NX Nastran 8.5.1 Release Guide

Contents

Proprietary & Restricted Rights Notice

© 2013 Siemens Product Lifecycle Management Software Inc. All Rights Reserved.

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

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

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

All other trademarks are the property of their respective owners.

TAUCS Copyright and License

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

TAUCS License:

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

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

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

Availability (TAUCS)

Proprietary & Restricted Rights Notice

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

Chapter

1 Dynamics

Residual vectors

The modal method is a common approach for dynamic forced response solutions because of the computational efficiencies gained by the normal mode reduction. In practice, a small subset of the modes are computed and used for the computation while the effects of truncated higher frequency modes are ignored. The typical rule of thumb for modal sufficiency is to compute modes up to a frequency twice as high as the excitation frequency range. This is suitable for most responses, but some responses, in particular force and stress, can have errors because of the missing truncated modes. Since the missing modes have high frequencies, their omission leads to errors in the static response contributions. Residual vectors are an effective means for reducing the response errors by adding the missing static flexibility from the truncated modes.

Residual vector control was enhanced in NX Nastran 8 with the introduction of the RESVEC case control command, then enhanced further in NX Nastran 8.5 with additional options.

To streamline the use of the RESVEC case control command, the default settings are changing.

New RESVEC Defaults *

- The defaults now apply even when the RESVEC case control command is undefined. Previously the RESVEC command was required to request residual vectors.
- For SOLs 103, 106 (PARAM,NMLOOP,0), 110, 115, 153 (PARAM,NMLOOP,0), and 187, the new default is RESVEC=COMPONENT.
- For SOLs 111, 112, 118, 146, and 200 (ANALYSIS=MODES, MCEIG, MTRAN, or MFREQ), the default is RESVEC=YES.

In addition, a new DAMPLOD describer requests residual vectors by default for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements.

Note that the existing RVEL describer requests residual vectors for the degree-of-freedom associated with the CBUSHi, CDAMPi, CELASi, and CVISC elements selected on a RVEL bulk entry, regardless if damping is defined or not. If DAMPLOD and RVEL are both defined (default), and RVEL bulk data entries are defined, residual vectors will be requested for the combined degree-of-freedom list.

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

* In order to transition to the new defaults, for this release only, you must remove the parameter settings PARAM,RESVEC,NO and PARAM,RESVINER,NO from the NX Nastran 8.5.1 rc file (*installation_path*/conf/nast8.rcf). If you do not remove these settings, the default behavior is the same as previous releases.

RESVEC Residual Vector Request

Used to control the computation of residual vectors.

Format:

```
RESVEC [( [APPLOD] , [RVDOF ] , [RVEL  ] , [INRLOD] ,
          [NOAPPL]   [NORVDO]   [NORVEL]   [NOINRL]

          [DAMPLOD] , [DYNRSP  ] )] = { SYSTEM      }
          [NODAMP ]   [NODYNRSP]      { NOSYSTEM    }
                                      { COMPONENT   }
                                      { NOCOMPONENT }
                                      { BOTH or YES }
                                      { NO          }
```

Examples:

```
RESVEC = SYSTEM
RESVEC(NOINRL) = COMPONENT
RESVEC = YES
```

Describers:

Describer	Meaning
APPLOD	Compute residual vectors for applied loads. See Remark 5. (Default)
NOAPPL	Do not compute residual vectors for applied loads.
RVDOF	Compute residual vectors for degree-of-freedom included in RVDOF and RVDOF1 bulk entries. See Remark 6. (Default)
NORVDO	Do not compute residual vectors for degree-of-freedom included in RVDOF and RVDOF1 bulk entries.
RVEL	Compute residual vectors for degree-of-freedom defining CBUSHi, CDAMPi, CELASi, and CVISC elements referenced by a RVEL bulk entry. See Remarks 7 and 9. (Default)
NORVEL	Do not compute residual vectors for degree-of-freedom defining CBUSHi, CDAMPi, CELASi, and CVISC elements referenced by a RVEL bulk entry.

Describer	Meaning
INRLOD	Compute residual vectors for inertia relief load. See Remark 4. (Default)
NOINRL	Do not compute residual vectors for inertia relief load.
DAMPLOD	Compute residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements. See Remark 9. (Default)
NODAMP	Do not compute residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements.
DYNRSP	Include the dynamic effect of residual vector modes in the forced response solution. See Remark 8. (Default)
NODYNRSP	Do not include the dynamic effect of residual vector modes in the forced response solution.

The defaults for the following describers are solution dependent. The defaults apply even when the RESVEC case control entry is undefined. See Remark 10.

SYSTEM	Request residual vectors for residual structure degree-of-freedom only (a-set).
NOSYSTEM	Do not compute residual vectors for residual structure degree-of-freedom (a-set).
COMPONENT	Request residual vectors for superelement degree-of-freedom only (o-set). (Default for SOLs 103, 106 with PARAM, NMLOOP=0, 110, 115, 153 with PARAM, NMLOOP=0, and 187.)
NOCOMPONENT	Do not compute residual vectors for superelement degree-of-freedom (o-set).
BOTH (or YES)	Request residual vectors for both residual structure DOF (a-set) and for superelement DOF (o-set). (Default for SOLs 111, 112, 118, 146, and 200 with ANALYSIS=MTRAN or MFREQ.)
NO	Turns off calculation of residual vectors.

Remarks:

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

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

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

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

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

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

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

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

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

7. If a CBUSHi, CDAMPi, CELASi, and CVISC element is grounded, the unit load is applied to only the ungrounded end.

8. DYNRSP/NODYNRSP is only applicable to modal forced response solutions (SOLs 111, 112, 146, and 200). For models containing superelements, DYNRSP/NODYNRSP is only applicable to the residual structure.

9. DAMPLOD requests residual vectors for the degree-of-freedom in which viscous damping is defined on the CBUSHi, CDAMPi, and CVISC elements. RVEL requests residual vectors for the degree-of-freedom associated with the CBUSHi, CDAMPi, CELASi, and CVISC elements selected on a RVEL bulk entry, regardless if damping is defined or not. If DAMPLOD and RVEL are both defined (default), and RVEL bulk data entries are defined, residual vectors will be requested for the combined degree-of-freedom list.

10. For SOLs 111, 112, 118, 146, and 200 (ANALYSIS=MODES, MCEIG, MTRAN, or MFREQ), the default is RESVEC=YES. For SOLs 103, 106 (PARAM,NMLOOP,0), 110, 115, 153 (PARAM,NMLOOP,0), and 187, the default is RESVEC=COMPONENT. These defaults apply even when the RESVEC case control entry is undefined.

Structural damping improvement

Previously in dynamic solutions, when the software computed force and stress for the CELASi and CBUSH elements, the following software behavior existed.

- The structural damping GE defined on the element and property entries was included for force and stress recovery.

- The structural damping defined with PARAM,G was included for force and stress recovery on the CELAS2 and CBUSH elements, but not on the CELAS1, CELAS3, and CELAS4 elements.

- The structural damping GE and G were only included for force and stress recovery in the frequency response solutions, SOL 108 and 111. They were not included in the transient response solutions, SOL 109 and 112.

Beginning with NX Nastran 8.5.1, the force and stress recovery on CELASi and CBUSH elements includes both PARAM,G and GE, and is now supported in both frequency response and transient solutions. You can optionally include the new parameter setting PARAM,GDAMPF,0 to remove PARAM,G from the force and stress computation.

Note that in a transient solution, the parameter W3 must be defined to include PARAM,G, and the parameter W4 must be defined to include GE. See the NX Nastran Basic Dynamics Guide for details.

See the updated CBUSH, CELAS1, CELAS2, CELAS3, and CELAS4 bulk entries.

Mathematical description

The structural damping used in the CELASi and CBUSH element force and stress calculation, g_t, is now based on the global damping, g, and elemental damping, g_e, as follows:

When PARAM,GDAMPF,1 (default), $g_t = g + g_e$

When PARAM,GDAMPF,0, g_t = ge

Note that the CELAS4 entry does not include the GE field, and it does not reference a PELAS entry. As a result, g_e=0 for the CELAS4 element.

In the following equations:

F_e is a CELASi or CBUSH element force.

K_e is a CELASi or CBUSH element stiffness.

U_e and $\mathring{U}_e$ are the displacement and velocity response for a CELASi or CBUSH element.

B_e is the viscous damping optionally defined on a PBUSH entry.

w3 and w4 are parameter inputs used to convert structural damping to viscous damping in transient response solutions.

Force recovery for CBUSH.

- Transient response solutions 109 and 112:

$$F_e(t) = K_e U_e(t) + \left[B_e + \left(\frac{g_t - g_e}{w_3} + \frac{g_e}{w_4} \right) K_e \right] \dot{U}_e(t)$$

- Frequency response solutions 108 and 111:

$$F_e(\omega) = \left[(1 + i g_t) K_e + i\omega B_e \right] U_e(\omega)$$

New Force recovery for CELAS1, CELAS2, CELAS3 and CELAS4.

- Transient response solutions 109 and 112:

$$F_e(t) = K_e U_e(t) + \left(\frac{g_t - g_e}{w_3} + \frac{g_e}{w_4} \right) K_e \dot{U}_e(t)$$

- Frequency response solutions 108 and 111:

$$F_e(\omega) = (1 + ig_t)K_e U_e(\omega)$$

Element stress is computed by multiplying the stress coefficient S with the computed element force. The stress coefficient S can be optionally defined on the PELAS, PBUSH, and CELAS2 entries. Stress recovery is not supported on the CELAS4 element.

CBUSH Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example 1:

Noncoincident grid points.

CBUSH	39	6	1	100	75				

Example 2:

GB not specified.

CBUSH	39	6	1					0	

Example 3:

Coincident grid points.

CBUSH	39	6	1	100				6	

Example 4:

Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Fields:

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PBUSH entry. (Integer > 0; Default = EID)

Field	Contents
GA, GB	Grid point identification number of connection points. See Remark 6. (Integer GA > 0)
Xi	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to end A. See Remark 3 . (Integer > 0)
CID	Element coordinate system identification. A 0 means the basic coordinate system. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 1-1 and Remark 3 . (Integer ≥ 0 or blank)
S	Location of spring damper. See Figure 1-1. (0.0 ≤ Real ≤ 1.0; Default = 0.5)
OCID	Coordinate system identification of spring-damper offset. See Remark 9 . (Integer ≥ -1; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 1-1.)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0. See Figure 1-2 and Remark 9 .. (Real)

Remarks:

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

2. Figure 1-1 shows the bush element geometry.

3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. For cylindrical or spherical coordinate systems, if GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis.

4. For noncoincident grids, when GO or X1, X2, X3 is given and no CID is specified, the line AB is the element x-axis and the orientation vector $\vec{v}$ lies in the x-y plane (similar to the CBEAM element).

5. For noncoincident grids, if neither GO or X1, X2, X3 is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both are specified on the PBUSH entry (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.

6. If GA and GB are coincident, or if GB is blank, then CID must be specified. When GB is blank, a grounded spring and damper is created at GA.

7. Only CBUSH elements in the residual structure that do not attach to any omitted degree-of-freedom can reference a PID identifying both a PBUSH entry and a PBUSHT entry.

8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.

9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0, then S is ignored and S1, S2, S3 are used.

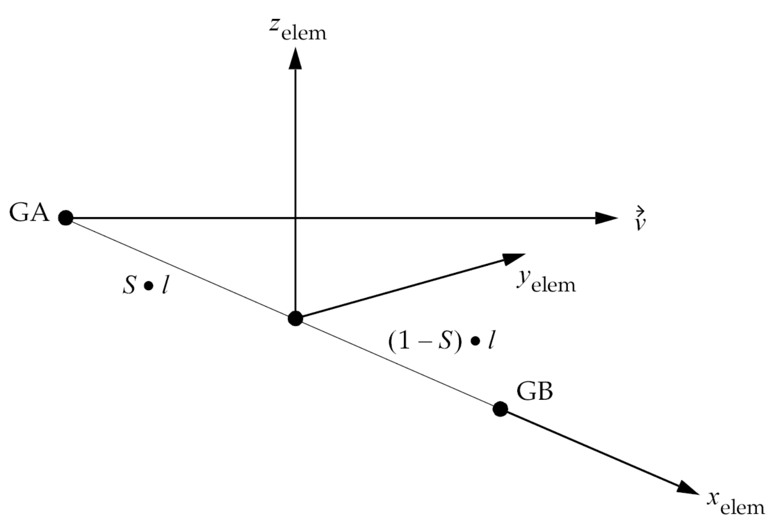

Figure 1-1. CBUSH Element

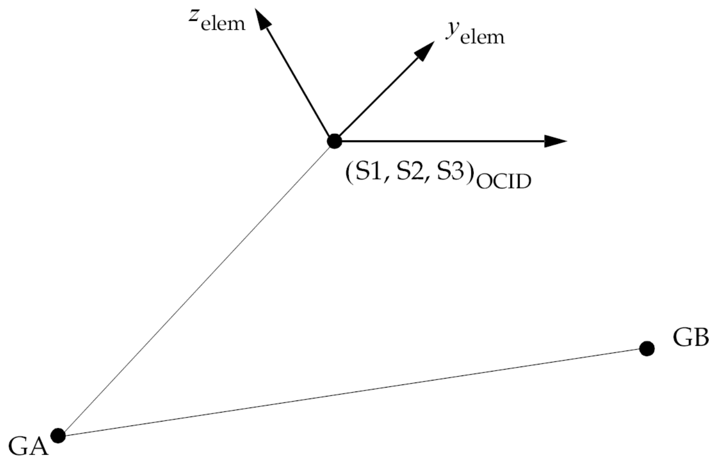

Figure 1-2. Definition of Offset S1, S2, S3

10. When CID ≥ 0, the element x-axis is set as in Remark 2 . This means that the element force is always computed as Ke. (UB – UA); if UA > UB, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation is tension and relative negative elongation is compression.

11. For this particular element, the effect of PARAM, G and the damping coefficient GE are included in the bushing stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters. For more details on element force calculation, see the NX Nastran Element Library.

Remarks related to SOL 601:

1. S, OCID, S1, S2 and S3 are not supported.

CELAS1 Scalar Spring Connection

Defines a scalar spring element.

Format:

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

Example:

CELAS1	2	6			8	1			

Fields:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.

2. In solutions 153 and 159, the software calculates heat conduction for CELAS1 elements as

   $q = K \times \Delta T$

   where K is the value entered on the PELAS entry. Both G1 and G2 must be defined, and C1 and C2 must be “1” when grid point ID’s are entered for G1 and G2.

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

4. The two connection points (G1, C1) and (G2, C2) must be distinct.

5. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.

7. If Gi refers to a grid point, then Ci refers to degree-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

8. The effect of PARAM,G and the damping coefficient GE (defined on the PELAS entry) are included in the spring stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters. For more details on element force and stress calculation, see the NX Nastran Element Library.

Remarks related to SOLs 601 and 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. In SOL 701, this spring element is not considered in the critical time step size calculation because it has no mass.

CELAS2 Scalar Spring Property and Connection

Defines a scalar spring element without reference to a property entry.

Format:

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

Example:

CELAS2	28	6.2+3	32		19	4			

Fields:

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. See Remarks 2 and 11. (Real)
G1, G2	Geometric grid point or scalar identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)
GE	Damping coefficient. See Remarks 9 and 10. (Real)
S	Stress coefficient. (Real)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.

2. In solutions 153 and 159, the software calculates heat conduction for CELAS2 elements as

   $q = K \times \Delta T$

   where K is the value entered on the CELAS2 entry. Both G1 and G2 must be defined, and C1 and C2 must be “1” when grid point ID’s are entered for G1 and G2.

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

4. The two connection points (G1, C1) and (G2, C2) must be distinct.

5. The element stress is computed by multiplying the stress coefficient S with the recovered element force.

6. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library.*

7. A scalar point specified on this entry need not be defined on an SPOINT entry.

8. If Gi refers to a grid point, then Ci refers to degree-of-freedom in the displacement coordinate system specified by CD on the GRID entry.

9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_o by 2.0.

10. The effect of PARAM, G and the damping coefficient GE are included in the spring stiffness for force and stress calculation in a frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters. For more details on element force calculation see the NX Nastran Element Library.

11. Rotational stiffness should be specified as moment per radian.

Remarks related to SOLs 601 and 701:

1. G1 and G2 must be grid points, i.e., they cannot be scalar points.

2. GE is ignored.

3. In SOL 701, this spring element is not considered in the critical time step size calculation because it has no mass.

CELAS3 Scalar Spring Connection to Scalar Points Only

Defines a scalar spring element that connects only to scalar points.

Format:

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

Example:

CELAS3	19	2	14	15					

Fields:

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.
4. For a discussion of the scalar elements, see "Overview of 0D (Scalar) Elements" in the *NX Nastran Element Library*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. In solutions 153 and 159, the software calculates heat conduction for CELAS3 elements as

   $q = K \times \Delta T$

   where K is the value entered on the PELAS entry. Both S1 and S2 must be defined.

7. The effect of PARAM,G and the damping coefficient GE (defined on the PELAS entry) are included in the spring stiffness for force and stress calculation in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See Parameters. For more details on element force calculation, see the NX Nastran Element Library.

CELAS4 Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

Format:

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

Example:

CELAS4	42	6.2-3	2						

Fields:

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring. See Remarks 8 and 9. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The structural damping coefficient GE is not available with CELAS4. The effect of PARAM,G is included in the spring stiffness for force calculations in frequency and transient analyses. The effects of PARAM,G can be excluded using PARAM,GDAMPF,0. If PARAM,W3 is not specified, PARAM,G is ignored in a transient analysis. See Parameters.
4. No stress coefficient is available with CELAS4.
5. Only one scalar spring element may be defined on a single entry.
6. For a discussion of the scalar elements, see “Overview of 0D (Scalar) Elements” in the *NX Nastran Element Library*.
7. A scalar point specified on this entry need not be defined on an SPOINT entry.

8. Rotational stiffness should be specified as moment per radian.

9. In solutions 153 and 159, the software calculates heat conduction for CELAS4 elements as

   $$q = K \times \Delta T$$

   where K is the value entered on the CELAS4 entry. Both S1 and S2 must be defined.

Chapter

2 Upward compatibility

Updated Modules

RSVECEL update

Converts RVEL bulk entries into residual vectors.

Updated Format:

```
RSVECEL    ECT,GEOM4,BGPDT,CSTM,EPT/
           RVELG/
           NOGSET/OPTION $
```

New Parameter:

OPTION   Input-Integer-Default=0. Residual vector request option.
=0 No RVEL/DAMP.
=1 RVEL only.
=2 DAMP only.
=3 RVEL and DAMP.

SDR2 updates

Creates tables based on output requests for forces of single-point and multipoint forces of constraint, applied loads, displacements, velocities, accelerations, element stresses, element strains, and element forces. These output tables are suitable for printing, plotting, and various other postprocessing.

Updated Format:

```
SDR2        CASECC,CSTM,MPT,DIT,EQEXIN,SILD,
            ETT,{OL or EDT},BGPDT,PG,QG,UG,EST,XYCDB,
            OINT,PELSET,VIEWTB,GPSNT,DEQATN,DEQIND,DITID,
            PCOMPT,GPKE,BOLTFOR,MDLIST,COMPEST,EPT/
            OPG1,OQG1,OUG1,OES1,OEF1,PUG,OGPKE1,OEFIIP,OEFIIS,OESRIP,OESRIS/
            APP/S,N,NOSORT2/NOCOMP/ACOUSTIC/METRIK/
            ISOFLG/GPF/ACOUT/PREFDB/TABS/
            SIGMA/ADPTINDX/ADPTEXIT/BSKIP/FREQW/
            BTBRS/LANGLE/OMID/SRCOMPS/APP1/GSPF $
```

New Parameters:

APP1 Input-character-default=' '. Analysis type. Required only if APP = 'MMREIG' and element force output is requested.

'FREQRESP'	Frequency response.
'TRANRESP'	Transient response.

GSPF Input-integer-default=1. PARAM,G inclusion in scalar element force calculation.

1	PARAM,G is included.
0	PARAM,G is not included.

Chapter

3 NX Nastran 8.5.1 Problem Report (PR) fixes

Problems identified in previous releases that have been fixed in NX Nastran 8.5.1 are listed below.

PR#	Problem Reported	Problem Description
1897304	V8.5	For SOL 601,106, stress results were incorrect on variable-node CTETRA elements.
1913903	V8.1	Performance issue with the external topology optimization interface has been corrected.
1914459	V8.5	Solid bolt preload was incorrect when using parabolic CTETRA elements to model the bolt.
1919125	V8.5	Solution failed with error “INSUFFICIENT MEMORY AVAILABLE FOR SUBROUTINE EXTERN”.
1922193	V8.5	Same as PR#1914459.
2203890	V7.1	Results were not stored when requesting strain in SORT2 format when using the element iterative solver.
2208380	V8.0	Performance issue when solving an extremely large model. The model, which included a glue definition, required extreme resources (over a terabyte of disk and 48GB of RAM) to complete.
2209531	V8.1	For SOL 601,106, performance issue when GPFORCE output is requested.
6728719	V8.1	Using Lagrange rigids and then restraining one of the grids on an RBE2 produced incorrect results.
6750101	V8.5	Fatal error occurred with specific model running SOL 103 with a bolt preload and contact in a statics subcase, then a STATSUB command in the modal subcase.
6750751	V8.1	Although the SEPDIS describer on the BCRESULTS case control is unsupported by SOL 601, if it is included in the input, no contact traction results were stored.

6770180	V8.1	CSET DOF were not correctly processed when doing a component mode synthesis (CMS).
6772145	V8.1	The structural damping values GE and G were applied inconsistently when computing force and stress on the CELASi and CBUSH elements. See Structural damping improvement.
6773201	V8.1	Same as PR#2209531.
6775064	V8.1	Punch file created from SOL 106 incorrectly included the text “NAN” (not a number) for a stress value.
6775070	V8.1	For SOL 106, modified 10 node CTETRA element to use 5 point integration by default. Also, investigate some convergence issues.
6775074	V8.1	HP-MPI was not compatible with a customer software product installed on their cluster. They needed an alternative MPI to run DMP.
6775442	V8.0	MATFT did not support a negative F_{12}. This restriction has been removed.
6775961	V8.1	SOL 111 can produce incorrect results if there are multiple subcases and the subcases include changes requiring a new eigensolution. Error messages have been added to trap changes that would require the new eigensolution between subcases.
6776193	V8.1	SOL 601, 106 terminated with an error when the 3D-iterative solver (SOLVER=2) was used with a CBUSH.
6776992	V7.1	An illegal DMIGSFIX=n was defined on EXTSEOUT, but the reported error message was not specific enough to trouble shoot the problem. The error is now specific.
6777527	V8.0	Fatal error occurred running a large model with many contact and bolt preload definitions.
6778230	V8.5	Error occurred with frequency dependent absorbers when multiple MAT10 cards were present.
6789017	V8.1	Incorrect SOL 200 sensitivities resulted when a MAT8 was used as a design variable along with connectivity related design variables.
6790645	V8.1	SOL 108, DMP failure when the number of frequencies was less than the number of processors defined with the “dmp” keyword.
6791034	V8.0	SOL 109 data recovery failed for an upstream superelement.
6794436	V8.1	Enhancement request to add RESVEC(DAMPLOD) along with changing the residual vector defaults to be ON. See Residual vectors.

6794440	V8.5	A very large model was run with the LP-64 executable and hung. The software began matrix factorization with BEND ordering but found it had insufficient memory. The BEND ordering has been enhanced to handle this class of model on LP-64.
6802683	V8.5	Fixed spurious warning generated from the FRLGEN module with AMLS.
6822104	V8.5	Although the SET selected on the ACMODL entry included CHEXA elements, the elements were ignored by the structural-acoustic coupling.
Other Fixes		
N/A	V8.5	Corrected EXTSEOUT processing in SOL 103 to simply output component modes rather than recalculate residual modes.
N/A	V8.5	Models containing SUPORTs and performing MBDEXPORT would experience a DMAP crash. This has been corrected.
N/A	V8.5	A multiply error for constraint force calculations using the absolute method of enforced motion has been corrected.
N/A	V8.5	Effective mass data blocks have been added to the OP2 for param,post,-1.
N/A	V8.5	Rotor dynamics models using the CBEAR bearing element gave wrong results for a synchronous solution. This has been corrected.
N/A	V8.5	A damping treatment error has been corrected, along with a few other minor DMAP issues.
N/A	V8.5	Some software functions have been renamed to prevent conflicts with a planned future MPI upgrade.
N/A	V8.5	Previously in an eigensolution, if the F1 field was defined on the EIGRL entry, but the largest mode was below F1, the Lanczos method correctly computed no modes, but incorrectly printed the fatal message “not all modes were found”. The software will no longer print this fatal message in this case.
N/A	V8.5	A SOL 103 DMP run which included structural-acoustic coupling failed with the message “cannot run DMP on a single processor”. This has been corrected.
N/A	V8.5	With structural-acoustic coupling, if the fluid element faces were highly warped, an incorrect material direction and hence face normal directions could be found. This has been corrected.

N/A	V8.5	With structural-acoustic coupling, free face candidates were computed with only the elements selected by FSET/SSET. As a result, it was possible that some faces found were not free faces. Now, all elements are considered when computing free face candidates.
N/A	V8.5	The use of system cells 413 - 416 has been prevented. They are used as internal tables within NX Nastran. The parameters OP2FMT, OP4FMT and INP4FMT should be used instead. The definition of these parameters in the QRG has been improved.
N/A	V8.5	Modified HP-MPI to allow dynamic memory allocation. This improves the performance of HP-MPI and may remove some conflicts with third party applications which expect HP-MPI to perform dynamic memory allocation.
N/A	V8.5	CWELD improvements: Improved error message for projection failures, allow coupling with projection off of face, and provide consistent behavior for different MSET options.
N/A	V8.5	For a bolt preload on solid elements, a problem where the incorrect bolt direction could be computed based on element connectivity has been corrected.
N/A	V8.5	Corrected the multiple subcase issue in the SDREE module for the external topology optimization interface.

Chapter

4 Changes to default settings

Changes to default settings

The following table summarizes changes to any default behavior or settings in NX Nastran 8.5.1.

Input Type	Default Changes
Keywords	None
Nastran Statements	None
File Management Statements	None
Executive Control Statements	None
Case Control Commands	To streamline the use of the RESVEC case control command, the default settings are changing. See Residual vectors for more information.
Parameters	
Bulk Entries	The force and stress recovery on CELASi and CBUSH elements now includes both PARAM,G and GE, and is supported in both frequency response and transient solutions. See Structural damping improvement for more information.

Chapter

5 System description summary

System description summary

The list of supported systems is included in the **README.txt** file located in the NX Nastran installation under the *nxn8p5* directory.