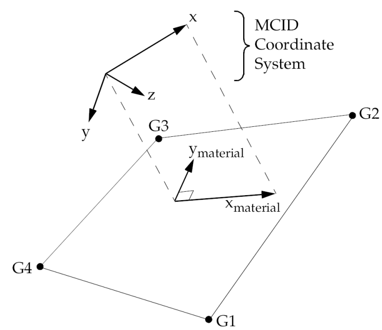
The following table summarizes the solid element support in solutions 106 and 129.

Table 9-1. SOL 106 and 129 Solid Element Support

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

Shell element material coordinate tolerance

The MCID field on the shell elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR selects a material coordinate system. The X-direction of the material coordinate system is projected onto the face of the shell element to determine the material X-direction.



If the X-direction of a material coordinate system is perpendicular or very close to being perpendicular to the element face, a projection is either impossible or unpredictable.

The new system cell 489 is now available to adjust the perpendicularity angle tolerance for shell element material orientations. A fatal error will occur when the angle between a shell element face normal and the X-direction of the material coordinate system (MCID) is less than the value of SYSTEM(489). The default is 1.0E-8 degrees.

Shell element thickness check

A more rigorous shell element thickness check now exists. A fatal error occurs if a Ti value of "0.0" exists on any of the elements CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAR8, CQUADR, CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8.

The system cell 495 is available to turn the new thickness check off.

If SYSTEM(495) = 0 (default), a fatal error occurs if a thickness value of "0.0" exists on any shell element.

If SYSTEM(495) = 1, the new thickness check is turned off.

CPYRAM in acoustics

The pyramid element CPYRAM can now be included in acoustic models. You can use these models to perform fully coupled fluid-structure analysis. Common applications are acoustic and noise control analysis; for example, in the passenger compartments of automobiles. The pyramid can be included as part of either the fluid or structure, and is permitted at the interface regions where fluid-structure coupling occurs.

Chapter 10 DMP improvements

Multilevel RDMODES enhancement

The multi-level Recursive Domain Lanczos method (RDMODES) was introduced in NX Nastran 6.1 for very large scale normal modes solutions. In NX Nastran 7, RDMODES was enabled for the frequency response solutions (SOL 111) including coupled acoustics problems. In addition, a new sparse eigenvector recovery option was added.

RDMODES includes the following enhancements in this release.

- The performance of RDMODES is enhanced in the partition phase. The RDMODES partitioning modification improves performance for multilevel RDMODES applications, especially with large nrec values. No additional inputs are required.

- You can now include superelements with an RDMODES modal analysis. This DMP superelement feature improves performance by allowing RDMODES to be used (using the nrec keyword) in a DMP superelement job.

- RDMODES with the rdsparse option now supports residual vectors (PARAM, RESVEC), panel participation factors (PARAM, PANELMP), absolute displacement enforced motion (sys422=1), and modal contributions. No new inputs are required.

  Note that the accelerated residual vector calculation with RDMODES takes advantage of the rdsparse option, and is more efficient than the original one in terms of computational time and I/O usage. The residual vectors with the accelerated calculation may differ slightly from the original, which cannot be used in conjunction with rdsparse. If necessary, you can request the original resvec method by specifying PARAM, RDRESVEC, NO in the bulk data. In this case, the rdsparse option will be disabled automatically, which is likely to result in dramatically reduced performance.

Inputs for RDMODES

Multilevel RDMODES is activated by the nastran keywords dmp and nrec:

NASTRAN dmp=*p* nrec=*m*

where p is the number of processors and m is the number of external partitions.

Remarks

1. The optional keyword "rdscale" is a factor to modify the selected frequency range in the EIGRL specification for eigensolutions of each substructure. The default value of rdscale is 3.5 for multilevel RDMODES.

2. There is no default value for the nrec keyword.

RDMODES performance case study example (partitioning performance improvement)

- Vehicle model
- About 546,000 grid points, 500,000 elements, 3,280,000 degrees of freedom
- SOL 111 frequency response analysis
- IBM Power 6/4.2GHz workstation

The following table compares multi-level RDMODES runs (parallel=8, nrec=128) with and without the partitioning improvement.

Method	**Partitioning** min:sec	**Elapsed** min:sec	**I/O** GBytes
RDMODES NXN71	3:58	115:01	1828.6
RDMODES NXN70	9:05	123:11	1837.5

With the improvement, it is more than a factor of 2 faster than version 7.0 in the partitioning phase (4 min vs. 9 min). I/O usage is also reduced. Larger improvements can be expected when the model and nrec values are large.

GDSTAT method improvement

The GDSTAT method (SOL 101 using geometric domain decomposition and DMP) in NX Nastran 7 partitioned the model using the SEQP module. Now in NX Nastran 7.1, two methods are available: using SEQP and using the GPARTN module. The SEQP method is selected with gpart=0, and GPARTN is selected with gpart=1 (default).

You can activate this method by the following:

nastran dmp=*n* gpart=1

where *n* denotes the number of processors. gpart=1 is optional since it is the default.

Remark

The GDSTAT method (gpart=1) can handle solution 101 jobs with CWELD elements, CFAST elements, and glue conditions. These elements and glue conditions are not supported in the SEQP method.

Chapter 11 Electromagnetics interface

Overview

NX Nastran 7.1 delivers a new capability for transferring an external force field to an NX NASTRAN structural load. The new functionality is useful when analyzing structural components of linear and torque motors. In these applications, the surface loads from a 3rd party electromagnetic simulation product such as MAXWELL are integrated into an NX Nastran solution. Responses of the structure to the combined structural and external electromagnetic field loads are computed.

The external force field is described by 3D force components and their spatial coordinates. These locations are referred to as external force points. A set of structural grid points in the vicinity of the external force points are selected to define a surface patch. NX Nastran transfers the external forces to the grids on the surface patch using an existing surface spline technique used in the aero-elastic solutions. The existing structural forces in the model will be augmented with the equivalent structural forces to execute the analysis.

Inputs

The coupling mechanism may be applied to linear static, normal modes and direct frequency response solutions in NX Nastran 7.1. The user interface is based on a new case control entry selection and DMI matrix input.

The availability of external field loads must be indicated by one or more subcases containing the new case control command

EFLOAD (NDIR=n, SCID=m) = yes/no.

The value n indicates a general surface normal of the surface patch upon which the external load is applied. The values of n are 1, 2 and 3, indicating a normal direction of z, y, or x, respectively. In the case of cylindrical local coordinate system specified by SCID=m, the values of n = 1, 2, or 3 indicate the surface normal orientation of z, theta, or r. In the cylindrical coordinate case the angles must be in the 0 to 360 degrees range. In this initial implementation, the cylindrical coordinate system's origin and z-axis must be consistent with the origin and z-axis of the basic coordinate system. This restriction will be removed in future versions.

The short version of the format EFLOAD (NDIR=n) = yes is also allowed, resulting in everything being in the basic coordinate system. The shortest form of EFLOAD = yes will use the default value of n=1 and is equivalent to EFLOAD (NDIR=1) = yes. Finally any form of the EFLOAD command with "no" on the right hand side will result in the omission of external field loads.

The same subcase where EFLOAD is applied defines the external load files via the existing command

P2G = efpoints, efforce

where the files are in standard DMI format with names that you specify. The efpoints file may contain points in rectangular or cylindrical coordinates, in order to follow the geometry conveniently. The corresponding efforce file, however, contains the forces in rectangular components in both cases, in adherence to the spatial external field solution.

The surface section of the structural model to receive the loads (in the proximity of, but not necessarily coincident with, the epoints) is defined by the existing mechanism

SET k = G1, G2, etc

PARTN k

There may be as many subcases with the above content as many external load surface segments.

The projection of the external solution points, contained in the efpoints file, onto the structure must fall into the envelope of the designated interface area of the structure specified by the SET and PARTN mechanism.

Note that the last subcase of any job must not contain external field loads, but only structural loads (if any) or general output requests. Thus, in the case of only one external field load surface, there must be two subcases.

Using the statically pre-stressed normal modes capability, you must run a static solution (SOL101) with EFLOAD to generate the .pch file. The SOL101.pch file simply needs to be included in the bulk data section of the normal modes restart run.

The static subcase of the normal modes restart run must contain:

EFLOAD = yes

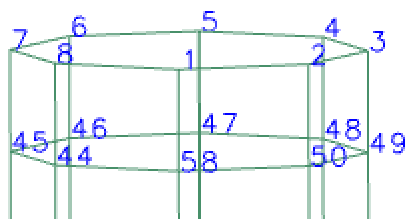
The normal modes subcase must contain the STATSUB case control word referring to the subcase with the EFFLOAD keyword. Otherwise the process is as usual in pre-stressed normal modes.

The set-up of the direct frequency response (108) solution is conceptually the same as the linear static run. The external field loads will constitute the scaling factor component of the dynamic load.

Example

A cylindrical shell modeled with CQUAD4 elements is constrained along the bottom edge. The external field loads are applied half way between the top two rings of nodes at 45–degree increments around the perimeter. The loads include FX and FZ of 100.0 units applied in the basic coordinate system at each location.

The grids shown in the partial model below define the surface patch.



The relevant components of the input file are:

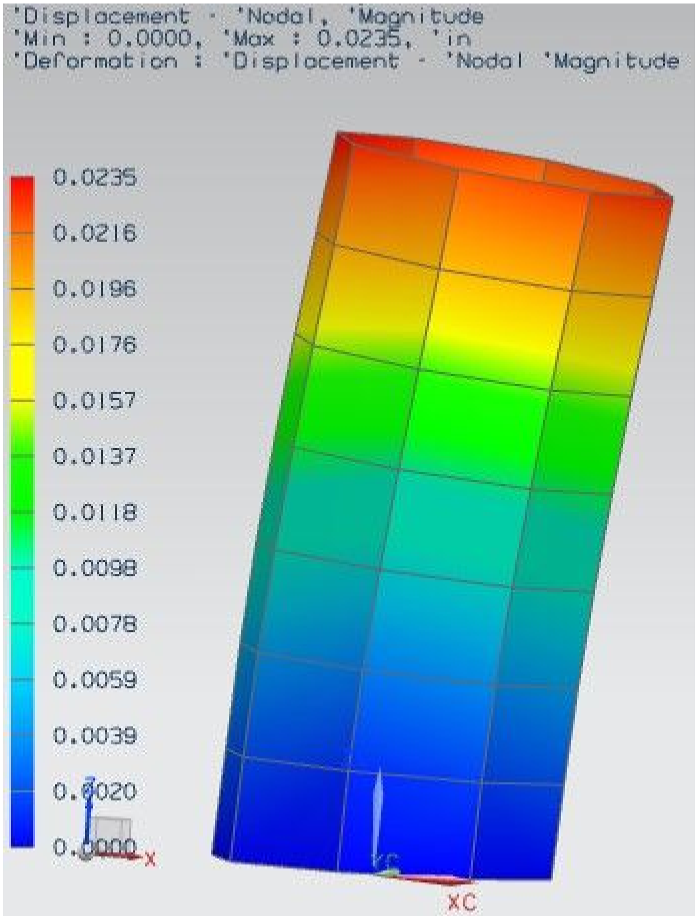
```
Case Control
....
SUBCASE 1
$
$SET selects the grids for the surface patch

SET 1 =  1 thru 8, 44 thru 50, 58
$
PARTN 1
p2g = ep1, ef1
efload (nDIR=3, SCID=9) = yes
$
SUBCASE 2
OUTPUT
DISPLACEMENT(PRINT) = ALL
STRESS(PRINT) = ALL

Bulk Data
.....
$Cylindrical coordinate system used for surface normal and external force locations
CORD2C, 9, , 0.0, 0.0, 0.0, 0.0, 0.0, 1.0, +C1
+C1, 20.0, 0.0, 1.0
......
$Force Locations entered in cylindrical coodinates
$
dmi,ep1,0,2,1,0,,3,8
dmi,ep1,1,1, 20. , 0.0, 74.286
dmi,ep1,2,1, 20. , 45., 74.286
```

```
dmi,ep1,3,1, 20. , 90.0, 74.286
dmi,ep1,4,1, 20. , 135.0, 74.286
dmi,ep1,5,1, 20. , 180.0, 74.286
dmi,ep1,6,1, 20. , 225.0, 74.286
dmi,ep1,7,1, 20. , 270.0, 74.286
dmi,ep1,8,1, 20. , 315.0, 74.286
$
$Force Vectors entered in basic coordinates
$
dmi,ef1,0,2,1,0,,3,8
dmi,ef1,1,1, 100.0,   0.0, 100.
dmi,ef1,2,1, 100.0,   0.0, 100.
dmi,ef1,3,1, 100.0,   0.0, 100.
dmi,ef1,4,1, 100.0,   0.0, 100.
dmi,ef1,5,1, 100.0,   0.0, 100.
dmi,ef1,6,1, 100.0,   0.0, 100.
dmi,ef1,7,1, 100.0,   0.0, 100.
dmi,ef1,8,1, 100.0,   0.0, 100.
$
```

The structural loads resulting from the external force vectors produce the following deformations.



Chapter 12 Acoustics

Default updates

The defaults for the NORMAL and OVLPANG parameters on the ACMODL bulk entry have been modified.

- The default for the parameter NORMAL when METHOD = “AS” has been modified from 0.2 to 0.5.
- The default for the parameter OVLPANG has been modified from 30.0 to 60.0.

Table 12-1. NORMAL and OVLPANG descriptions

Field	Contents
NORMAL	Outward normal search distance to detect fluid-structure interface. (Real > 0.0; Default = 0.5 for METHOD = “AS”) If METHOD = “AS” and SRCHUNIT is “REL”, NORMAL is a ratio of the height of the fluid box in the outward normal direction to the fluid surface to the maximum edge length of the fluid free face. IF METHOD = “AS” and SRCHUNIT is “ABS”, NORMAL is the outward search distance in the model/absolute units.
OVLPANG	Angular tolerance in degrees used to decide whether a fluid free face and a structural face can be considered as overlapping. If the angle between the normals of the fluid and structural faces exceeds this value, they cannot be coupled. (Real > 0.0; Default = 60.0).

See the updated ACMODL bulk entry.

CPYRAM in acoustics

The pyramid element CPYRAM can now be included in acoustic models. You can use these models to perform fully coupled fluid-structure analysis. Common applications are acoustic and noise control analysis; for example, in the passenger compartments of automobiles. The pyramid can be included as part of either the fluid or structure, and is permitted at the interface regions where fluid-structure coupling occurs.

Alternative coupling option

The new AREAOP field on the ACMODL bulk entry is available to select an alternate fluid-to-structural coupling option. AREAOP=0 selects the default recommended NX Nastran coupling option. Specifying AREAOP=1 selects the alternate option, which applies an area correction and removes parallel disconnected faces from coupling.

See the updated ACMODL bulk entry.

ACMODL Fluid-Structure Interface Modeling Parameters

Defines modeling parameters for the interface between the fluid and the structure.

Format:

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	OVLPANG	SRCHUNIT	
	INTOL	AREAOP	reserved						

Example:

ACMODL					0.25	AS			

Fields:

Field	Contents
INTER	INTER is ignored when METHOD="AS". When METHOD = "CP", INTER defines the type of interface between the fluid and the structure. (Character = "IDENT" or "DIFF"; Default = "DIFF").
INFOR	Indicates if FSET and SSET are to be used to define the fluid-structure interface. (Character = "ALL", "ELEMENTS", "PID" or "NONE", Default = "NONE"). See Remark 2. Acceptable values for AS method are "ELEMENTS" and "PID". Acceptable values for CP method are "ALL" or "NONE".

Field	Contents
FSET	Defines the fluid elements for the interface.
	When METHOD = "AS", FSET can be blank, or is the +/-ID of a SET1 bulk entry. The SET1 is interpreted according to the value of INFOR:
	If INFOR = "ELEMENTS", SET1 is a list of fluid element IDs. If INFOR = "PID", SET1 is a list of fluid element property IDs.
	When a positive SET1 ID is entered on FSET, only these fluid elements are considered. When a negative SET1 ID is entered on FSET, these elements will be excluded, and all other fluid elements are considered. See Remark 3. (Integer or blank)
	If METHOD = "CP", FSET represents the SET1 entry that contains a list of fluid grid points on the interface. (Integer > 0 or blank)
SSET	Defines the structural elements for the interface.
	When METHOD = "AS", SSET can be blank, or is the +/-ID of a SET1 bulk entry. The SET1 is interpreted according to the value of INFOR:
	If INFOR = "ELEMENTS", SET1 is a list of structural element IDs. If INFOR = "PID", SET1 is a list of structural element property IDs.
	When a positive SET1 ID is entered on SSET, only these structural elements are considered. When a negative SET1 ID is entered on FSET, these elements will be excluded, and all other structural elements are considered. See Remark 3. (Integer or blank)
	If METHOD = "CP", SSET represents the SET1 entry that contains a list of structure grid points on the interface. (Integer > 0 or blank)
NORMAL	Outward normal search distance to detect fluid-structure interface. (Real > 0.0; Default = 0.5 for METHOD = "AS") See Remark 4.
	If METHOD = "AS" and SRCHUNIT is "REL", NORMAL is a ratio of the height of the fluid box in the outward normal direction to the fluid surface to the maximum edge length of the fluid free face. IF METHOD = "AS" and SRCHUNIT is "ABS", NORMAL is the outward search distance in the model/absolute units.
	If METHOD = "CP" and INTER = "IDENT", NORMAL represents a tolerance in units of length used in determining the fluid-structure interface. (Real > 0.0; Default = 0.001 for METHOD = "CP")

Field	Contents
METHOD	Interface calculation method. “AS” = Accelerated Search Method (Default). “CP” = Closed Pressure Method (old method).
OVLPANG	Angular tolerance in degrees used to decide whether a fluid free face and a structural face can be considered as overlapping. If the angle between the normals of the fluid and structural faces exceeds this value, they cannot be coupled. (Real > 0.0; Default = 60.0).
SRCHUNIT	Search units. (Character; Default=’REL’). See Remark 4. = ‘ABS’ for absolute model units. = ‘REL’ for relative model units based on element size.
INTOL	Inward normal search distance to detect fluid-structure interface. See Remark 4. INTOL is ignored when METHOD = “CP”. If METHOD = “AS” and SRCHUNIT is “REL”, INTOL is a ratio of the height of the fluid box in the inward normal direction to the fluid surface to the maximum edge length of the fluid free face. IF METHOD = “AS” and SRCHUNIT is “ABS”, INTOL is the inward search distance in the model/absolute units (Real > 0.0; Default = 0.20)
AREAOP	Alternative fluid-structure coupling method selection. See Remark 5. (Integer; Default=0) = 0 The recommended method is used (default). = 1 The alternative method is used.

Remarks:

1. Only one ACMODL entry is allowed. If this entry is not present, defaults will be used.

2. The INFOR field is interpreted as follows:

   AS Method:

   The acceptable values for this method are INFOR = “ELEMENTS” or “PID”. The AS method will interpret the FSET and SSET as described in Remark 4.

   If INFOR = “ALL” or “NONE”, this field will be ignored.

   CP Method:

If INTER = “IDENT” and INFOR = “NONE”, the FSET and SSET entries are ignored.

When INTER = “IDENT” and INFOR = “ALL”, matching is checked only at those grid points referenced by FSET and SSET.

If INTER = “DIFF”, the FSET and SSET are used as described in Remark 4.

If INFOR = “ELEMENTS”, NX Nastran will reset this field to INFOR = “NONE” .

3. The FSET and the SSET fields are interpreted as follows:

AS Method:

FSET and SSET are used only when INFOR = “ELEMENTS” or “PID”.

- If FSET > 0, only the fluid elements included in this set will participate in the coupling. All the other fluid elements will be ignored.
- If FSET < 0, the fluid elements included in this set will be suppressed from coupling computations.
- If FSET is blank, all fluid elements will be used for coupling computations.
- If SSET > 0, only the structural elements included in this set will participate in the coupling. All the other fluid elements will be ignored.
- If SSET < 0, the structural elements included in this set will be suppressed from coupling computations.
- If SSET is blank, all structural elements will be used for coupling computations.

CP Method:

If INTER = “IDENT”, FSET and SSET should be greater than zero. If FSET/SSET is <= 0, a FATAL message is issued.

The grid points defined by FSET and SSET must lie exactly on the fluid-structure interface. A FATAL message is issued if the FSET or SSET do not have a corresponding and coincident point in SSET or FSET, respectively.

If INTER = “DIFF” a FATAL message is issued if any point in SSET or FSET does not lie exactly on the interface.

4. The fields NORMAL and INTOL are interpreted as follows:

AS Method: (Real > 0.0; Default = 0.20)

If SRCHUNIT = “REL”, NORMAL is a ratio of the outward height of the bounding box to the maximum edge length of the fluid free face. That is, if L is the largest edge of the fluid free face, the height H of the bounding box used to search for structural faces will be NORMAL * L. INTOL applies similarly, but the inward direction.

If SRCHUNIT = “ABS”, NORMAL defines the outward height of the fluid bounding box in the model/absolute units. INTOL defines the inward height of the fluid bounding box in the model/absolute units.

If SRCHUNIT = “ABS” and NORMAL or INTOL are blank, SRCHUNIT will be reset to “REL” and their corresponding default value is used.

CP Method: (Real > 0.0; Default = 0.001)

INTOL is not used when METHOD=CP.

If INTER = “IDENT” or “DIFF”, NORMAL represents a tolerance in units of length used in determining the fluid-structure interface. The default value is 0.001.

5. AREAOP=0 selects the default, recommended NX Nastran coupling option. Specifying AREAOP=1 selects an alternate option, which applies an area correction and removes parallel disconnected faces from the coupling.

Chapter 13 Windows I/O Option

The default I/O option on the windows platforms have changed from File Mapping to Raw I/O. This is necessary for Windows 7, Windows Server 2008, and Vista systems but might slightly degrade the performance on Windows XP and Windows Server 2003; especially on systems that have a lot of memory. If you have a Windows XP or Windows Server 2003 system and wish to use File Mapping, remove the entry “sysfield=...” from the nastran rcf file.

Chapter

14 Upward compatibility

Updated modules

ACMG

Computes the coupling matrix for fluid/structure interface at all points or only points on a given structural panel.

Updated Format:

```
ACMG        PANSLT,BGPDT,CSTM,SIL,ECT,EQACST,NORTAB,EQEXIN,EDT,GEOM2/
            {AGG or APART}/
            LUSET/MPNFLG/NUMPAN/S,N,PANAME/IPANEL/MATCH/
            PNLPTV $
```

New Input Data Block:

GEOM2	Table of bulk data images related to element connectivity

Updated Input Data Block:

EDT	Element deformation table, which contains SET1 **and SET3** entries

CNTITER

Perform surface to surface contact using the element iterative solver.

Updated Format:

```
CNTITER    KELM,PGALL,KDICT,SILS,ECT,BGPDT,CSTM,EDT,CASECC,USET,RG,MPT,
           YG,SLT,MDICT,MELM,EPT,CNELM,ELCNST,ELCTST,ELGSTF,GLUESEQ,BIN/
           UGV1,QG1,OQGCF1,OBC1,CONFON,ELTRCT1,OSPDSI1,OSPDS1,BOUT/
           NSKIP/NLOADS/NOFAC/
           S,N,MAXO/S,N,MAXI/S,N,CNTS/S,N,AITK/S,N,RESET/S,N,MINOLP/
           S,N,TARPEN/S,N,ADAPT/S,N,SCALMT/S,N,IMODE/WTMASS $
```

New Input Data Block:

BIN	Results from bolt preload phase.

New Output Data Block:

BOUT	Results from bolt preload phase.

New Parameter:

WTMASS	Input-real-default=1.0. Weight to mass conversion factor.

DDRMM

Performs matrix method data recovery

Computes data recovery items (stress, displacements, forces, strains, forces) directly from the modal solution in frequency response, transient response, or scaled response spectra analysis using the matrix method.

Updated Format:

```
DDRMM      CASECC,UH,OL,IUG,IQG,IES,IEF,XYCDB,IUG1,IQG1,IES1,IEF1,UHD/
           OUG,OQG,OES,OEF,UNUSED5/
           OPTION/NOCOMP/PEXIST/ACOUSTIC/ACOUT/PREFDB/SEID/APP $
```

New Input Data Block:

UHD	Modal velocities solution matrix for the h-set (modal degrees-of-freedom). Required only for frequency response.

DMIIN

Inputs DMI entries to DMAP.

Input matrices referenced on DMI Bulk Data entries.

Updated Format:

```
DMIIN        DMI,DMINDX/DMI1,DMI2,DMI3,DMI4,DMI5,DMI6,DMI7,
             DMI8,DMI9,DMI10/PARM1/PARM2/PARM3/PARM4/PARM5/
             PARM6/PARM7/PARM8/PARM9/PARM10/NAME1/NAME2 $
```

New Parameters:

NAME1	Input-character-default = " ". If NAME1 is specified, read the DMI matrix associated with NAME1 instead of the output data block DMI1.
NAME2	Input-character-default = " ". If NAME2 is specified, read the DMI matrix associated with NAME2 instead of the output data block DMI2.

DOM12

Performs soft and hard convergence checks in design optimization.

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/
             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 Blocks:

OBJTBG	Overall (global) objective function data block.
ALBULK	Family of data blocks for bulk files for the original analysis/design model as well as for all auxiliary models, if any.

AMLIST	List of auxiliary model identification numbers.

DOM9

Performs the approximate optimization problem.

Performs the approximate optimization problem using design variables, constraints, responses and sensitivity information.

Updated Format:

```
DOM9        XINIT,DESTAB,CONSBL*,DPLDXI*,XZ,
            DXDXI,DPLDXT*,DEQATN,DEQIND,DXDXIT,
            PLIST2*,OPTPRMG,R1VALRG,RSP2RG,R1TABRG,
            CNTABRG,DSCMG,DVPTAB*,PROPI*,CONS1T,
            OBJTBG,COORDO,CON,SHPVEC,DCLDXT,
            TABDEQ,EPTTAB*,DBMLIB,BCON0,BCONXI,
            BCONXT,DNODEL,RR2IDR,RESP3RG,CVALRG/
            XO,CVALO,R1VALO,R2VALO,PROPO,R3VALO/
            OBJIN/S,N,OBJOUT/PROTYP/PROPTN/UNUSED1/
            UNUSED2/UNUSED3/UNUSED4/UNUSED5/UNUSED6/
            UNUSED7/UNUSED8/UNUSED9/UNUSED10 $
```

Updated Input Data Block:

CVALRG	The global data block for retained constraint values.

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,TMPFRL,DEQIND,
            DEQATN,DESGID,DVPTAB,VIEWTB,OINT,PELSET,DFRFNC,
            TSPAN23,DIT,EDOM/
            OBJTAB,CONTAB,R1TAB,RESP12,RSP1CT,FRQRSP,CASEDS,
            OINTDS,PELSETDS,DESELM,RESP3,TNSPAN23,SPAN1RG/
            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 $
```

New Input Data Block:

EDOMS	Table of Bulk Data entries related to design sensitivity and optimization for current superelement.

Updated Input Data Block:

EDOM	Global table of Bulk Data entries related to design sensitivity and optimization.

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/EFOPT $
```

New Output Data Block:

MDLIST	List of modes for output selected by effective mass fraction

FOCOEL

Form contact elements.

Updated Format:

```
FOCOEL      CASECC,BGPDT,CSTM,GEOM2,EST,MPT,CONTACT,SIL,GPSNTC/
            CNELM,GPECTC/
            S,N,NSKIP/S,Y,CONTGLUE/S,N,NLHEAT/S,N,GLUSET/S,N,NCELS/S,N,MAXO/
            S,N,MAXI/S,N,CNTS/S,N,AITK/S,N,MPLI/S,N,RESET/S,N,FRICTM/
            S,N,TARPEN/S,N,ADAPT/S,N,SCALMT//S,N,REFOPT/S,N,MEL $
```

New Input Data Block:

GPSNTC	Table of grid point shell normals.

New Parameter:

MEL Input-integer-no default. Maximum number of elements connected to a grid.

GP5

Creates table of static loads for panels in coupled fluid/structure analysis.

Updated Format:

```
GP5          ECT,BGPDT,EQEXIN,EDT,SIL,GEOM2/
             PANSLT,EQACST,NORTAB/
             S,N,MPNFLG/S,N,NUMPAN/S,N,MATCH/NASOUT/GETNUMPN/
             S,N,METHOD/S,N,SKINOUT/S,N,NORMAL/S,N,OVLPANG/
             S,N,INTOL/S,N,ABSFLG/S,N,NGAUSS $
```

New Input Data Block:

GEOM2 Table of bulk data images related to element connectivity

Updated Input Data Block:

EDT Table of Bulk Data entry images related to element deformation, aerodynamics, p-element analysis, divergence analysis, and the iterative solver. Also contains SET1 **and SET3** entries

IFP

Reads Bulk Data Section

Reads in the Bulk Data and outputs the finite element model in table form.

Updated Input Data Block:

EDT Table of Bulk Data entry images related to element deformation, aerodynamics, p-element analysis, divergence analysis, and the iterative solver. Also contains SET1 **and SET3** entries

MATMOD

Transforms matrix or table data blocks according to one of many options into output matrix or table data blocks.

Updated 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 $
```

Updated Input Data Blocks:

Ii	Input data blocks. I1 is required; I2 through **I15** may not be necessary depending on the value P1.

Updated Option P1=40

Remove unused q-set degrees-of-freedom from USET.

Updated Format:

```
MATMOD     USET,VAXW,,,,,,,,,,,,,/USETN,VAXWN/40 $
```

New Output Data Block:

VAXWN	Modified VAXW corresponding to USETN.

Updated Option P1=44

Create a g-set partitioning vector for sparse data recovery based on the output requests in the case control section. This option is only valid for frequency and transient response output requests.

Updated Format:

```
MATMOD      BGPDTS,SILS,CASES,USETS,EPTS,GEOM2S,EQEXINS,
            XYCDB,GMTG,GOATG,GPECT,PCDB,POSTCDB,SETMC,EDT/PARTV,
            /44/NOEPT/NOSE/IWHO///NUMPAN $
```

New Input Data Block:

EDT	Element deformation table. Contains PANEL, SET1, and SET3 bulk entries.

New Parameter:

NUMPAN	Input-integer default=0. Number of acoustic panels.

Updated Remark:

4. The supported case control output requests are:

a. DISPLACEMENT
b. VELOCITY
c. ACCELERATION
d. OLOAD
e. STRESS/ELSTRESS
f. STRAIN/ELSTRAIN
g. FORCE/ELFORCE
h. EDE
I. EKE
j. ESE
k. XYPEAK
l. XYPRINT
m. XYPLOT
n. XYPUNCH
o. MPCFORCE
p. SPCFORCE
q. GPFORCE

r. GPSTRESS
s. GPSTRAIN
t. MODCON
u. PANCON

New Option P1=51

Generate a surface spline to connect two point fields in close proximity, but not necessarily coincident.

Format:

```
MATMOD        IM1,IM2,,,,,,,,,,,,,/OM1,/51/P2/P3/P4/P5 $
```

Input Data Blocks:

IM1 Matrix of independent points. 3 rows by NI columns. NI = number of independent points.

IM2 Matrix of dependent points. 3 rows by ND columns. ND = number of dependent points.

Output Data Block:

OM1 Transformation matrix describing the surface spline connection. NI rows by ND columns. NI = number of independent points. ND = number of dependent points.

Parameters:

P2,P3 Input-integer default=0,0. Used to define boundary condition of the surface spline.

0,0 Surface spline is unconstrained at the edges.

-1,0 A zero moment X-directional end condition is imposed on the surface spline.

0,-1 A zero moment Y-directional end condition is imposed on the surface spline.

P4 Input-integer default=1. Coordinate pair selection.

1 XY point pairs are used for the surface spline fit.

2 XZ point pairs are used for the surface spline fit.

3 YZ point pairs are used for the surface spline fit.

P5 Input-real default=1.0. Used to stiffen springs applied during the surface spline generation.

New Option P1=53

Generate a g-size partitioning vector and a list of grid IDs that correspond to the structural grids that are a part of an acoustic coupling matrix.

Format:

```
MATMOD      EQEXIN,USET,SIL,AG0MAX,,,,,,,,,,,/PARTV,ACGRIDS/53///
            S,N,NOPARTV $
```

Input Data Blocks:

EQEXIN Equivalence table between external and internal grid/scalar identification numbers.

USET Degree-of-freedom set membership table for g-set.

SIL Scalar index list.

AG0MAX Column matrix of maximum values in each row of the AG0 matrix (usually derived using MATMOD option 6).

Output Data Block:

PARTV Column partitioning vector for all 6 degrees-of-freedom of grids which are a part of the acoustic coupling.

ACGRIDS Column vector containing the grid IDs of the grids which are a part of the acoustic coupling; the grid IDs are represented as real values.

Parameters:

NOPARTV	Output-integer. NOPARTV is set to -1 if the output data blocks are null or cannot be generated. Otherwise, it is set to zero.

Remarks:

1. None of the data blocks can be purged.

MODACC

OFREQ and OTIME command processor.

Removes columns in solution and load matrices based on the OTIME and OFREQ case control commands.

Updated Format:

```
MODACC       CASECC,OL,U,P1,P2,P3/
             OL1,U1,P11,P21,P31/APP/IOPT $
```

New Parameter:

IOPT   Input-integer-default=0. Processing options:

0=process OFREQ or OTIME.

1=process SETMC for MODCON.

2=process SETMC for PANCON.

MODEPF

Computes mode participation factors for fluid-structure models in frequency response analysis.

Updated Format:

```
MODEPF       BGPDT,USET,CASECC,EDT,ABESF*,
             PHASH2,UHFS,PHDFH,MFHH,BFHH,
             KFHH,FOL,ABEH*,PHDFH1,PHDFH2,
             UHFF,AH,PFHF,UNUSED,PNLLST,
             VGA,GEOM2/
             GPMPF,FMPF,SMPF,PMPF,LMPF,
             MPFMAP/
             NOFREQ/NOLOADF/GRIDFMP/NUMPAN/PNQALNAM/
             SYMFLG/MPNFLG/FLUIDMP/STRUCTMP/PANELMP/
             GRIDMP/NOSASET/FILTERF/FILTERS $
```

New Input Data Block:

GEOM2	Table of bulk data images related to element connectivity

MTRXIN

Converts matrices input on DMIG Bulk Data entries to matrix data blocks.

Updated Formats:

Form 1 – Simplified (CASECC is purged)

MTRXIN

$$,,\text{MATPOOL},\begin{Bmatrix}\text{EQEXIN}\\ \text{EQDYN}\end{Bmatrix},,/$$

$$\text{NAME1},\text{NAME2},\text{NAME3},,/$$

$$\begin{Bmatrix}\text{LUSET}\\ \text{LUSETD}\end{Bmatrix}/\text{S,N,NONAME1}/\text{S,N,NONAME2}/\text{S,N,NONAME3}/\quad \$$$

Note

An output data block has been added.

Form 2 – Case Control Command Selection of stiffness, mass, and damping (or square) matrices (IOPT=1 for K2GG, and so on, and IOPT=0 for K2PP, and so on, and TF)

MTRXIN

$$\text{CASECC, MATPOOL,} \left\{ \begin{matrix} \text{EQEXIN} \\ \text{EQDYN} \end{matrix} \right\} \text{,,} \left\{ \begin{matrix} \\ \text{TFPOOL} \end{matrix} \right\} /$$

$$\left\{ \begin{matrix} \text{K2GG, M2GG, B2GG, K42GG, } \mathbf{A2GG} \\ \text{K2PP, M2PP, B2PP,,} \end{matrix} \right\} /$$

$$\left\{ \begin{matrix} \text{LUSET} \\ \text{LUSETD} \end{matrix} \right\} \text{/ S, N, NOK2 / S, N, NOM2 / S, N, NOB2 / S, N, NOK42 } \mathbf{/S,N,NOA2/} \left\{ \begin{matrix} 1 \\ 0 \end{matrix} \right\} \ \$$$

Form 3 – Case Control Command selection of load (or rectangular) matrix (IOPT=2)

MTRXIN

$$\begin{matrix} \text{CASECC, MATPOOL, EQEXIN,, /} \\ \text{P2G,,,, /} \\ \text{LUSET / S, N, NOP2G ////}\mathbf{/}\text{2} \quad \$ \end{matrix}$$

Form 4 – Selection of DMIK, DMIJ and DMIJI by data block names MATKi, MATJi, and MATJIi.

MTRXIN

$$\text{,, MATPOOL,} \left\{ \begin{matrix} \text{AEBGPDTK} \\ \text{AEBGPDTJ} \\ \text{AEBGPDTI} \end{matrix} \right\} \text{,, /}$$

$$\left\{ \begin{matrix} \text{MATK1} \\ \text{MATJ1} \\ \text{MATI1} \end{matrix} \right\}, \left\{ \begin{matrix} \text{MATK2} \\ \text{MATJ2} \\ \text{MATI2} \end{matrix} \right\}, \left\{ \begin{matrix} \text{MATK3} \\ \text{MATJ3} \\ \text{MATI3} \end{matrix} \right\} \text{,, /}$$

$$\left\{ \begin{matrix} \text{S, N, LKSET} \\ \text{S, N, LJSET} \\ \text{S, N, LISET} \end{matrix} \right\} \text{/ S, N, NOMAT1 / S, N, NOMAT2 / S, N, NOMAT3 ///} \left\{ \begin{matrix} 3 \\ 4 \\ 5 \end{matrix} \right\} \ \$$$

Form 5 – Selection of stiffness, mass, damping, and loads (or square) matrices by K2PNAM, and so on, input parameter values (IOPT=10 through 12).

MTRXIN

$$,,\text{MATPOOL},\left\{\begin{matrix}\text{EQDYN}\\ \text{EQEXIN}\\ \text{EQEXIN}\end{matrix}\right\},,\left\{\begin{matrix}\text{TFPOOL}\\ \\ \end{matrix}\right\}\ /$$

$$\left\{\begin{matrix}\text{MATP1}\\ \text{MATG1}\\ \text{RMATG}\end{matrix}\right\},\left\{\begin{matrix}\text{MATP2}\\ \text{MATG2}\\ \end{matrix}\right\},\left\{\begin{matrix}\text{MATP3}\\ \text{MATG3}\\ \end{matrix}\right\},\left\{\begin{matrix} \\ \text{MATG4}\\ \end{matrix}\right\},\left\{\begin{matrix} \\ \text{MATG5}\\ \end{matrix}\right\}\ /$$

$$\left\{\begin{matrix}\text{LUSETD}\\ \text{LUSET}\\ \text{LUSET}\end{matrix}\right\}/\text{S, N, NOMAT1}/\text{S, N, NOMAT2}/\text{S, N, NOMAT3}/\text{S, N, NOMAT4}/\text{S,N,NOMAT5}$$

$$\text{MATNAM1}/\text{MATNAM2}/\text{MATNAM3}/\text{MATNAM4}/\text{MATNAM5}/\left\{\begin{matrix}\text{TFLID}\\ \end{matrix}\right\}/\text{NFEXIT}\quad \$$$

Form 6 - Selection of DMIK, DMIJ, and DMIJI matrices by the MATNAMi input parameter values (IOPT=13 through 15)

MTRXIN

$$,,\text{MATPOOL},\left\{\begin{matrix}\text{AEBGPDTK}\\ \text{AEBGPDTJ}\\ \text{AEBGPDTI}\end{matrix}\right\},,\ /$$

$$\left\{\begin{matrix}\text{MATK1}\\ \text{MATJ1}\\ \text{MATI1}\end{matrix}\right\},\left\{\begin{matrix}\text{MATK2}\\ \text{MATJ2}\\ \text{MATI2}\end{matrix}\right\},\left\{\begin{matrix}\text{MATK3}\\ \text{MATJ3}\\ \text{MATI3}\end{matrix}\right\},,/$$

$$\left\{\begin{matrix}\text{S, N, LKSET}\\ \text{S, N, LJSET}\\ \text{S, N, LISET}\end{matrix}\right\}/\text{S, N, NOMAT1}/\text{S, N, NOMAT2}/\text{S, N, NOMAT3}///\left\{\begin{matrix}13\\ 14\\ 15\end{matrix}\right\}/$$

$$\text{MATNAM1}/\text{MATNAM2}/\text{MATNAM3}\quad \$$$

Updated Output Data Block:

K2GG, etc.	Matrices defined on DMIG Bulk Data entries and referenced by the K2GG, M2GG, B2GG, K42GG, **A2GG**, K2PP, M2PP, B2PP, or P2G Case Control commands

Updated Parameter:

IOPT		Input-integer-default=0. Case Control command selection flag.
	0	No Case Control command selection (see Form 1) or K2GG, and so on, and TFL Case Control command selection (see Form 2)
	1	K2GG, and so on, Case Control command selection (see Form 2)
	2	P2G Case Control command selection (see Form 3)
	3	DMIK selection by output data block name (see Form 4)
	4	DMIJ selection by output data block name (see Form 4)
	5	DMIJI selection by output data block name (see Form 4)
	10	K2PP, M2PP, and B2PP selection by input parameter value (see Form 5)
	11	K2GG, M2GG, B2GG, K42GG, **and A2GG** selection by input parameter value (see Form 5)
	12	P2G selection by input parameter value (see Form 5)
	13	DMIK selection by input parameter value (see Form 6)
	14	DMIJ selection by input parameter value (see Form 6)
	15	DMIJI selection by input parameter value (see Form 6)

Updated Remark 3:

3. Forms 2 and 3 are used to select the matrices with Case Control commands: K2GG, M2GG, B2GG, K42GG, **A2GG,** K2PP, M2PP, B2PP, or P2G. “-2GG” matrices are of dimension g by g. “-2PP” matrices are of dimension p by p. The P2G matrix has g-rows, with the number of columns determined by the several methods used to input rectangular matrices described on the DMIG entry.

Updated Examples:

1. Assume the Bulk Data contains two DMIG matrices, named M1 and M2, which reference grid and/or scalar points only. The following set of DMAP instructions generate these two matrices in matrix format, multiply them, and print the result.

```
MTRXIN   ,,MATPOOL,EQEXIN,,/M1,M2,,,/LUSET/S,N,NOM1/S,N,NOM2$
IF (NOM1 > -1 AND NOM2 > -1) THEN $
  MPYAD   M1,M2,/PRODUCT $
  MATPRN  PRODUCT//$
ENDIF $
```

2. Assume the Bulk Data contains two DMIG matrices, MASS and STIFF, which reference grid and/or scalar points only. The following Case Control and DMAP instructions generate these two matrices in matrix format and add them to the structural mass and stiffness.

   Case Control:

```
M2GG = MASS
K2GG = STIFF
```

   DMAP instructions:

```
MTRXIN   CASECC,MATPOOL,EQEXIN,,/STIFF,MASS,,,/
         LUSET/S,N,NOSTIFF/S,N,NOMASS///1 $
IF (NOSTIFF > -1) THEN $
  ADD    KGG,STIFF/KGGNEW $
  EQUIVX KGGNEW/KGG/ALWAYS $
ENDIF $
IF (NOMASS > -1) THEN $
  ADD    MGG,MASS/MGGNEW $
  EQUIVX MGGNEW/MGG/ALWAYS $
ENDIF
```

OUTPRT

Constructs sparse load reduction and sparse data recovery partitioning vectors.

Updated Parameter:

MCFLAG Input-integer-default=0. Modal contribution set usage flag.

0	grids and elements
1	grids only
2	elements only
3	**acoustic/fluid points only**

RESTART

Data block comparison

Compares two data blocks and invokes dependencies.

Updated Format:

```
RESTART      DB1,DB2,DLSTIN/DLSTOUT/
             INVOKE/SPEXP/DPEXP/NDDLNAM/S,CMPDIF $
```

New Parameter:

CMPDIF Output-integer. If CMPDIF=1, the comparison between DB1 and DB2 found differences. Only valid for DB1 and DB2 existing with DLSTIN purged.

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/
          OPG1,OQG1,OUG1,OES1,OEF1,PUG,OGPKE1/
          APP/S,N,NOSORT2/NOCOMP/ACOUSTIC/METRIK/
          ISOFLG/GPF/ACOUT/PREFDB/TABS/
          SIGMA/ADPTINDX/ADPTEXIT/BSKIP/FREQW/
          BTBRS/LANGLE/OMID $
```

New Input Data Block:

MDLIST List of modes for output selected by effective mass fraction.

VDRMC

Creates modal contribution tables based on modal contribution output requests.

Updated Format:

```
VDRMC     CASEG,SETMC,AMC,NMC,MAG,OL,MFRQ,ECT,BGPDT,TEXTSE/
          OUTFLE/
          APP/S,N,NOSORT2/S,N,NOSOUT/FMODE/IRTYPE/FSFLAG $
```

New Parameter:

FSFLAG Input-integer-default=0. Fluid/structure flag. Only applies to IRTYPEs 1 thru 7.
0 = structural modal contribution results from structural modes
1 = fluid modal contribution results from structural modes
11 = fluid modal contribution results from fluid modes

New modules

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.

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,U7DOF,U8DOF,//
             SEID/WTMASS/FLXERR $
```

Input Data Blocks:

CASES	Case control table associated with superelement.
LAMA	Eigenvalue summary table for superelement.
PHIG	Mode shape matrix (U8-set) for superelement.
EMVD	Equivalent modal viscous damping matrix for superelement.
AMAT	State-space [A] matrix for superelement.
BMAT	State-space [B] matrix for superelement.
CMAT	State-space [C] matrix for superelement.
U7DOF	U7-set DOF vector (input DOF) for superelement.
U8DOF	U8-set DOF vector (output DOF) for superelement.
MFORC	Modal force vector for superelement.

Parameters:

SEID	Input-integer-no default. Superelement ID number.
WTMASS	Input-real-no default. Value of WTMASS parameter from PARAM,WTMASS,value.
FLXERR	Output-integer-no default. Error flag. Options are: 0: No error 1: Error

Remarks:

This module will generate a MATLAB .m script file.

VDRPC

Creates panel or grid contribution tables based on panel/grid contribution output requests.

Format:

VDRPC

$$\text{CASEG,SETMC,APC,NPC,MAG,FOL,BGPDT,}\begin{Bmatrix}\text{PNLLST}\\ \text{SLGRIDS}\end{Bmatrix}/$$

OUTFLE /

APP/S, N, NOSORT2 /S, N, NOSOUT / IRTYPE / DATTYP $

Input Data Blocks:

CASEG	Table of Case Control command images for current subcase.
SETMC	Table of SETMC Case Control definitions.
APC	Absolute panel contributions results matrix.
NPC	Normalized panel contributions results matrix.
MAG	Magnitudes of modal contributions matrix.
FOL	Frequency response frequency output list.
BGPDT	Basic grid point definition table.
PNLLST	Panels list table (for DATTYP=1).
SLGRIDS	List of selected grids (for DATTYP=2).

Output Data Blocks:

OUTFLE	Table of panel or grid contributions in SORT1 format.

Parameters:

APP Input-character-no default. Analysis type. Allowable values:

‘FREQRESP’ Frequency response

NOSORT2 Output-integer-no default. Solution set SORT2 format flag. Set to 1 if SORT2 format is requested; -1 otherwise.

NOSOUT Output-integer-default=0. Output flag. Set to 1 if any output is requested; -1 otherwise.

IRTYPE Input-integer-default=1. Panel contributions results type.
1 = pressures
2 = first derivative of pressures
3 = pressure and first derivatives
4 = second derivative of pressures
5 = pressures and second derivatives
6 = first and second derivative of pressures
7 = pressures, first derivatives, and second derivatives

DATTYP Input-integer-no default. Data type.
1 = panel contributions
2 = grid contributions

Remarks:

1. Contribution totals will appear as ID number 0 in OUTFLE and will be printed to the .f06 file with the label “TOTAL”.

2. None of the data blocks can be purged.

Updated datablocks

AXIC68

Element property table (Pre-MSC Nastran Version 69).

New RECORD – FORCE(4201,42,18)

Static force.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	G	I	Grid point identification number
3	CID	I	Coordinate system identification number

Word	Name	Type	Description
4	F	RS	Scale factor
5	N1	RS	Vector component in the coordinate system defined by CID
6	N2	RS	Vector component in the coordinate system defined by CID
7	N3	RS	Vector component in the coordinate system defined by CID

New RECORD – GRAV(4401,44,26)

Acceleration or gravity load.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	CID	I	Coordinate system identification number
3	A	RS	Acceleration scale factor
4	N1	RS	Vector component in the coordinate system defined by CID
5	N2	RS	Vector component in the coordinate system defined by CID
6	N3	RS	Vector component in the coordinate system defined by CID
7	MB	I	Bulk data section with CID definition: -1=main, 0=partitioned

New RECORD – LOAD(4551,61,84)

Static load combination.

Word	Name	Type	Description
1	SID	I	Load set identification number

Word	Name	Type	Description
2	S	RS	Overall scale factor
3	Si	RS	Scale factor on Li
4	Li	I	Load set identification number
Words 3 and 4 repeat until (-1,-1) occurs			

New RECORD – MOMENT(4801,48,19)

Static moment.

Word	Name	Type	Description
1	SID	I	Load set identification number
2	G	I	Grid point identification number
3	CID	I	Coordinate system identification number
4	M	RS	Moment scale factor
5	N1	RS	Vector component in the coordinate system defined by CID
6	N2	RS	Vector component in the coordinate system defined by CID
7	N3	RS	Vector component in the coordinate system defined by CID

New RECORD – MPCADD(4891,60,83)

Multipoint constraint set combination.

Word	Name	Type	Description
1	SID	I	Set identification number
2	S	I	Set identification number of multipoint constraint set

Word	Name	Type	Description
Word 2 repeats until -1 occurs			

New RECORD – SEQGP(5301,53,4)

Grid and scalar point resequencing.

Word	Name	Type	Description
1	ID	I	Grid or scalar point identification number
2	SEQID	I	Sequenced identification number

New RECORD – SPCADD(5491,59,13)

Single-point constraint set combination.

Word	Name	Type	Description
1	SID	I	Set identification number
2	S	I	Single-point constraint set identification number
Word 2 repeats until -1 occurs			

New RECORD – TEMPD(5641,65,98)

Grid point temperature field default.

Word	Name	Type	Description
1	SID	I	Temperature set identification number
2	T	RS	Temperature

CASECC

Case control information

Updates for Record – REPEAT

Word	Name	Type	Description
....			
304	PANCON	I	Panel contributions set (PANCON)
305	PCMEDIA	I	Panel contributions media (PANCON)
306	PCFMT	I	Panel contributions format (PANCON)
307	PCFORM	I	Panel contributions form (PANCON)
308	PCTOPP	I	Panel contributions TOPP (PANCON)
309	PCTOPG	I	Panel contributions TOPG (PANCON)
310	PCSOL	I	Panel contributions SOLUTION (PANCON)
311	PCPAN	I	Panel contributions PANEL (PANCON)
312	PCGRID	I	Panel contributions GRID (PANCON)
313	MODSLF	I	Mode selection set (fluid)
....			
317	A2GG(2)	CHAR4	Name of direct input (g-set) acoustic coupling matrix (A2GG)
....			
333	UNDEF	None	
334	NA2GG	I	Internal set id for A2GG

Word	Name	Type	Description
....			
392	EXSEDMFX(2)	CHAR4	External Superelement DMIGSFIX String
394	NSMID	I	Non-Structural Mass Set ID
395	UNDEF(2)	None	
397	OP4UNIT	I	MBDEXPORT OP4 logical unit number
398	UNDEF	None	
399	CHECK	I	ADAMSMNF/MBDEXPORT CHECK flag
400	ADMOUT	I	ADAMSMNF ADMOUT flag/MBDEXPORT RECVROP2 flag
401	FLEXBODY	I	ADAMSMNF/MBDEXPORT FLEXBODY flag
402	FLEXONLY	I	ADAMSMNF/MBDEXPORT FLEXONLY flag
403	MINVAR	I	ADAMSMNF/MBDEXPORT MINVAR parameter
404	PSETID	I	ADAMSMNF/MBDEXPORT PSETID parameter
405	OUTGSTRS	I	ADAMSMNF/MBDEXPORT OUTGSTRS flag
406	OUTGSTRN	I	ADAMSMNF/MBDEXPORT OUTGSTRN flag
407	UNDEF(6)	None	
....			

Word	Name	Type	Description
430	MCSOL	I	Modal contributions SOLUTION (MODCOM)
431	MCPAN	I	Modal contributions PANELMC (MODCOM)
....			
442	MCTOPF	I	Modal contributions TOPF (MODCON)
....			
445	MODSEL	I	Mode selection set (structural)
....			
455	UNDEF(14)	None	
469	MCTOPS	I	Modal contributions TOPS (MODCON)
....			
475	FK2PP	I	Internal set id for K2PP scale factor
476	FM2PP	I	Internal set id for M2PP scale factor
477	FB2PP	I	Internal set id for B2PP scale factor
478	FK2GG	I	Internal set id for K2GG scale factor
479	FM2GG	I	Internal set id for M2GG scale factor
480	FB2GG	I	Internal set id for B2GG scale factor

Word	Name	Type	Description
481	FK42GG	I	Internal set id for K42GG scale factor
482	FP2G	I	Internal set id for P2G scale factor
483	FA2GG	I	Internal set id for A2GG scale factor
484	GPRSORT	I	Global ply results sorted with global ply ID numbers
485	EFLOAD1	I	External field load orientation
486	EFLOAD2	I	External field coordinate system
487	UNDEF(113)	None	
....			

EPT

Element property table.

Updated 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	

New Record – PPLANE(3801,38,978)

Word	Name	Type	Description
1	PID	I	Property identification number
2	MID	I	Material identification number
3	T	RS	Default membrane thickness for Ti on the connection entry
4	NSM	RS	Nonstructural mass per unit area
5	UNDEF(4)	None	

EPT705

Element property table (Pre-MSC Nastran 2001).

New RECORD – NSM(3201,32,991)

Defines the properties of a nonstructural mass.

Word	Name	Type	Description
1	SID	I	Set identification number
2	PROP(2)	CHAR4	Set of properties or elements
4	ORIGIN	I	Entry origin
5	ID	I	Property or element identification number
6	VALUE	RS	Nonstructural mass value
Words 5 through 6 repeat until End of Record			

New RECORD – NSM1(3301,33,992)

Defines the properties of a nonstructural mass.

Word	Name	Type	Description
1	SID	I	Set identification number
2	PROP(2)	CHAR4	Set of properties or elements
4	ORIGIN	I	Entry origin
5	VALUE	RS	Nonstructural mass value
6	SPECOPT	I	Specification option
SPECOPT=1		By IDs	
7	ID	I	
Word 7 repeats until End of Record			
SPECOPT=2		All	
7	ALL(2)	CHAR4	
Words 7 and 8 repeat until End of Record			
SPECOPT=3		Thru range	
7	ID	I	
8	THRU(2)	CHAR4	
10	ID	I	
Words 7 through 10 repeat until End of Record			
SPECOPT=4		Thru range with by	
7	ID	I	
8	THRU(2)	CHAR4	
10	ID	I	
11	BY(2)	CHAR4	
13	N	I	

Word	Name	Type	Description
Words 7 through 13 repeat until End of Record			
End SPECOPT			

New RECORD – NSMADD(3401,34,993)

Combines the nonstructural mass inputs.

Word	Name	Type	Description
1	SID	I	Set identification number
2	ID	I	Set of properties or elements
Word 2 repeats until End of Record			

New RECORD – NSML(3501,35,994)

Defines a set of lumped nonstructural mass by ID.

Word	Name	Type	Description
1	SID	I	Set identification number
2	PROP(2)	CHAR4	Set of properties or elements
4	ID	I	Property of element identification number
5	VALUE	RS	Lumped nonstructural mass value
Words 4 and 5 repeat until -1 occurs			

New RECORD – NSML1(3701,37,995)

Alternate form of NSML entry. Defines lumped nonstructural mass entries by VALUE, ID list.

Word	Name	Type	Description
1	SID	I	Set identification number

Word	Name	Type	Description
2	PROP(2)	CHAR4	Set of properties or elements
4	VALUE	RS	Lumped nonstructural mass value
5	SPECOPT	I	Specification option
SPECOPT=1		By IDs	
6	ID	I	Property of element identification number
Word 6 repeats until -1 occurs			
SPECOPT=2		All	
6	ALL(2)	CHAR4	Keyword ALL
Words 6 and 7 repeat until -1 occurs			
SPECOPT=3		Thru range	
6	ID1	I	Starting identification number
7	THRU(2)	CHAR4	Keyword THRU
9	ID2	I	Ending identification number
Words 6 through 9 repeat until -1 occurs			
SPECOPT=4		Thru range with by	
6	ID1	I	Starting identification number
7	THRU(2)	CHAR4	Keyword THRU
9	ID2	I	Ending identification number
10	BY(2)	CHAR4	Keyword BY
12	N	I	Increment
Words 6 through 12 repeat until -1 occurs			

New RECORD – PCOMPG(15006,150,604)

Word	Name	Type	Description
1	PID	I	Property identification number
2	LAMOPT	I	Laminate option
3	Z0	RS	Distance from the reference plane to the bottom surface
4	NSM	RS	Nonstructural mass per unit area
5	SB	RS	Allowable shear stress of the bonding material
6	FT	I	Failure theory
7	TREF	RS	Reference temperature
8	GE	RS	Damping coefficient
9	GPLYIDi	I	Global ply IDs.
10	MID	I	Material identification number
11	T	RS	Thicknesses of the ply
12	THETA	RS	Orientation angle of the longitudinal direction of the ply
13	SOUT	I	Stress or strain output request of the ply
Words 9 through 13 repeat N times			

New RECORD – PFAST(3601,36,55)

Word	Name	Type	Description
1	PID	I	Property identification number
2	D	RS	Diameter of the spot weld
3	MCID	I	Element stiffness coordinate system

Word	Name	Type	Description
4	MFLAG	I	Defines MCID as absolute or relative
5-7	KT(3)	RS	Translational stiffness
8-10	KR(3)	RS	Rotational stiffness
11	MASS	RS	Lumped mass
12	GE	RS	Structural damping

New RECORD – PPLANE(3801,38,978)

Word	Name	Type	Description
1	PID	I	Property identification number
2	MID	I	Material identification number
3	T	RS	Default membrane thickness for Ti on the connection entry
4	NSM	RS	Nonstructural mass per unit area
5	UNDEF(4)	None	

GEOM168

Table of Bulk Data entry images related to geometry (Pre-MSC Nastran Version 69).

Updated Record 18 – SEBULK(1427,14,465)

Word	Name	Type	Description
1	SEID	I	Superelement identification number
2	TYPE	I	Superelement type
3	RSEID	I	Reference superelement identification number

Word	Name	Type	Description
4	METHOD	I	Boundary point search method: 1=automatic or 2=manual
5	TOL	RS	Location tolerance
6	LOC	I	Coincident location check option: yes=1 or no=2
7	MEDIA	I	Media format of boundary data of external SE
8	UNIT	I	FORTRAN unit number of OP2 and OP4 input of external SE

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 – CPLSTN3(8801,88,980)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(3)	I	Grid point identification numbers of connection points
6	THETA	RS	Material property orientation angle or coordinate system ID
7	UNDEF(10)	None	

New Record – CPLSTN4(9301,93,981)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(4)	I	Grid point identification numbers of connection points
7	THETA	RS	Material property orientation angle or coordinate system ID
8	UNDEF(9)	None	

New Record – CPLSTN6(9401,94,982)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(6)	I	Grid point identification numbers of connection points
9	THETA	RS	Material property orientation angle or coordinate system ID
10	UNDEF(7)	None	

New Record – CPLSTN8(9501,95,983)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(8)	I	Grid point identification numbers of connection points

Word	Name	Type	Description
11	THETA	RS	Material property orientation angle or coordinate system ID
12	UNDEF(5)	None	

New Record – CPLSTS3(9601,96,984)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(3)	I	Grid point identification numbers of connection points
6	UNDEF	None	
7	THETA	RS	Material property orientation angle or coordinate system ID
8	UNDEF(4)	None	
12	TFLAG	I	Flag signifying meaning of T(3) values
13	T(3)	RS	Membrane thickness of element at grid points
16	UNDEF	None	

New Record – CPLSTS4(8401,84,985)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(4)	I	Grid point identification numbers of connection points

Word	Name	Type	Description
7	THETA	RS	Material property orientation angle or coordinate system ID
8	UNDEF(4)	None	
12	TFLAG	I	Flag signifying meaning of T(4) values
13	T(4)	RS	Membrane thickness of element at grid points

New Record – CPLSTS6(9801,98,986)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(6)	I	Grid point identification numbers of connection points
9	UNDEF(2)	None	
11	THETA	RS	Material property orientation angle or coordinate system ID
12	TFLAG	I	Flag signifying meaning of T(3) values
13	T(3)	RS	Membrane thickness of element at grid points
16	UNDEF	None	

New Record – CPLSTS8(9901,99,987)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number

Word	Name	Type	Description
3	G(8)	I	Grid point identification numbers of connection points
11	THETA	RS	Material property orientation angle or coordinate system ID
12	TFLAG	I	Flag signifying meaning of T(4) values
13	T(4)	RS	Membrane thickness of element at grid points
17	UNDEF(8)	None	

New Record – CPYRA5FD(25700,257,9948)

Same as record CPYRAM description.

New Record – CPYRA13F(25800,258,9947)

Same as record CPYRAM description.

New Record – CPYRAMPR(7909,79,9946)

Word	Name	Type	Description
1	EID	I	Element identification number
2	PID	I	Property identification number
3	G(13)	I	Grid point identification numbers of connection points
16	UNDEF	None	

GEOM3

Table of Bulk Data entry images related to static and thermal loads.

New Record – BOLTLD(7601,76,577)

Word	Name	Type	Description
1	SID	I	Set identification number of BOLTLD bulk entry
2	S	RS	Overall scale factor
3	Si	RS	Scale factor on Li
4	Li	I	Set identification number of BOLTFOR bulk entries
Word 4 repeats until (-1,-1) occurs			

GEOM4705

Table of Bulk Data entry images related to constraints (Pre-MSC Nastran 2001).

Table of Bulk Data entry images related to constraints, degree-of-freedom membership and rigid element connectivity.

New Record – BNDFIX(110,1,584)

Word	Name	Type	Description
1	ID	I	Grid or scalar point identification number
2	C	I	Component numbers

New Record – BNDFIX1(210,2,585)

Word	Name	Type	Description
1	C	I	Component numbers
2	THRUFLAG	I	Thru range flag
THRUFLAG=0		No	
3	ID	I	Grid or scalar point identification number

Word	Name	Type	Description
Word 3 repeats until End of Record			
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			

New Record – BNDFREE(310,3,586)

Word	Name	Type	Description
1	ID	I	Grid or scalar point identification number
2	C	I	Component numbers

New Record – BNDFREE1(410,4,587)

Word	Name	Type	Description
1	C	I	Component numbers
2	THRUFLAG	I	Thru range flag
THRUFLAG=0		No	
3	ID	I	Grid or scalar point identification number
Word 3 repeats until End of Record			
THRUFLAG=1		Yes	
3	ID1	I	First grid or scalar point identification number
4	ID2	I	Second grid or scalar point identification number

Word	Name	Type	Description
End THRUFLAG			

New Record – RWELD(11901,119,561)

Word	Name	Type	Description
1	EID	I	Element ID
2	GA	I	Grid ID of GA
3	TYPE	I	Type of shell element
4	GI(8)	I	Grid IDs of shell element
12	GS	I	Grid ID of GS

MPT

Table of Bulk Data entry images related to material properties.

New Record – MAT11(2903,29,371)

Solid element orthotropic material property definition.

Defines the material properties for a 3-D orthotropic material for isoparametric solid elements.

Word	Name	Type	Description
1	MID	I	Material identification number
2	E1	RS	Modulus of elasticity in the longitudinal direction or 1-direction
3	E2	RS	Modulus of elasticity in the lateral direction or 2-direction
4	E3	RS	Modulus of elasticity in the thickness direction or 3-direction
5	NU12	RS	Poisson's ratio ($\epsilon 2/\epsilon 1$ for uniaxial loading in the 1-direction)

Word	Name	Type	Description
6	NU13	RS	Poisson's ratio ($\epsilon3/\epsilon1$ for uniaxial loading in the 1-direction)
7	NU23	RS	Poisson's ratio ($\epsilon3/\epsilon2$ for uniaxial loading in the 2-direction)
8	G12	RS	In-plane shear modulus
9	G13	RS	Transverse shear modulus for shear in the 1–3 plane
10	G23	RS	Transverse shear modulus for shear in the 2–3 plane
11	RHO	RS	Mass density
12	A1	RS	Thermal expansion coefficient in the longitudinal direction
13	A2	RS	Thermal expansion coefficient in the lateral direction
14	A3	RS	Thermal expansion coefficient in the thickness direction
15	TREF	RS	Reference temperature for calculation of thermal loads
16	GE	RS	Structural damping coefficient
17	UNDEF(16)	None	

New Record – MATT11(3303,33,988)

Solid orthotropic material temperature dependence.

Defines the temperature dependent material property for a 3-D orthotropic material for isoparametric solid elements.

Word	Name	Type	Description
1	MID	I	Material identification number

Word	Name	Type	Description
2	TE1	I	TABLEMi ID for modulus of elasticity in the 1-direction
3	TE2	I	TABLEMi ID for modulus of elasticity in the 2-direction
4	TE3	I	TABLEMi ID for modulus of elasticity in the 3-direction
5	TNU12	I	TABLEMi ID for Poisson's ratio ($\epsilon 2/\epsilon 1$ for uniaxial loading in the 1-direction)
6	TNU13	I	TABLEMi ID for Poisson's ratio ($\epsilon 3/\epsilon 1$ for uniaxial loading in the 1-direction)
7	TNU23	I	TABLEMi ID for Poisson's ratio ($\epsilon 3/\epsilon 2$ for uniaxial loading in the 2-direction)
8	TRHO	I	TABLEMi ID for mass density
9	TG12	I	TABLEMi ID for shear modulus in 1–2 plane
10	TG13	I	TABLEMi ID for shear modulus in 1–3 plane
11	TG23	I	TABLEMi ID for shear modulus in 2–3 plane
12	TA1	I	TABLEMi ID for thermal expansion coefficient in the 1-direction
13	TA2	I	TABLEMi ID for thermal expansion coefficient in the 2-direction
14	TA3	I	TABLEMi ID for thermal expansion coefficient in the 3-direction
15	UNDEF	None	
16	TGE	RS	TABLEMi ID for structural damping coefficient
17	UNDEF(16)	None	

OEF

Table of element forces

Also contains composite failure indices and analysis types (real and complex), and SORT1 and SORT2 formats.

Updated Record - DATA

Word	Name	Type	Description
....			
ELTYPE =79		Acoustic velocity/pressures in five-sided solid element (CPYRAM)	
2	ELNAME(2)	CHAR4	Element name: “PYRAMPR”
TCODE,7 = 0 or 2		Real or Random Response	
4	AX	RS	Acceleration in x
5	AY	RS	Acceleration in y
6	AZ	RS	Acceleration in z
7	VX	RS	Velocity in x
8	VY	RS	Velocity in y
9	VZ	RS	Velocity in z
10	PRESSURE	RS	Pressure in dB
TCODE,7 = 1			
4	AXR	RS	Acceleration in x – real/mag. part
5	AYR	RS	Acceleration in y – real/mag. part
6	AZR	RS	Acceleration in z – real/mag. part
7	VXR	RS	Velocity in x – real/mag. part
8	VYR	RS	Velocity in y– real/mag. part
9	VZR	RS	Velocity in z– real/mag. part

Word	Name	Type	Description
10	PRESSURE	RS	Pressure in dB
11	AXI	RS	Acceleration in x – imag./phase part
12	AYI	RS	Acceleration in y – imag./phase part
13	AZI	RS	Acceleration in z – imag./phase part
14	VXI	RS	Velocity in x – imag./phase part
15	VYI	RS	Velocity in y – imag./phase part
16	VZI	RS	Velocity in z – imag./phase part
End TCODE,7			
....			

OES

Table of element stresses or strains — For all analysis types (real and complex) and SORT1 and SORT2 formats.

Record 2 - DATA

New ELTYPEs 271-278, 281, 283–285, 287-298

ELTYPE =271		Triangle plane strain (CPLSTN3) – Center	
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

ELTYPE =272	Quadrilateral plane strain (CPLSTN4) – Center and Corners

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			

ELTYPE =273		Triangle plane strain (CPLSTN6) – Center and Corners	
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			

ELTYPE =274		Quadrilateral plane strain (CPLSTN8) – Center and Corners	
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			

ELTYPE =275		Triangle plane stress (CPLSTS3) – Center	
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

ELTYPE =276		Quadrilateral plane stress (CPLSTS4) – Center and Corners	
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

ELTYPE =277		Triangle plane stress (CPLSTS6) – Center and Corners	
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			

ELTYPE =278		Quadrilateral plane stress (CPLSTS8) – Center and Corners	
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			

ELTYPE =281		Triangle plane strain (CPLSTN3) – Nonlinear format – Center	
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	ES	RS	Equivalent stress
7	EPS	RS	Effective plastic/inelastic strain
8	ECS	RS	Effective creep strain
9	EX	RS	Strain in x
10	EY	RS	Strain in y
11	EZ	RS	Strain in z
12	ETXZ	RS	Shear strain in xz

ELTYPE =283		Triangle plane strain (CPLSTN6) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain

10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 4 times			

ELTYPE =284		Quadrilateral plane strain (CPLSTN8) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain
10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 5 times			

ELTYPE =285		Triangle plane stress (CPLSTS3) – Nonlinear format – Center	
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	ES	RS	Equivalent stress
7	EPS	RS	Effective plastic/inelastic strain
8	ECS	RS	Effective creep strain
9	EX	RS	Strain in x
10	EY	RS	Strain in y
11	EZ	RS	Strain in z
12	ETXZ	RS	Shear strain in xz

ELTYPE =287		Triangle plane stress (CPLSTS6) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain

10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 4 times			

ELTYPE =288		Quadrilateral plane stress (CPLSTS8) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain
10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 5 times			

ELTYPE =289		Quadrilateral plane strain (CPLSTN4) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain
10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 5 times			

ELTYPE =290		Quadrilateral plane stress (CPLSTS4) – Nonlinear format – Center and Corners	
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	ES	RS	Equivalent stress
9	EPS	RS	Effective plastic/inelastic strain
10	ECS	RS	Effective creep strain
11	EX	RS	Strain in x
12	EY	RS	Strain in y
13	EZ	RS	Strain in z
14	ETXZ	RS	Shear strain in xz
Words 3 through 14 repeat 5 times			

ELTYPE =291		Triangle plane strain (CPLSTN3) – Hyperelastic - Grid	
2	TERM	CHAR4	“GRID”
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz

Words 3 through 13 repeat 3 times

ELTYPE =292		Quadrilateral plane strain (CPLSTN4) – Hyperelastic - Grid	
2	TERM	CHAR4	"GRID"
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz
Words 3 through 13 repeat 4 times			

ELTYPE =293		Triangle plane strain (CPLSTN6) – Hyperelastic - Grid	
2	TERM	CHAR4	"GRID"
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz
Words 3 through 13 repeat 3 times			

ELTYPE =294		Quadrilateral plane strain (CPLSTN8) – Hyperelastic - Grid	
2	TERM	CHAR4	“GRID”
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz

Words 3 through 13 repeat 4 times

ELTYPE =295		Triangle plane stress (CPLSTS3) – Hyperelastic - Grid	
2	TERM	CHAR4	“GRID”
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz
Words 3 through 13 repeat 3 times			

ELTYPE =296		Quadrilateral plane stress (CPLSTS4) – Hyperelastic - Grid	
2	TERM	CHAR4	“GRID”
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz
Words 3 through 13 repeat 4 times			

ELTYPE =297		Triangle plane stress (CPLSTS6) – Hyperelastic - Grid	
2	TERM	CHAR4	“GRID”
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz

Words 3 through 13 repeat 3 times

ELTYPE =298		Quadrilateral plane stress (CPLSTS8) – Hyperelastic - Grid	
2	TERM	CHAR4	"GRID"
3	ID	I	Point ID
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	PRESSURE	RS	Pressure
9	VOLSTR	RS	Volume strain
10	EX	RS	Strain in x
11	EY	RS	Strain in y
12	EZ	RS	Strain in z
13	ETXZ	RS	Shear strain in xz
Words 3 through 13 repeat 4 times			

Word	Name	Type	Description
End ELTYPE			

New datablocks

OUGGC

Table of grid contributions.

For frequency analysis types (complex), and SORT1 and SORT2 formats.

Record 0 - 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 1 - IDENT

Word	Name	Type	Description
1	ACODE(C)	I	Device code + 10*Approach Code
2	TCODE(C)	I	Table Code
3	GCODE	I	Grid 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	GID	I	Grid ID (0 for TOTAL)
7	UNDEF	None	

Word	Name	Type	Description
End TCODE,1			
8	DATTYP	I	Data Type (1=pressure, 2=first derivative, 3=second derivative)
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
83	SUBTITL(32)	CHAR4	Subtitle
115	LABEL(32)	CHAR4	Label

Record 2 - DATA

Word	Name	Type	Description
TCODE,1=01		Sort 1	
1	GID	I	Grid ID (0 for TOTAL)
2	UNDEF	None	
TCODE,1=02		Sort 2	
1	FREQ	RS	Frequency (Hz)
2	UNDEF	None	
End TCODE,1			
TCODE,2=01		Pressure	
TCODE,7=01		Real/Imaginary	
3	GCR	RS	Grid contribution – real part
4	GCI	RS	Grid contribution – imaginary part

Word	Name	Type	Description
End TCODE,7			

Word	Name	Type	Description
TCODE,2=10		First derivative of pressure	
TCODE,7=01		Real/Imaginary	
3	GCR	RS	Grid contribution – real part
4	GCI	RS	Grid contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2=11		Second derivative of pressure	
TCODE,7=01		Real/Imaginary	
3	GCR	RS	Grid contribution – real part
4	GCI	RS	Grid contribution – imaginary part
End TCODE,7			

Record 3 - TRAILER

Word	Name	Type	Description
1	NREC	I	Number of records
2	UNDEF(5)	None	

OUGPC

Table of panel contributions.

For frequency analysis types (complex), and SORT1 and SORT2 formats.

Record 0 - 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 1 - IDENT

Word	Name	Type	Description
1	ACODE(C)	I	Device code + 10*Approach Code
2	TCODE(C)	I	Table Code
3	PCODE	I	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			

Word	Name	Type	Description
8	DATTYP	I	Data Type (1=pressure, 2=first derivative, 3=second derivative)
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
83	SUBTITL(32)	CHAR4	Subtitle
115	LABEL(32)	CHAR4	Label

Record 2 - 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		Pressure	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Panel contribution – real part
4	PCI	RS	Panel contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2=10		First derivative of pressure	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Panel contribution – real part
4	PCI	RS	Panel contribution – imaginary part
End TCODE,7			

Word	Name	Type	Description
TCODE,2=11		Second derivative of pressure	
TCODE,7=01		Real/Imaginary	
3	PCR	RS	Panel contribution – real part
4	PCI	RS	Panel contribution – imaginary part
End TCODE,7			

Record 3 - TRAILER

Word	Name	Type	Description
1	NREC	I	Number of records
2	UNDEF(5)	None	

Chapter

15 NX NASTRAN 7.1 – problem report (PR) fixes

PR#	Problem Reported	Problem Originated	Description
1701255	V6.0	V6.0	A model produces contact and glue refinement failures and performance issues.
1708954			Same as PR#1701255.
1714716			Same as PR#1701255.
1735610	V6.1	–	Unable to specify redundant license server using a comma separated list.
1744990	V6.1	V6.1	Using SOL 601, the 3D iterative solver fails during a linear static analysis using multiple time steps.
1750083	V7.0	V6.1	SOL 601 lists numerous Jacobian errors for a single element, but does not list all the elements that failed.
1751390	V6.1	V6.1	License borrowing is not fully functional. For older versions, the work around points to the local copy of the license file on the disconnected client.
1756907	V7.0	V4.1	During a linear contact analysis, CGAP incorrectly uses separation data from the INIPENE parameter of the BCTPARM card.
1760330	V6.1	V4.0	The msrmode=2 option of the ADMRECVR case control command is not working properly causing the ADMRECVR case control command to solve a much larger problem than necessary.
1767317	V7.0	V7.0	A SOL 601 run solved to only 98% of the applied load with AUTO=3 or 4 and bolt preload.

1769119	V6.1	V6.1	A SOL 601 run fails using 64-bit NX Nastran 6.1. The run does not fail using 32-bit NX Nastran 6.1.
1770633	V7.0	V4.0	Runs having a large number of active DRESP2 based constraints resulting from a large number of specified DRESP2 based constraints and a loose DSCREEN requirement fail due to insufficient local memory.
1774623	V6.1	V1.0	Random analysis in SOL 111 failed on 64-bit platforms because of a memory coding error.
1777383	V7.0	V7.0	Runs fail due to incorrect partitioning with DMP RDMODES.
1777426	V7.0	V7.0	The MAXRATIO message is misleading/wrong in DMP RDMODES.
1780451	V7.0	V3.0	SOL 601 may crash when BUFFSIZE > 8193 with a large model.
1784580	V7.0	V1.0	A run failed on Linux and reported error message 3008. The work around is to specify less than 7.9GB.
2135045	V6.1	V6.1	A SOL 601 run containing 8-node composite shell elements and stress output requests failed.
2136995	V6.1	V4.0	With only the SDISP case control output request, sparse data recovery was incorrectly turned off, resulting in an excessively large problem solution.
2137567	V6.1	V6.1	For a conical shell, the azimuthal locations for stress output are limited to four places.
2138258	V6.1	V6.1	When the RMAXMIN option NPAVG is set greater than 1, the disk space required to solve NX Nastran 6.1 and NX Nastran 7 models becomes quite large. If the problem size is large, the disk requirement can exceed that available on even large systems.
2141050	V7.0	V6.1	Columns greater than 16777217 (2^24) are not included in the OUTPUT4 file.
2141142	V6.1	V6.1	The .op2 file is not converted from big endian to little endian correctly when the integer IDs are greater than 2^23.
2143793	V7.0	V7.0	During SOL 111 restart runs, geometry data blocks are not written to the .op2 file.

2145154	P7.0	V2.0	Runs using the element iterative solver produce incorrect results when multiple subcases with multiple GRAV entries are present and the first GRAV entry is used in all subcases.
6161112	V6.1	V4.0	The error message prompted when contact is not found in a linear contact analysis is inadequate.
6202164			Same as PR#1751390.
6205065			Same as PR#1701255.
6205381	V7.0	V1.0	Jobs including both RMS responses and other types of frequency responses like FR and PSD are likely to generate incorrect sensitivity analysis results for some responses under certain conditions.
6210057	V7.0	V6.1	SOL 601 does not support variable node pyramid and tetra elements in surface contact.
6211625	V7.0	V1.0	For solves that include superelements, non-existent referenced DRESPi are not trapped and related constraints are ignored.
6219609	V6.1	V6.1	An invalid warning message that MATHP does not support TABi and TABD occurs.
6225650			Same as PR#1751390.
6229910			Same as PR#6161112.
6241419	V6.1	V1.0	The existence of a comma at the end of a SET case control command is not recognized, which results in the next case control command being ignored. The comma at the end of a SET case control command is now trapped as a syntax error.
6244819	V7.0	V5.1	OLOAD results are not written to an .op2 file for a subcase that includes stress stiffening and bolt preload conditions.
6247629	V7.0	V6.1	A SOL 601 solve may hang when rigid links and BGSET are present.
6250642	V7.0	V7.0	AUTOMPC fails when the GN grid of an RBE2 is the only grid in the O-set.
6252275	V7.0	V7.0	Glue separation distance messages are sometimes reported incorrectly. A new reporting method has been created which only reports the ten largest values for each pair.

6254705	V7.0	V7.0	A SOL 110 rotor dynamics solve creates an empty .csv file when using the new mode tracking algorithm and the user requests Campbell diagram data.
6255599	V7.0	V7.0	Using MFLUID with VMOPT=2 produces the incorrect modal effective mass.
6261459	P7.1	V7.1	A memory error occurs on multiple subcases. The pipe opened for executing the command to get memory is not being closed.
6263081	V7.0	V1.0	During a SOL 111 run, the sorting/look-up code caused a failure and issued warning message 979 when processing all of the XYPEAK output requests.
6264879	V6.0	V1.0	When using SORT2, OLOAD results are not written into an .op2 file.
6268819	V7.0	V7.0	Incorrect results are produced by a linear contact analysis that uses the iterative solver when all bodies are restrained to prevent rigid body motion and contact does not exist between any of the bodies.
6269008	V6.0	–	A SOL 601 solve fails when the number of PELAS elements exceeds 9999.
6272973	V7.0	V1.0	For certain circumstances when insufficient memory is available, a run can produce corrupt element stress tables without warning. A FATAL error is now issued when insufficient memory is available for stress table processing.
6274156	V7.0	V7.0	A model containing glue does not produce six rigid body modes.
6274522	V7.0	V1.0	Direct frequency analysis (SOL 108) fails to ignore FREQ3/FREQ4 bulk entries as stated in the Quick Reference Guide. These bulk entries are intended to only be valid for modal frequency analyses (SOL 111, 146, and 200 (MFREQ)).
6281779	V7.0	V1.0	When solving a random response solution using mem=estimate, the run fails and prompts system fatal message 3007 (DDRMM). The message has now been enhanced to inform the user how to better allocate memory.

6282350	V7.0	V1.0	A SOL 106 run produces system fatal message 3002 and does not write results to the .op2 file. The problem has been traced to the load factor for subcase 2 exceeding 1.0. This causes SUBCASE 2 to be solved twice. Resolving SUBCASE 2 produces too many results and causes the end of file problem.
6282820	V7.0	V3.0	When a model is defined using connection dof, the modes in the .op2 file differ from the modes in the RFI file.
6284706	V7.0	V1.0	A SOL 106 run produces error message 3204 unexpectedly. The following information has been added to the error message: "This message will also be output if the case control LOAD card references only SPCD cards. If this is the case, this message can be ignored as there are no follower forces associated with SPCD loading."
6285173	V7.0	V6.1	When using SOL 601 with element birth and death specified for CDAMP1 or CDAMP2, all CDAMP1 and CDAMP2 elements with the same PID are mistakenly removed.
6286416	V7.0	V1.0	The virtual mass defined with MFLUID produces results that are not independent of element grid ordering.
6292253	V6.1	V1.0	Strains for CTRIA6 elements are reported incorrectly when PARAM, OMID,YES is used to print results in the material coordinate system.
6293567	V7.0	V6.0	When generating element forces using EXTOUT, error message 3008 incorrectly occurs.
6298909	V7.0	V7.0	BCPROP does not trap non-existent PIDs causing an error to occur later in the run.
6299413	V6.1	V6.1	A SOL 601,129 large displacement analysis does not converge because the CBUSH1D node-to-node damper effect is incorrect.
6302615	V7.0	V6.1	In a SOL 601, 106 run, reaction forces are missing when the grids are part of a 2D contact region and there are constraints on the grids.

6303510	V7.0	V7.0	Several additional data items were added to the DDAM input file for NX Nastran 7. There was a problem reading old files input without the additional entries present. This has been corrected and the additional data entries do not need to present. If they have not been entered, a reasonable default will be used.
6305946	V7.0	V5.0	An MFLUID enclosed volume fatal error occurs when a fluid cavity (void) is modeled in an infinite fluid and the VMOPT=2 option is used.
6306164	V7.0	V7.0	Processing of U6 dof (defined via USET/USET1 bulk data) fails when generating residual vectors.
6313421	P7.1	V1.0	MCEIG analysis fails when earlier subcases relating to eigenvalue analysis (such as frequency response) are present.
6314316	V7.0	V1.0	The combination of MFLUID with the SPCD enforced motion capability works incorrectly. This has been corrected for the constraint mode method of enforced motion (SYSTEM(422) / ENFMOTN = 0, the default).
6317037			Same as PR#6303510.
6319514			Same as PR#1751390.
6321451			Same as PR#6332167.
6322247	V7.0	V1.0	CHBDYE end area SIDE selection on CTUBE is unknown. The error occurs when the SIDE field on the CHBDYE bulk entry is 3 or 4 (indicating the point sides).
6332135	P7.1	V7.0	Grounding problems occur when overlapping shells are glued together with no offsets.
6332167	P7.1	V7.0	A contact analysis model containing acceleration loads fails when using the element iterative solver.
6336763	P7.1	V7.0	In SOL 111, element forces for CDAMPi and CVISC elements are calculated incorrectly.
6338526	P7.1	V7.0	Windows 64 fails during a SOL 112 restart run because of a pivot ratio error.

6344799	V7.0	V7.0	Certain optimization problem types fail on the Windows 64 platform. The ilp64 version of the Windows 64 platform and other platforms do not have this problem.
6350305	V7.0	V5.0	A SOL 112 performance/disk usage error occurs when generating the right-hand side constraint forces for SPCD enforced motion and calculating SPC forces.
6357654	V7.0	V3.0	A SOL 601 run gives incorrect results when PCOMP has only one layer.
6358791	V7.0	V1.0	A run produces fatal error 9056. The error message is not adequately descriptive. The error message has been updated to: "This error might also be caused by no residual subcase specified in a superelement model".

Chapter 16 System description summary

Beginning with this release, the list of supported systems is included in the **README.txt** file located in the NX Nastran installation under the *nxn7p1* directory.