

NX Nastran 4.1

Release Guide

Proprietary & Restricted Rights Notice

© 2006 UGS Corp. All Rights Reserved. This software and related documentation are proprietary to UGS Corp. LIMITATIONS TO U.S. GOVERNMENT RIGHTS. UNPUBLISHED - RIGHTS RESERVED UNDER THE COPYRIGHT LAWS OF THE UNITED STATES. This computer software and related computer software documentation have been developed exclusively at private expense and are provided subject to the following rights: If this computer software and computer software documentation qualify as "commercial items" (as that term is defined in FAR 2.101), their use, duplication, or disclosure by the U.S. Government is subject to the protections and restrictions as set forth in the UGS Corp. commercial license for the software and/or documentation as prescribed in FAR 12.212 and FAR 27.405(b)(2)(i) (for civilian agencies) and in DFARS 227.7202-1(a) and DFARS 227.7202-3(a) (for the Department of Defense), or any successor or similar regulation, as applicable or as amended from time to time. If this computer software and computer documentation do not qualify as "commercial items," then they are "restricted computer software," and are provided with "restrictive rights," and their use, duplication or disclosure by the U.S. Government is subject to the protections and restrictions as set forth in FAR 27.404(b) and FAR 52-227-14 (for civilian agencies), and DFARS 227.7203-5(c) and DFARS 252.227-7014 (for the Department of Defense), or any successor or similar regulation as applicable or as amended from time to time. UGS Corp. , Suite 600 - 5800 Granite Parkway, Plano, Texas 75024.

NASTRAN is a registered trademark of the National Aeronautics and Space Administration. NX Nastran is an enhanced proprietary version developed and maintained by UGS Corp.

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.

C O N T E N T S

NX Nastran Release Guide

1

Introduction

- NX Nastran 4.1 Overview, 2

2

Dynamics

- SPC/SPCD Enforced Motion, 6
 - Introduction, 6
 - Residual Vector Update, 6
 - Mathematical Details for Relative Displacement Formulation, 8
 - Updated DISPLACEMENT Bulk Data Entry, 9
 - Updated VELOCITY Bulk Data Entry, 11
 - Updated ACCELERATION Bulk Data Entry, 13
- Random Response Analysis, 16
 - Introduction, 16
 - Theory for Cross-power Spectral Density and Cross-correlation Functions, 17
 - New RCROSS Case Control Card, 19
 - New RCROSS Bulk Data Entry, 20

3

Advanced Nonlinear

- Improvements to the Contact Algorithm, 24
 - Updated **NXSTRAT** Card, 25

4

Miscellaneous Enhancements

- Surface-to-Surface Gluing, 38
 - Introduction, 38
 - Defining Glue Regions (BSURF, BCPROP and BSURFS Bulk Entries), 38
 - Defining Glue Pairs (New BGSET Case Control and Bulk Entry), 39
 - New BGSET Case Control Card, 40
 - New BGSET Bulk Data Entry, 40
- Improvements to AUTOMPC, 42

A

Upward Compatibility

B

System Descriptions

- Updated Version of the **AUTOMPC** Parameter, 43
- Automatic Resubmitting of NX Nastran Jobs, 44
- Improvements to AUTOSPC, 45

- DMAP Module Additions/Changes from NX Nastran 4.0 to 4.1, 48
- Datablock Additions/Changes from NX Nastran 4.0 to 4.1, 55

- Supported System Configurations for NX Nastran 4.1, 60

CHAPTER

1

Introduction

- NX Nastran 4.1 Overview

1.1 NX Nastran 4.1 Overview

This section presents a short overview of the features introduced in NX Nastran 4.1.

Chapter 1, Introduction. Gives an overview of new features in NX Nastran 4.1.

Chapter 2, Dynamics. Features related to dynamic analysis have been implemented in this release. These features are summarized below:

- **SPC/SPCD Enforced Motion.** The current SPC/SPCD enforced motion capability has been enhanced to replace the current “absolute” displacement formulation with a “relative” displacement formulation. This enhancement will eliminate the need for automatic residual vector generation (which was done in the 4.0 release). In addition to the relative motion calculation, new output control options have been added to the DISPLACEMENT, VELOCITY, and ACCELERATION case control entries to designate whether you want absolute (default) or relative output.
- **Random Response Analysis.** The new RCROSS case control and bulk data commands are now available for requesting Cross-Power Spectral Density output.

Chapter 3, Advanced Nonlinear. This chapter describes new enhancements as related to solutions 601 and 701:

- **Improvements to the contact algorithm.** The contact algorithm used in SOL 601 and 701 has been updated with a new contact segment option. Specifically, the order of the segments created will match the order of the elements thus improving the output of contact results.

Chapter 4, Miscellaneous Enhancements. This chapter summarizes other useful enhancements that do not fit into the previously mentioned categories and covers the following topics:

- **Surface-to-Surface Gluing.** A new option to “glue” element faces together during a solution is available in this release of NX Nastran.

- **Improvements to AUTOMPC.** Performance improvements have been made to the AUTOMPC capability introduced in NX Nastran 3. The best performance improvements are seen with models containing large numbers of MPC equations. Also, a new option is available to force a degree-of-freedom (dof), which is included in an MPC equation, to be part of the m-set. You can now force a dof to be included, or not be included in the m-set.
- **New AUTOSPC method is available.** The new AUTOSPC method Singular Value Decomposition (SVD) is now available. The eigenvalue method is the default, but SVD can be selected by including the new system cell SVDSPC=1. The SVD method is numerically more reliable and should be turned on in cases when you find unexpected singularities remaining after the AUTOSPC operation.
- **Automatic Resubmitting of NX Nastran Jobs.** A new option is available to automatically resubmit jobs which fail to run because of a failed license request. The program will retry the job every minute up to the value of the new AUTHQUEUE keyword.

Appendix A, Upward Compatibility. This section contains the DMAP and Datablock changes from NX Nastran 4.0 to NX Nastran 4.1.

Appendix B, System Descriptions. This section contains a list of the system configurations which NX Nastran supports for this release.

CHAPTER

2

Dynamics

- SPC/SPCD Enforced Motion
- Random Response Analysis

2.1 SPC/SPCD Enforced Motion

Introduction

The SPC/SPCD method of enforced motion allows the direct specification of displacements, velocities, or accelerations via SPC/SPC1/SPCD data, thus eliminating the need to employ large mass or Lagrange Multipliers. Though it is a more straightforward method as compared to the large mass method of enforced motion, the current SPC/SPCD method using the absolute displacement formulation has shown to give poor results under certain circumstances, particularly when damping is included.

To correct these problems, a new “relative” displacement formulation has been introduced in this release to replace the current “absolute” displacement formulation. As the name implies, it is now based on a relative motion approach versus the absolute motion approach.

With the new relative approach, the issues associated with the SPC/SPCD method have been resolved, thus the SPC/SPCD method may now be considered as the enforced motion method of choice.

In addition to the formulation change, the new descriptors ABS/REL have been added to the DISPLACEMENT, VELOCITY, and ACCELERATION case control entries. Having these options on each entry allows you the flexibility to request, for example, relative displacement output with absolute acceleration output in the same run. They are also supported on a subcase level to allow for different output requests in different subcases. The updated DISPLACEMENT, VELOCITY, and ACCELERATION case control entries are included at the end of this section.

Residual Vector Update

In NX Nastran version 4.0, there was a change to make residual vector generation happen automatically when there was enforced motion present. This was done to improve enforced motion results.

With the new relative displacement formulation in release 4.1, the enforced motion results are of good quality without the addition of residual vector generation. Residual vector generation is no longer automatic in this release, and you are required to use PARAM,RESVEC,YES and PARAM,RESVINNER,YES if you want residual vector creation to occur. Below are the updated parameters showing their new defaults:

RESVEC Default=NO (Default updated to NO for SOL 111 & 112 when enforced motion is present.)

By default, residual vectors are not computed. If you want residual vectors due to:

- applied loads (LOAD or LOADSET Case Control commands) then specify PARAM,RESVEC,YES.
- inertial loads (unit acceleration of mass) then specify PARAM,RESVINER,YES.
- unit loads, then specify USETi,U6 entries at the desired dofs and PARAM,RESVEC,YES.

If the model is unrestrained, you will need to supply SUPORTi entries.

Related parameters are:

PARAM name	Default Value	Description
RESVEC	NO	Augment static shapes due to applied loads.
RESVINER	NO	Augment static shapes due to inertial loads (unit acceleration of mass).
RESVSO	YES	Reorthogonalize static shapes with mode shapes. If the stiffness is well-conditioned, this is not required.
RESVSE	NO	Print strain energy of the static shapes.
RESVSLI	YES	Remove linearly dependent shapes.

RESVINER Default=NO (Default = NO for SOL 111 & 112 when enforced motion is present.)

See related parameter RESVEC.

Mathematical Details for Relative Displacement Formulation

The SPC/SPCD enforced motion formulation uses an absolute displacement formulation per the equation:

$$M_{ff}\ddot{u}_f + C_{ff}\dot{u}_f + K_{ff}u_f = P_f - (M_{fs}\ddot{u}_s + C_{fs}\dot{u}_s + K_{fs}u_s)$$

where:

M, C, K = mass, damping, and stiffness matrices

u_f = absolute displacements of the f-set dof

P_f = loads applied to the f-set dof

u_s = applied enforced motion

The damping loading term $C_{fs}\dot{u}_s$ is not included.

The absolute displacement formulation equation will be replaced by the more accurate enforced motion equations:

$$M_{ff}\ddot{y}_f + C_{ff}\dot{y}_f + K_{ff}y_f = P_f - (-M_{ff}K_{ff}^{-1}K_{fs} + M_{fs})\ddot{u}_s - (-C_{ff}K_{ff}^{-1}K_{fs} + C_{fs})\dot{u}_s$$

$$u_f = y_f - K_{ff}^{-1}K_{fs}u_s$$

where y_f = relative displacement of the f-set dof.

Using the two equations above requires that the residual vector be computed from the term:

$$-M_{ff}K_{ff}^{-1}K_{fs} + M_{fs}$$

If there is discrete damping present, an additional residual vector will be computed from the term:

$$-C_{ff}K_{ff}^{-1}K_{fs} + C_{fs}$$

The damping loading term $(-C_{ff}K_{ff}^{-1}K_{fs} + C_{fs})\dot{u}_s$ will also be included, although this term will not have a large impact since the damping is usually specified in terms of modal diagonal damping.

Updated DISPLACEMENT Bulk Data Entry

DISPLACEMENT Displacement Output Request

Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.

Format:

$$\text{DISPLACEMENT} \left[\left(\left[\begin{array}{c} \text{[SORT1]} \\ \text{[SORT2]} \end{array} \right], \left[\begin{array}{c} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{array} \right], \left[\begin{array}{c} \text{[REAL or IMAG]} \\ \text{PHASE} \end{array} \right], \left[\begin{array}{c} \text{[ABS]} \\ \text{[REL]} \end{array} \right] \right) \left[\begin{array}{c} \text{[PSDF]} \\ \text{[ATOC]} \\ \text{[RALL]} \end{array} \right] \right]$$

$$= \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Examples:

DISPLACEMENT=5

DISPLACEMENT(REAL)=ALL

DISPLACEMENT(SORT2, PUNCH, REAL)=ALL

Describers	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of load, frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, displacement data.
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.
ABS	For enforced motion dynamic analysis, displacement results will be output as absolute displacement.

Describers	Meaning
REL	For enforced motion dynamic analysis, displacement results will be output relative to the enforced motion input.
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.
RALL	Requests that power spectral density function and 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.
ALL	Displacements for all points will be output.
NONE	Displacement for no points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

- Both PRINT and PUNCH may be requested.
- The type of analysis determines which "SORT" is the default:
 - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis.
 - 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, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO, then the remaining commands will also be output in SORT1 format. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.
 - XY plot requests will force SORT2 format overriding SORT1 format requests.

3. VECTOR and PRESSURE are alternate forms and are entirely equivalent to DISPLACEMENT.
4. DISPLACEMENT=NONE overrides an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer output request is present for magnitude/phase representation.
6. The units of translation are the same as the units of length of the model. Rotations are in units of radians.
7. Displacement results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
8. The PSDF, ATOC, and RALL options are for future support of random analysis post-processing. They provide no output to the .f06.

Remarks related to SOLs 601 and 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC and RALL are ignored.
2. Displacements, velocities and accelerations must be output for the same set of grid points if requested. Output requested for set n in this command will be combined with the sets requested in the VELOCITY and ACCELERATION commands, and displacements will be output at the grid points of the combined set.

Updated VELOCITY Bulk Data Entry

VELOCITY Velocity Output Request

Requests the form and type of velocity vector output.

Format:

$$\text{VELOCITY} \left[\left(\left[\begin{array}{l} \text{[SORT1]} \\ \text{[SORT2]} \end{array} \right], \left[\begin{array}{l} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{array} \right], \left[\begin{array}{l} \text{[REAL or IMAG]} \\ \text{PHASE} \end{array} \right], \left[\begin{array}{l} \text{[ABS]} \\ \text{[REL]} \end{array} \right], \left[\begin{array}{l} \text{[PSDF]} \\ \text{[ATOC]} \\ \text{[RALL]} \end{array} \right] \right) \right]$$

$$= \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Examples:

VELOCITY=5
 VELOCITY(SORT2,PHASE,PUNCH)=ALL

Describers	Meaning
SORT1	Output will be presented as a tabular listing of grid points 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 grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, velocities.
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.
ABS	For enforced motion dynamic analysis, velocity results will be output as absolute velocity.
REL	For enforced motion dynamic analysis, velocity results will be output relative to the enforced motion input.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests that both the power spectral density function and the autocorrelation function be calculated and stored in the database for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the Case Control.
ALL	Velocity for all solution points will be output.
NONE	Velocity for no solution points will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. Velocity output is only available for transient and frequency response problems.
3. See Remark 2 under “**DISPLACEMENT**” on page 9 for a discussion of SORT1 and SORT2.
4. VELOCITY=NONE overrides an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer request is present for the magnitude/phase representation.
6. Velocity results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. The PSDF, ATOC, and RALL options are for future support of random analysis post-processing. They provide no output to the .f06.

Remarks related to SOLs 601 and 701:

1. Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC and RALL are ignored.
2. Displacements, velocities and accelerations must be output for the same set of grid points if requested. Output requested for set n in this command will be combined with the sets requested in the DISPLACEMENT and ACCELERATION commands, and velocities will be output at the grid points of the combined set.

Updated ACCELERATION Bulk Data Entry

ACCELERATION

Acceleration Output Request

Requests form and type of acceleration vector output.

Format:

$$\text{ACCELERATION} \left(\begin{array}{c} \left[\text{SORT1} \right] \\ \left[\text{SORT2} \right] \end{array} , \begin{array}{c} \left[\text{PRINT, PUNCH} \right] \\ \left[\text{PLOT} \right] \end{array} , \begin{array}{c} \left[\text{REAL or IMAG} \right] \\ \left[\text{PHASE} \right] \end{array} , \begin{array}{c} \left[\text{ABS} \right] \\ \left[\text{REL} \right] \end{array} , \begin{array}{c} \left[\text{PSDF} \right] \\ \left[\text{ATOC} \right] \\ \left[\text{RALL} \right] \end{array} \right) \\
 = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

ACCELERATION=5
 ACCELERATION(SORT2, PHASE)=ALL
 ACCELERATION(SORT1, PRINT, PUNCH, PHASE)=17

Describers	Meaning
SORT1	Output will be presented as a tabular listing of grid points 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 grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Computes, but does not print or punch, acceleration output.
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.
ABS	For enforced motion dynamic analysis, acceleration results will be output as absolute acceleration.
REL	For enforced motion dynamic analysis, acceleration results will be output relative to the enforced motion input.
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.
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.
RALL	Request both the power spectral density function and 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.
ALL	Accelerations at all points will be output.
NONE	Accelerations at no points will be output.
n	Set identification of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. Acceleration output is only available for transient and frequency response problems.
3. See Remark 2 under “**DISPLACEMENT**” on page 9 for a discussion of SORT1 and SORT2.
4. ACCELERATION=NONE allows overriding an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer request is present for magnitude/phase representation.
6. Acceleration results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. The PSDF, ATOC, and RALL options are for future support of random analysis post-processing. They provide no output to the .f06.

Remarks related to SOLs 601 and 701:

Output is restricted to REAL format. IMAG, PHASE, PSDF, ATOC and RALL are ignored.

2.2 Random Response Analysis

Introduction

In many applications, the loading on a structure is not known in a definite (or deterministic) sense, and instead is quantified in a statistical sense with properties such as the mean load and standard deviation of load. Such loadings are known as random or stochastic loads. Examples of random loads include: rough road surface loads on automobiles, ocean wave loads on offshore platforms, wind loads on buildings, vibration loads from rocket engines, and earthquake ground motions.

It should be expected that the response of the structure due to random loads will also be random in nature and analysis methods that determine the statistical properties of response are appropriate. The most commonly used approach for random analysis uses Power Spectral Density (PSD) functions to define loading and the calculated response. A PSD function describes how the variance of a time series of the product of two variables is distributed with frequency. If the two variables are the same, the PSD function is known as an auto spectrum, if the variables are different, the PSD function is called a cross spectrum.

NX Nastran has the capability for performing random analysis. The procedure is a two-step process in which first a frequency response analysis is performed to calculate transfer functions which are the ratio of output to a unit input. In the second step, the PSD loading function is used to scale the transfer functions to compute the random PSD response. The input PSD functions represent the statistics of the applied loads. If there are several applied loads and they are uncorrelated from each other, only the auto spectra of the loads need be applied. If on the other hand the loads are not independent of each other, the cross spectrum of the loads need also be input to define the correlation of the loadings. Note that although the process is described as two steps, it can be performed with NX Nastran as only one solution submittal.

Random response output consists of the response PSD, autocorrelation functions, number of zero crossings with positive slope per unit time, and RMS (root-mean-square) values of response. Up until the release of NX Nastran version 4.0, only auto spectrum PSD functions of response could be output for random results. Users were required to use DMAP methods if they were interested in obtaining cross spectrum results. Now with NX Nastran 4.1, it is possible to request cross spectrum output using standard PSD solutions. A new Case Control command, **RCROSS** and Bulk Data entry, **RCROSS** have been created for this purpose. Formats of these entries are included at the end of this section.

The following examples have been provided in the install location `NX_Nastran_Install_Path/nxn4/nast/tpl` to demonstrate capabilities and to show the correct syntax for using the new RCROSS commands: `rcross01.dat`, `rcross02.dat`, `rcross03.dat`, `rcross04.dat`.

Theory for Cross-power Spectral Density and Cross-correlation Functions

Assuming that a random process is ergodic, the cross correlation function between a pair of time records $u_a(t)$ and $u_b(t)$ is defined as

$$R_{ab}(\tau) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_0^T u_a(t) u_b(t - \tau) dt \quad (1)$$

where $u_a(t)$ and $u_b(t)$ can be displacement, velocity, acceleration or single-point constraint force responses at the same or different grid points; or stress, strain, and force components in the same or different elements. The assumption of ergodicity means that as the length of the record is increased the average approaches the corresponding ensemble averages. The operation of integration over a time record of length T and dividing that quantity by T represents an averaging process over time.

If we denote the Fourier transform of a time history as

$$U_a(\omega, T) = \int_0^T u_a(t) e^{-i\omega t} dt \quad (2)$$

we can define the cross-power spectral density (cross-PSD) as

$$S_{ab}(\omega) = \lim_{T \rightarrow \infty} \frac{2}{T} U_a(\omega, T) U_b^*(\omega, T) \quad (3)$$

where ω is angular frequency in rad/sec and the symbol (*) denotes complex conjugate. Simply stated, the cross-PSD is the product of the Fourier transform of two records.

It can be shown using equations (1) and (2) that the cross correlation function and cross spectrum are related as Fourier transform pairs given as

$$S_{ab}(\omega) = 2 \int_{-\infty}^{\infty} R_{ab}(\tau) e^{-i\omega \tau} d\tau \quad (4)$$

$$R_{ab}(\tau) = \frac{1}{4\pi} \int_{-\infty}^{\infty} S_{ab}(\omega) e^{i\omega \tau} d\omega \quad (5)$$

It is noted that the cross correlation function is always real valued and the cross spectrum function is in general complex valued. In addition, it can be shown to possess the following properties:

$$R_{ab}(-\tau) = R_{ba}(\tau) \quad (6)$$

$$S_{ab}(-\omega) = S_{ba}(\omega) = S_{ab}^*(\omega) \quad (7)$$

It should also be noted that we have been referring to the general case where $u_a(t)$ and $u_b(t)$ are different responses. In many situations, only a single response, rather than the joint response is of interest. In that case, auto-correlation functions, $R_{aa}(\tau)$, and auto-spectrum functions, $S_{aa}(\omega)$, are computed. The auto-spectrum by definition is completely real.

Returning to the more general case, we can rewrite $S_{ab}(\omega)$ in terms of its real and imaginary parts as

$$S_{ab}(\omega) = S_{ab}^r(\omega) + iS_{ab}^i(\omega) \quad (8)$$

From the properties in equation (5) it can be determined that $S_{ab}^r(\omega)$ is an even function of ω and $S_{ab}^i(\omega)$ is an odd function of ω .

Substituting equation (8) into the equation (7) and recalling that $e^{i\omega t} = \cos(\omega t) + i \sin(\omega t)$, it can be shown that

$$R_{ab}(\tau) = \frac{1}{2\pi} \int_0^\infty [S_{ab}^r(\omega) \cos(\omega\tau) - S_{ab}^i(\omega) \sin(\omega\tau)] d\omega \quad (9)$$

A particular case of interest is the value of the cross-correlation function for $\tau = 0$. As evidenced by equation (1) it represents the mean cross product. Using equation (9) it can be computed as

$$R_{ab}(0) = \frac{1}{2\pi} \int_0^\infty S_{ab}^r(\omega) d\omega = \overline{u_a(t)u_b(t)} \quad (10)$$

where $(\overline{\quad})$ denotes mean average.

The mean cross product is an important statistical quantity. For cross-correlation functions, it is known as the covariance. For auto-correlation functions the mean cross-product is known as the variance and its square root is the standard deviation. By using the standard assumption that loadings and response are Gaussian (normally distributed), knowing the covariance allows you to define the probability distribution and thus determine likelihood of occurrence of response levels. For example, if the standard deviation of response $u_a(t)$ is determined to be σ_a , the likelihood that at any time the response is less than $3\sigma_a$ is 99.9%.

If a system is subject to multiple sources of excitation, the cross-PSD of responses $u_a(t)$ and $u_b(t)$ is related to the cross-PSDs of the excitation sources and the frequency response functions by the equation

$$S_{ab}(\omega) = \sum_j \sum_k H_{aj}(\omega) H_{bk}^*(\omega) S_{jk}(\omega) \quad (11)$$

where $S_{jk}(\omega)$ is the cross-PSD of loadings $P_j(t)$ and $P_k(t)$. $H_{aj}(\omega)$ is the frequency response function of $u_a(t)$ corresponding to the excitation source $P_j(t)$ and similarly $H_{bk}(\omega)$ is the complex conjugate of the frequency response function of $u_b(t)$ corresponding to the excitation source $P_k(t)$. If the two excitation sources are not correlated, we have $S_{jk}(\omega) = 0$.

New RCROSS Case Control Card

RCROSS

Cross-Power Spectral Density Functions Output Request

Requests computation and output of cross-power spectral density functions in random analysis.

Format:

$$\text{RCROSS} \left[\left(\left[\frac{\text{REAL or IMAG}}{\text{PHASE}} \right], \left[\frac{\text{PRINT}}{\text{NOPRINT}} \right], [\text{PUNCH}], [\text{PSDF}] \right) \right] = n$$

Examples:

RCROSS(PHASE) = 10

RCROSS(PSDF,NOPRINT,PUNCH) = 20

RCROSS = 30

Describers	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output (for cross-power spectral density function). Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output (for cross-power spectral density function). Phase output is in degrees.
PRINT	Writes output to print file. (Default)

Describers	Meaning
NOPRINT	Does not write output to the print file.
PUNCH	Writes output to punch file.
PSDF	Requests the cross-power spectral density function be calculated and output for random analysis post-processing. (Default)
n	Identification number of RCROSS bulk data entry to be used in random analysis. (Integer >0)

Remarks:

1. The case control RCROSS entry must be used in conjunction with the case control RANDOM entry. See remarks under the RANDOM case control entry.
2. Response quantities, such as DISPLACEMENT, STRESS, and FORCE, must be requested by corresponding case control entries in order to compute cross-power spectral density between the two response quantities specified by the RCROSS bulk data entry. It is recommended that the DISPLACEMENT, STRESS, and FORCE requests be put above the subcase level to ensure that these response quantities exist when the random analysis post-processing occurs.
3. The response quantities must belong to the same superelement. The cross-power spectral density functions between two responses that belong to different superelements are not supported.

New RCROSS Bulk Data Entry

RCROSS Cross-Power Spectral Density Functions Output

Defines a pair of response quantities for computing the cross-power spectral density functions in random analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RCROSS	SID	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID	

Example:

RCROSS	20	DISP	50	2	STRESS	150	8	4	
--------	----	------	----	---	--------	-----	---	---	--

Field	Contents
SID	Case control RCROSS identification number for cross-power spectral density function. (Integer>0)
RTYPEi	Type of response quantity. At least one field must be selected. See remark 2. (Character or blank)
IDI	Element, grid, or scalar point identification number. (Integer > 0)
COMPi	Component code (item) identification number. See remark 3. (Integer > 0)
CURID	Curve identification number. See remark 4. (Integer ≥ 0 or blank)

Remarks:

1. This entry is required for computing the cross-power spectral density function. SID must be selected with the case control command (RCROSS=SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity; fields RTYPE2, ID2, and COMP2 represent the second response quantity.
2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

If either RTYPE1 or RTYPE2 is blank, then the blank field takes the default from the defined field.

3. For elements, the item code $COMP_i$ represents a component of the element stress, strain or force and is described in Tables “Element Stress-Strain Item Codes” and “Element Force Item Codes”. For an item having both a real and imaginary part, the code of the real part must be selected.

For grid points, the item code is one of 1,2,3,4,5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar points, always use 1.

4. Field CURID is optional. It is for your convenience to identify the output by using a single index.

CHAPTER

3

Advanced Nonlinear

- Improvements to the Contact Algorithm

3.1 Improvements to the Contact Algorithm

When contact definitions exist in an Advanced Nonlinear Solution (SOLs 601 and 701), contact segments are created during the solution in the regions where contact has been defined. These contact segments are created on the shell and solid element faces.

A new contact segment option is available for SOL 601 in NX Nastran 4.1 that gives improved contact results. Specifically, the order of the segments created (linear or quadratic) will match the order of the elements. When quadratic contact segments are created on quadratic elements, the resulting contact tractions are significantly better.

The new parameter CSTYPE has been created on the bulk data entry NXSTRAT to support the new contact segment creation.

CSTYPE Selects the type of contact segment to use. (Integer; Default = 0)
 0 - Use the old type of contact segment.
 1 - Use the new type of contact segment.

CSTYPE=1 is more accurate and will give better contact traction results, especially when used with quadratic elements (for example, 10-grid tetrahedral elements). However, certain restrictions apply in NX Nastran 4.1 when CSTYPE=1 is used. These restrictions, listed below, will be eliminated in future releases. Note that the default settings comply with the restrictions.

- Only constraint function algorithm (default) may be used, i.e., TYPE=0 in BCTPARA
- Contact surface offset must be zero (default), i.e., OFFSET=0.0 in BCTPARA
- Only single-sided contact (default) is allowed, i.e., NSIDE=1 in BCTPARA
- Only frictionless contact (default) or regular Coulomb friction is allowed, i.e., FRICMOD=0 or 1 in BCTPARA
- No tied contact (default), i.e., TIED=0 in BCTPARA
- Consistent contact stiffness cannot be used, i.e., CSTIFF=0 in BCTPARA
- Continuous normal must be used (default), i.e., SEGNORM=0 or 1 in BCTPARA

- Initial penetration cannot be ignored (default), i.e., INIPENE=0 or 1 in BCTPARA

Updated NXSTRAT Card

(This includes a change to the param LSEARCH which is not specifically related to the contact output improvements.)

NXSTRAT Strategy Parameters for SOLs 601 and 701

Defines parameters for solution control and strategy in advanced nonlinear structural analysis.

Format:

	1	2	3	4	5	6	7	8	9	10
NXSTRAT	ID	Param1	Value1	Param2	Value2	Param3	Value3			
	Param4	Value4	Param5	Value5	-etc-					

Example:

NXSTRAT	1	AUTO	1	MAXITE	30	RTOL	0.005			
	ATSNEXT	3								

Field	Contents
ID	Identification number. Currently not used. (Integer ≥ 0)
PARAMi	Name of the NXSTRAT parameter. Allowable names are given in Table 8-1. See remark 1 for parameters applicable to SOL 701. (Character)
VALUEi	Value of the parameter. See Table 8-1. (Real or integer)

Table 8-1 NXSTRAT Parameters

Name	Description
Analysis Control Parameters	
SOLVER	Selects the solver to use. (Integer; Default = 0) 0 – Direct sparse solver 1 – Multigrid solver
AUTO	Indicates whether automatic incrementation scheme is enabled. (Integer; Default = 0) 0 – No automatic incrementation scheme is used 1 – Automatic time stepping (ATS) scheme is enabled 2 – Automatic load-displacement control (LDC) scheme is enabled
NPOSIT	Indicates whether analysis continues when the system matrix is not positive definite. (Integer; Default = 0) 0 – Analysis may stop 1 – Analysis continues Notes: If NPOSIT=0, analysis stops unless the ATS or LDC scheme is enabled; i.e. AUTO=1 or AUTO=2 contact analysis is being performed It is not recommended to set NPOSIT=1 for a linear analysis.
MASSTYP	Selects the type of mass matrix to be used in dynamic analysis. (Integer; Default = 1) 0 – Lumped mass is used 1 – Consistent mass is used
Analysis Options	
TINT*	Integration order for the local t-direction (through thickness) of shell elements with elasto-plastic materials. By default, 5-point Newton-Cotes is used for single-layered shell and 3-point Newton-Cotes is used for multi-layered shell. Note that 2-point Gauss integration is always used for all shell elements with elastic materials. (Integer; Default = 0). $1 \leq \text{TINT} \leq 6$ – Gauss integration method with integration order TINT -3, -5, -7 – Newton-Cotes integration with order -TINT
ICMODE*	Indicates whether incompatible modes are used for 4-node shell elements. (Integer; Default =1 for SOL 601 and 0 for SOL 701) 0 - Incompatible modes are not used 1 - Incompatible modes are used

Name	Description
MSTAB	<p>Indicates whether the stiffness matrix stabilization feature is used. (Integer; Default = 0)</p> <p>0 – Matrix stabilization is not used</p> <p>1 – Matrix stabilization is used</p>
MSFAC	<p>Matrix stabilization factor. (Real; Default = 1.0E-12)</p>
DTDELAY*	<p>Element death time delay. (Real; Default = 0.0)</p> <p>When an element is too deformed and becomes “dead”, its contribution to the overall stiffness of the structure is removed. By specifying DTDELAY > 0.0, the contribution from the element stiffness is gradually reduced to zero over time DTDELAY instead of being suddenly removed. This may help in the convergence of the solution.</p>
SDOFANG*	<p>Angle used to determine whether a shell mid-surface node is assigned 5 or 6 degrees of freedom. (Real; Default = 5.0)</p>
UPFORM*	<p>Indicates whether u/p formulation is used for elements. Note that u/p formulation is always used for hyperelastic elements and always not used for hyperfoam elements and elastic elements with Poisson’s ratio less than 0.48. It is also not used for gasket elements (Integer; Default = 0)</p> <p>0 – u/p formulation is not used</p> <p>1 – u/p formulation is used instead of displacement-based formulation</p>
ULFORM*	<p>Indicates which large strain formulation is used for 4-node shell elements. (Integer; Default = 0)</p> <p>0 – Updated Lagrangian-Jaumann (ULJ) formulation is used if rigid-target algorithm contact is used or SOL 701 is run. Otherwise, Updated Lagrangian-Hencky (ULH) formulation is used.</p> <p>1 – Use ULH formulation</p> <p>2 – Use ULJ formulation</p> <p>Note: For shell elements which are not 4-noded, the ULJ formulation is always used for large strains. For 3-D solid, plane strain and axisymmetric elements, ULH formulation is always used for large strains. In restarting from SOL 601 to 701 or vice versa, ULFORM needs to be specified such that both analyses use the same formulation.</p>

Name	Description
DISPOPT	Indicates whether prescribed displacements are applied to the original configuration or the deformed configuration. This option is only applicable for a restart analysis or when a delay (or arrival) time is specified for the prescribed displacement. (Integer; Default = 0) 0 – Applied to original configuration 1 – Applied to deformed configuration
LOADOPT*	Indicates whether prescribed loads (pressure and centrifugal) are deformation-dependent, i.e. the direction and magnitude of the load may change due to large deformation of the structure. This option is only applicable for large displacement analysis, i.e. PARAM,LGDISP,1 (Integer; Default = 1) 0 – Load is independent of structural deformation 1 – Load is affected by structural deformation

Time Integration

ALPHA	Alpha coefficient for the Newmark time integration method. (Real; Default = 0.25)
DELTA	Delta coefficient for the Newmark method. (Real; Default = 0.5)

SOL 701 Time Stepping

XSTEP ^o	Selects time step method used in an explicit time integration analysis. (Integer; Default = 0) 0 - Time step size is calculated by the program based on the critical time step size. The data in the selected TSTEP bulk data entry is used to calculate the total solution time for the analysis. 1 - The number of time steps and the time step size as specified in the selected TSTEP bulk data entry is used.
XDTCAL ^o	Calculation of the critical time step size may be computationally expensive. This parameter specifies that the critical time step size be re-calculated every XDTCAL time steps. (Integer > 0, Default = 1)
XDTFAC ^o	The critical time step size is calculated based on certain assumptions. It is often necessary, especially for nonlinear analysis, to use a time step size smaller than the calculated critical time step size. The factor multiplied by the calculated critical time step size gives the time step size used in the analysis. (4.0 > Real > 0.0, Default = 0.9)

Name	Description
XMSCALE ^o	Specifies the factor to scale the mass (densities) of the entire model (at the beginning of the analysis) to increase the critical time step size required for stability when the explicit time integration scheme is used. See warning in Remark 2. (Real = 1.0, Default = 1.0)
XDTMIN1 ^o	The minimum time step size used to determine if mass scaling will be applied to elements (at the beginning of the analysis) whose critical time step size is smaller than DTMIN1. The amount of mass scaling is calculated for each element so that the critical time step size is equal to DTMIN1. See Remark 2 and warning in Remark 3. (Real = 0.0, Default = 0.0)
XDTMIN2 ^o	The minimum time step size used to determine whether an element will be removed in an explicit time integration analysis. In explicit time integration, the smaller an element size is, the smaller will the critical time step size be. If the critical time step size for an element is smaller than XDTMIN2, the element will be removed in the analysis. See Remark 2 and warning in Remark 3. (Real > 0.0, Default = 0.0)

Multigrid Solver

ITEMAX	Maximum number of iterations allowed for the multigrid solver to converge. (Integer > 0; Default = 1000)
EPSIA	Convergence tolerance EPSIA. (Real; Default = 1.0E-6)
EPSIB	Convergence tolerance EPSIB. (Real; Default = 1.0E-4)
EPSII	Convergence tolerance EPSII. (Real; Default = 1.0E-8)

Equilibrium Iteration and Convergence

LSEARCH	Flag to indicate the use of line searches within the iteration scheme. (Integer; Default = 0) 0 – Line search is not used 1 – Line search is used
LSLOWER	Lower bound for line search. ($0.0 \leq \text{Real} < 1.0$; Default = 0.001)
LSUPPER	Upper bound for line search. ($1.0 \leq \text{Real}$; Default = 2.0)

Name	Description
MAXITE	Maximum number of iterations within a time step. If the maximum number of iterations is reached without achieving convergence (see CONVCR1 parameter), the program will stop unless the automatic time stepping (ATS) or load displacement control scheme is selected (see parameter AUTO). ($1 \leq \text{Integer} \leq 999$; Default = 15)
CONVCRI	Convergence Criteria. (Integer; Default = 0) 0 – Convergence based on energy 1 – Convergence based on energy and force 2 – Convergence based on energy and displacement 3 – Convergence based on force 4 – Convergence based on displacement
ETOL	Relative energy tolerance. (Real; Default = 0.001)
RTOL	Relative force (and moment) tolerance (Real; Default = 0.01)
RNORM	Reference force. (Real)
RMNORM	Reference moment. (Real)
RCTOL	Relative contact force tolerance. (Real; Default = 0.05)
DTOL	Relative displacement (translation and rotation) tolerance. (Real; Default = 0.01)
DNORM	Reference translation. (Real)
DMNORM	Reference rotation. (Real)
STOL	Line search convergence tolerance. (Real; Default = 0.5)
RCONSM	Reference contact force. (Real; Default = 0.01)
ENLSTH	Line search energy threshold. (Real; Default = 0.0)

Automatic Time Stepping (ATS) Scheme

ATSSUBD	Number that limits the smallest time step size when the automatic time stepping (ATS) scheme is used. For a time step size of DT, the program will stop if convergence is not achieved and the next subdivided time step size is less than $DT / \text{ATSSUBD}$. (Integer ≥ 1 ; Default = 10)
ATSMXDT	Factor that limits the maximum time step size when the automatic time stepping (ATS) scheme is used. The ATS scheme may increase the time step size after convergence is achieved. However, for a time step size of DT, the program will not use a time step size greater than $\text{ATSMXDT} * DT$. (Real; Default = 3.0)

Name	Description
ATSNEXT	<p>Flag controls what time step size to use once convergence is reached after an ATS subdivision. (Integer; Default = 0)</p> <p>0 – Automatically set by program. For contact analysis, ATSNEXT = 2, otherwise ATSNEXT = 1.</p> <p>1 – Use the time step size that gave convergence, i.e., the reduced time step that led to convergence is used again.</p> <p>2 – Return to the original time step size, i.e., the original time step size before any subdivision took place is used.</p> <p>3 – Use a time step size such that the solution time matches the original solution time specified by the user.</p>
ATSDFAC	<p>Division factor used calculate the sub-increment time step size. If current time step size is DT and convergence is not achieved, the next time step size will be DT/ATSDFAC. (Real > 1.0; Default = 2.0)</p>
ATSLOWS	<p>Flag whether a low-speed dynamics analysis is performed instead of a static analysis. (Integer; Default = 0)</p> <p>0 – Low-speed dynamics option is not activated</p> <p>1 – Low-speed dynamics is performed</p>
ATSDAMP	<p>Damping factor used in low-speed dynamics analysis. (Real \geq 0.0; Default = 1.0e-4)</p>

Load Displacement Control (LDC) Scheme

LDCGRID	<p>Grid point id at which a displacement is prescribed for the first solution step. (Integer > 0)</p>
LDCDOF	<p>Degree of freedom for prescribed displacement at grid point LDCGRID. ($1 \leq$ Integer \leq 6)</p> <p>1 – X translation</p> <p>2 – Y translation</p> <p>3 – Z translation</p> <p>4 – X rotation</p> <p>5 – Y rotation</p> <p>6 – Z rotation</p>
LDCDISP	<p>Prescribed displacement at grid point LDCGRID for the first solution step. (Real)</p>
LDCIMAX	<p>Displacement convergence factor used to limit the maximum incremental displacement during a solution step. (Real; Default = 3.0)</p>

Name	Description
LDCDMAX	Maximum (absolute magnitude) displacement (for the degree of freedom specified by LDCDOF) at the grid point LDCGRID allowed during the analysis. When the displacement reaches or exceeds LDCDMAX, the program will stop the analysis. See Section 6.2.4 in Advanced Nonlinear Theory and Modeling Guide for other criteria that determines when an LDC solution will stop. (Real)
LDCCONT	Flag whether the solution is terminated when the first critical point on the equilibrium path is reached. (Integer; Default = 0) 0 – Solution stops 1 – Solution continues
LDCSUBD	Maximum number of arc length subdivisions allowed. (Integer \geq 1; Default = 10)

Contact Control

IMPACT	Impact control scheme (Integer; Default = 0) 0 – No special treatment is applied for impact problems 1 – Post impact adjustment of velocities and accelerations is applied 2 – Modified parameters are used in Newmark time integration scheme
NSUPP	Number of iterations for pairing contactor node to target segment. If NSUPP > 0, during the first NSUPP iterations, the pairing target segment is recorded for each contactor node. From iteration NSUPP+1, if a target segment in the recorded list is repeated, it is “frozen” to be the pairing target segment for the remaining equilibrium iterations in that time step. Specifying NSUPP > 0 may help in the convergence for certain problems. (0 \leq Integer \leq 99; Default = 0)
RTSUBD	Selects the subdivision scheme used in the implicit rigid-target contact algorithm when the tensile contact force is too large. (Integer; Default = 0) 0 – Subdivision is based on the magnitude of the tensile contact force, i.e., the larger the magnitude, the smaller will be the subdivided time step size. 1 – Subdivision is based on the global automatic time stepping (ATS) subdivision settings.
CSTYPE	Selects the type of contact segment to use. (Integer; Default = 0) 0 - Use the old type of contact segment. 1 - Use the new type of contact segment.

Name Description

Restart Options

MODEX*	<p>Indicates the mode of execution. (Integer; Default = 0) 0 – Normal analysis run, i.e. not a restart analysis 1 – Restart analysis</p> <p>The restart (.res) file from a previous run must exist to do a restart analysis. The filename and location of the restart file is determined by the “dbs” keyword. By default, dbs points to the current working directory with the prefix of the current job name.</p>
TSTART*	<p>Solution starting time. If MODEX=1, TSTART must equal a solution time in which data was saved in a previous run. If TSTART = 0.0, the last time step in the restart file is used. (Real, Default = 0.0)</p>
IRINT*	<p>Frequency of saving the analysis results in the restart file. (Integer; Default = 0) 0 – IRINT is set to 1 when implicit time integration is used and set to the number of steps in the first time step block when explicit time integration is used. > 0 – Restart file is overwritten every IRINT time steps < 0 – Restart file is appended every IRINT time steps</p>

Other Parameters

NSUBGRP*	<p>Number of sub-groups to divide large number of elements with same property ID into. Normally, elements with same type and property ID are placed into a group. If a group contains more than 1000 elements and NSUBGRP > 1, the elements are placed into NSUBGRP sub-groups for more efficient processing. (Integer > 0; Default = 1)</p>
----------	--

Translation Options

Name	Description
ELCV	<p>Convert 8-node to 9-node quadrilateral (plane strain, axisymmetric and shell) elements and 20-node to 27-node brick elements. Note that this also converts 6-node to 7-node triangular (plane strain and axisymmetric) elements and 10-node to 11-node tetrahedral elements. (Integer; Default = 0)</p> <p>0 – No conversion of elements</p> <p>1 – Convert elements as described above; nodal coincidence is not checked against existing nodes and new generated nodes are always created.</p> <p>2 – Convert elements as described above; nodal coincidence is checked against existing nodes and a new node will not be created at a location if a node already exist at that location.</p>
EQRBAR	<p>Indicates how RBAR elements are handled. (Integer; Default = 0)</p> <p>0 – RBAR is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis.</p> <p>1 – RBAR is simulated using rigid option (i.e. simulated by rigid link or constraint equations as determined by program)</p> <p>2 – RBAR is simulated using flexible option (i.e. simulated by spring or beam elements as determined by program)</p> <p>3 – RBAR is simulated by spring elements</p> <p>See Section 2.7 of Advanced Nonlinear Theory and Modeling Guide for details on how RBAR elements are handled.</p>
EQRBE2	<p>Indicates how RBE2 elements are handled. (Integer; Default = 0)</p> <p>0 – RBE2 is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis.</p> <p>1 – RBE2 is simulated using rigid option (i.e. simulated by rigid links or constraint equations as determined by program)</p> <p>2 – RBE2 is simulated using flexible option (i.e. simulated by spring or beam elements as determined by program)</p> <p>3 – RBE2 is simulated by spring elements</p> <p>See Section 2.7 of Advanced Nonlinear Theory and Modeling Guide for details on how RBE2 elements are handled.</p>
SPRINGK	<p>Stiffness of spring elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0)</p> <p>If SPRINGK = 0.0, program automatically sets SPRINGK according to the following calculations.</p> <p>$SPRINGK = EMAX * LMODEL$</p> <p>where EMAX = maximum Young's Modulus of materials in the model and LMODEL = largest dimension of the model. If no material is specified in the model, EMAX is set to 1.0E12.</p>

Name	Description
BEAME	<p>Young's Modulus of material assigned to beam elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0)</p> <p>If BEAME = 0.0, BEAME is set to EMAX * 100.0 where EMAX = maximum Young's Modulus of materials in the model. If no material is specified in the model, EMAX is set to 1.0E12.</p>
BEAMA	<p>Circular cross section area of beam elements that simulate RBAR or RBE2 elements. (Real, Default = 0.0)</p> <p>If BEAMA = 0.0, program automatically sets BEAMA according to the following calculation: $BEAMA = (LMODEL * .01)^2$ where LMODEL = largest dimension of the model</p>
RBLCRIT	<p>Critical length for determining how RBAR and RBE2 elements are simulated when the rigid or flexible option is used to simulate RBAR (see EQRBAR) and RBE2 (see EQRBE2). (Real, Default = 0.0)</p> <p>If RBLCRIT = 0.0, then</p> <ul style="list-style-type: none"> • if EQRBAR (or EQRBE2) = 1, RBLCRIT = LMODEL * 1.0E-6 • if EQRBAR (or EQRBE2) = 2, RBLCRIT = LMODEL * 1.0E-3

Remarks:

1. Parameters applicable to SOL 701 are:
 - XSTEP, XDTCAL, XDTFAC, XMSCALE, XDTMIN1 and XDTMIN2 are only used for SOL 701. These parameters are indicated in the table with a superscript ^{'0'}.
 - TINT, ICMODE, DTDELAY, SDOFANG, UPFORM, LOADOPT, MODEX, TSTART, IRINT and NSUBGRP parameters can be used for SOL 701. These parameters are indicated in the table with a superscript ^{'*}.
2. XMSCALE, XDTMIN1 and XDTMIN2 may be used together. XDTMIN1 and XDTMIN2 are applied after XMSCALE is applied. If XDTMIN1 and XDTMIN2 are both used, XDTMIN1 should be greater than XDTMIN2. If XDTMIN2 = XDTMIN1 is specified, XDTMIN1 will be ignored.
3. WARNING: Specifying XMSCALE > 1.0, XDTMIN1 > 0.0 or XDTMIN2 > 0.0 may change the model significantly. Hence, extra care should be exercised in examining the results when any of these parameters are used.

CHAPTER

4

Miscellaneous Enhancements

- **Surface-to-Surface Gluing**
- **Improvements to AUTOMPC**
- **Automatic Resubmitting of NX Nastran Jobs**
- **Improvements to AUTOSPC**

4.1 Surface-to-Surface Gluing

Introduction

A new option to “glue” element faces together during a solution is available in this release of NX Nastran. Glue definitions can be used in all solution sequences except for SOL 601 and 701. The glue option creates stiff springs to connect the pre-defined surfaces and prevents relative motion in all directions. For discussion purposes, the springs created by the solver from glue definitions will be referred to as “glue elements” in the documentation.

Pre-defined regions of element free faces are used to detect where the glue elements are created. From each element free face, a normal is projected, then the solver checks to see if any of these normals intersect with other element free faces. A glue element is created during the solution if:

- NX Nastran finds an intersection between element faces,
- and the distance between the two faces is equal to or less than the defined separation distance.

The following should be noted:

- Superelements can be used with surface-to-surface gluing as long as the superelements are defined with the SESET bulk entry.
- In a linear statics solution (SOL 101), surface gluing and surface contact can be defined for the same subcase.
- A glue set can only be defined in the global or first subcase and it will be used in all subcases. The exception to this is when linear contact is defined in SOL 101. For this special case, glue sets can be defined in any subcase. This limitation will be removed in NXN5.

Defining Glue Regions (BSURF, BCPROP and BSURFS Bulk Entries)

You must first define the regions where you would like the solver to create glue elements. A region is a collection of element free faces in a section of the model where you expect gluing (or contact) to occur. These regions can be created using shell elements (BSURF and BCPROP) and using solid element free faces (BSURFS). The BSURF entry is defined by its own unique ID and is a list or range of shell element IDs to include in the region. The BCPROP entry is defined by its own unique ID and is a list of shell element property IDs. Shell elements which use any of the property IDs listed in the BCPROP entry will be included in the

region. The BSURFS entry is defined by its own unique ID and is a list of solid element IDs each followed by 3 grid points defining which face of the 3D element to include in the glue region. An element should not appear more than once in the regions that are part of a glue/contact pair. The BSURF, BCPROP and BSURFS entries described are used to define glue and/or contact regions.

When a source region is made from shell elements (BSURF or BCPROP), the SURF field on the BCRPARA bulk entry may be needed to alter the search direction. The top side of a shell element is determined by the order and direction of grid points on the element's definition and the right-hand rule. (See chapter 15 of the NX Nastran User's Guide for information on using consistent normals) When shell elements are used as the source in a contact pair, by default, the solve will use the top side to project the element normal while searching for the target elements. In the case where the target is on the bottom side, you can set the SURF option to "BOT", and the normal will be projected from the bottom side. Understanding shell element normals, and making sure they are consistent in the source regions is very important to ensuring that glue elements will be created as expected. On the BCRPARA bulk entry, CRID and SURF are the only fields used by glue regions.

Defining Glue Pairs (New BGSET Case Control and Bulk Entry)

Two new commands have been created in this release to define glue conditions for the solution, the new BGSET case control command, and the new BGSET bulk entry, both of which are included below.

A glue pair is a way to combine two regions, source and target, in which gluing will be analyzed during the solution. The new BGSET bulk entry is used to define each glue pair. The GSID field will need to match the value of 'n' on the new BGSET case control entry for the solution to recognize this glue definition. The SIDi and TIDi fields refer to regions created by the BSURF, BCPROP and BSURFS entries, and are used to define source and target regions respectively for a pair. As many pairs as desired can be included on a single BGSET entry.

The search distance field (SDIST) defines the distance in which the solver can initially determine if the distance between element faces in a particular pair are within the threshold for creating glue elements. The default value of SDIST of 100 is large enough to handle most geometry situations, but can be adjusted as needed. This value is used once, at the beginning of the solution, to determine where glue elements need to be initially created. Recall that NX Nastran projects normals from element faces and then checks to see if any of these normals intersect with another element free face. If the

projected normal intersects an element face, and the distance between the two element faces is within the range defined on the SDIST fields, a glue element is created.

The remaining field on the BGSET bulk entry is the penalty (PEN) factor, which is used to adjust the stiffness of the glue condition. The default penalty factor is sufficient for most cases, but can be adjusted if separation of the surfaces is observed.

Examples have been provided in the install location `NX_Nastran_Install_Path/nxn4/nast/tpl` to demonstrate capabilities and to show the correct syntax for surface-to-surface gluing. These files will all start with "glue" following with a solution number. For example, one of the files is `glue103.dat`.

New BGSET Case Control Card

BGSET Glue Contact Set Selection

Selects the glue contact set.

Format:

BGSET=n

Examples:

BGSET=5

Describers	Meaning
n	Glue contact set identification of a BGSET Bulk Data entry. (Integer>0)

New BGSET Bulk Data Entry

BGSET 3D Glue Contact Set Definition

Defines glued contact pairs of a 3D set.

Format:

1	2	3	4	5	6	7	8	9	10
BGSET	GSID	SID1	TID1	SDIST1	PEN				
		SID2	TID2	SDIST2					
		-etc-							

Example:

BGSET	4	1	2						
		4	3						

Field	Contents
GSID	Glue set identification number. (Integer>0)
SIDi	Source region identification number of glue pair i. (Integer > 0)
TIDi	Target region identification number for glue pair i. (Integer > 0)
SDISTI	Search distance for glue surfaces (Real);(Default= 100)
PEN	Penalty value used in calculations. The value defined on the first line will be used for all pairs. (Real);(Default = 1.0E5)

Remarks:

1. The default search distance will glue all overlapping sections of the source and target regions. This value can be used in special cases to control what subregions are to be glued.
2. The default penalty factor will be sufficient for most cases. Increase it if separation of the surfaces is observed. If defined too large, numerical problems may occur.

4.2 Improvements to AUTOMPC

In version 3 of NX Nastran, the Parameter AUTOMPC was introduced as an automatic way of processing the dependent and independent degree-of-freedom for rigid elements, constraint elements and MPC equations. Before AUTOMPC was created, you were required to select independent and dependent degree-of-freedom when creating rigid elements, constraint elements and MPC equations. This was a tedious process for models with large numbers of constraints.

In version 4.1, the following improvements have been made to AUTOMPC:

- Models with large numbers of MPC equations, particularly with large numbers of RBE3 elements containing large numbers of grids, will see performance improvements.
- When the AUTOMPC was originally created in version 3, an option was created to specify that a dof (a dof which is included in an MPC equation) **must not** be made part of the m-set. This is done by including a list of degree-of-freedom on USET/USET1 bulk data cards and making them part of the U5 set. Now in version 4.1, an option to do the opposite has been created. Including dof as part of the U4 set specifies they **must** be made part of the m-set.

An updated version of the AUTOMPC parameter is included below.

Updated Version of the AUTOMPC Parameter

AUTOMPC Default=NO

AUTOMPC=YES specifies that the software automatically selects the m-set dofs rather than use the m-set values as specified on MPC or RIGID element cards (RBE*, RBAR, RROD, etc.) in the bulk data definition except as noted below.

This option relieves the need to carefully define rigid elements (or MPCs) so as to not have a conflict of the m-sets between elements. In addition, any redundant constraints will be eliminated.

This option is not available in some circumstances and will be automatically set to NO for:

- 1) A p-element analysis with local coordinate systems or RSSCON elements.
- 2) A design optimization solution (SOL=200) with DVGRID data.

The AUTOMPC option is recommended for use in models with RSSCON elements connected to CPENTA elements.

If an RBE3 element contains UM information on the m-set data, those dofs will be used in the m-set. SPOINTS in MPC equations will always be made part of the m-set.

When using this option, it is possible to specify that selected dof **must not** be made part of the m-set. This is accomplished by defining these dofs on USET/USET1 bulk data cards making them part of the U5 set. If specified, the UM information on RBE3 elements will be ignored.

An option also exists to specify that selected dof **must** be made part of the m-set. Again, using USET/USET1 bulk data cards, but making them part of the U4 set.

4.3 Automatic Resubmitting of NX Nastran Jobs

When an NX Nastran job fails because of a failed license request, an option to have the job retried automatically is now available. NX Nastran will now retry a failed license request job every minute up to the value of the AUTHQUEUE keyword. The default is 20 minutes.

4.4 Improvements to AUTOSPC

The Singular Value Decomposition (SVD) method is now available to AUTOSPC unrestrained conditions in your model. The eigenvalue method is still used by default, but SVD can be selected using the new system cell `SVDSPC=1`. The SVD method is numerically more reliable and should be turned on in cases when you find unexpected singularities remaining after the AUTOSPC operation.

System Cell Number	System Cell Name	Function and Reference
419	SVDSPC	Determines which AUTOSPC method is used. 0: Use the eigenvalue method (Default) 1: Use the Singular Value Decomposition (SVD) method

APPENDIX

A

Upward Compatibility

- DMAP Module Additions/Changes from NX Nastran 4.0 to 4.1
- Datablock Additions/Changes from NX Nastran 4.0 to 4.1

A.1 DMAP Module Additions/Changes from NX Nastran 4.0 to 4.1

The following DMAP modules are additions or have changed in this release:

- **GP4**
- **OFF**

GP4 Generates the degree-of-freedom set table

Generates the degree-of-freedom set table based on single point constraints, multipoint constraints, rigid elements, and set membership assignment Bulk Data entries (e.g., ASET). Also generates the enforced displacement matrix, the multipoint constraint equation matrix, and the enforced motion partitioning vector.

Format:

```
GP4          CASECC , GEOM4 , EQEXIN , SIL , GPDT , BGPDT , CSTM ,
            MEDGE , MFACE , MBODY , GEOM2 , GDNTAB /
            RMG , YSO , USET0 , PARTV /
            LUSET / S , N , NOMSET / S , N , MPCF2 / S , N , NOSSET / S , N , NOOSET /
            S , N , NORSET / S , N , NSKIP / S , N , REPEAT / S , N , NOSET / S , N , NOL /
            S , N , NOA / SEID / ALTSHAPE / SEBULK / DMAPNO / AUTOMPC $
```

Input Data Blocks:

CASECC	Table of Case Control command images.
GEOM4	Table of Bulk Data entry images related to constraints, degree-of-freedom membership and rigid element connectivity.
EQEXIN	Equivalence table between external and internal grid/scalar identification numbers.
SIL	Scalar index list.
GPDT	Grid point definition table.
BGPDT	Basic grid point definition table.
CSTM	Table of coordinate system transformation matrices.
MEDGE	Edge table for p-element analysis.
MFACE	Face table for p-element analysis.
MBODY	Body table for p-element analysis.
GEOM2	Table of Bulk Data entry images related to element connectivity and scalar points.
GDNTAB	Table of grid points generated for p-element analysis.

Output Data Blocks:

RMG	Multipoint constraint equation matrix.
YS0	Matrix of enforced displacements.
USET0	Degree-of-freedom set membership table for g-set.
PARTV	Partitioning vector of enforced motion dof.

Parameters:

LUSET	Input-integer-default=0. The number of degrees-of-freedom in the g-set.
NOMSET	Output-integer-no default. Number of degrees-of-freedom in the m-set or multipoint constraint and rigid element flag. Set to -1 if there are none.
MPCF2	Output-integer-no default. Multipoint constraint set identification number change flag. Set to 1 if the current subcase contains a different multipoint constraint set from the previous subcase. Set to -1 otherwise or if there are no multipoint constraints in the current subcase.
NOSSET	Output-integer-no default. Number of degrees-of-freedom in the s-set, or single point constraint flag. Set to -1 if there are none.
NOOSET	Output-integer-no default. Number of degrees-of-freedom in the o-set or omitted degree-of-freedom flag. Set to -1 if there are none.
NORSET	Output-integer-no default. Number of degrees-of-freedom in the r-set, or supported degree-of-freedom flag. Set to -1 if there are none.
NSKIP	Input/output-integer-no default. The record number in CASECC corresponding to the first subcase of the current boundary condition.
REPEAT	Output-integer-no default. Last boundary condition flag. Set to -1 at the last boundary condition; +1 otherwise.
NOSET	Output-integer-no default. Constraint, omit, and support set flag. Set to -1 if NOMSET=-1, NOSSET=-1, NOOSET=-1, NORSET=-1 and no degrees-of-freedom defined in the a-set (e.g., ASETi, QSETi Bulk Data entries); +1 otherwise.
NOL	Output-integer-default=1. Dependent set flag. Set to -1 if all degrees-of-freedom belong to m-set, s-set, o-set, and/or r-set; otherwise, the degrees-of-freedom in the l-set.
NOA	Output-integer-default=1. Constraint and omit set flag. Set to -1 if NOMSET=-1, NOSSET=-1, and NOOSET=-1; otherwise the number of degrees-of-freedom in the a-set.
SEID	Input-integer-default=0. Superelement identification number.

- ALTSHAPE Input-integer-default=0. Specifies set of displacement functions in p-element analysis. ALTSHAPE=0 selects the MacNeal set and 1 selects the Full Product Space set.
- SEBULK Input-logical-default=FALSE. Partitioned superelement presence flag. Set to TRUE if partitioned superelements are present or BEGIN SUPER is specified for the first BEGIN BULK Case Control command.
- DMAPNO Input-integer-default=0. DMAP solution number.
- AUTOMPC Input-character-default=NO. Auto MPC processing flag.

Remarks:

1. YS and PARTV will be purged if SPCD or SPC Bulk Data entries do not specify nonzero values for displacement, velocity, or acceleration.
2. GEOM4 may be purged.
3. CSTM may be purged if no coordinate systems are used.

OFF Output file processor

To output (print or punch) data blocks prepared by other modules in user-oriented, self-explanatory formats.

Format:

OFF OFF1 , OFF2 , OFF3 , OFF4 , OFF5 , OFF6 ,
 CSTM , BGPDTVU , ERROR1 , DEQATN , DEQIND , DIT //
 S , N , CARDNO / ODCODE / PVALID / DFLAG / VFLAG / AFLAG \$

Input Data Blocks:

OFFi Output table suitable for processing by the OFF module. See Remark 2.
 CSTM Table of coordinate system transformation matrices.
 EHT Element hierarchical table for p-element analysis.
 BGPDTVU Basic grid point definition table for a superelement and related to
 geometry with view-grids added.
 ERROR1 Error-estimate table updated for current superelement or adaptivity
 loop.
 DEQATN Table of DEQATN Bulk Data entry images.
 DEQIND Index table to DEQATN data block.
 DIT Table of TABLEDi Bulk Data entry images.

Output Data Blocks:

None.

Parameters:

CARDNO Input/output-integer-default=0. Punch file line counter. CARDNO is
 incremented by one for each line written to the punch file and is also
 written into columns 73-80 of each line.
 ODCODE Input-integer-default=-1. Output device code override. See Remark 4.
 ODCODE overrides the code stored in the DBi's according to the
 following table:

ODCODE	Output Directed to:
1	Print
2	Plot
3	Print and Plot

4	Punch
5	Print and Punch
6	Plot and Punch
7	Print, Plot, and Punch

PVALID Input-integer-default=0. P-element adaptivity loop identification number.

DFLAG Input-integer-default=0. Displacement output flag.
 = 0 output displacement "as is"
 = 1 output displacement as "relative motion"
 = 2 do not output displacement

VFLAG Input-integer-default=0. Velocity output flag.
 = 0 output velocity "as is"
 = 1 output velocity as "relative motion"
 = 2 do not output velocity

AFLAG Input-integer-default=0. Acceleration output flag.
 = 0 output acceleration "as is"
 = 1 output acceleration as "relative motion"
 = 2 do not output acceleration

Remarks:

1. Any or all data blocks may be purged.
2. DMAP modules READ (LAMA, OEIGS, LAMX, CLAMX), CEAD (CLAMA and OCEIGS), and LAMX (LAMB) are matrix operation modules that prepare OFP formatted data blocks. Modules SDR2, SDR3, VDR, VDRE, ADR, CURV, DDRMM, DRMH3, ELFDR, GPFDR, GPWG, LAMX, MDATA, SDRCOMP, SDRX, and SDRHT also prepare OFP formatted data blocks.
3. Parameter ODCODE is not honored by data blocks LAMA, OEIGS, LAMX, CLAMA, and OGPWG, which are created by READ, CEAD, LAMX, and GPWG.
4. CSTM, EHT, BGPDTVU, and ERROR1 are required if p-elements are specified and only for data recovery; i.e., displacement, stress, strain, etc.

5. CSTM, DEQATN, DEQIND, and DIT are required if the CORD3G Bulk Data entry is present and only for element data recovery; i.e., displacement, stress, strain, etc.

Example:

Print the OUG1 table from the SDR2 module:

```
OFP OUG1/ $
```

A.2 Datablock Additions/Changes from NX Nastran 4.0 to 4.1

The following Datablocks are additions or have changed in this release:

- **OCPSDF**

OCPSDF Table of cross-power spectral density functions.

Table of cross-power spectral density functions.

Record 0 - HEADER

Word	Name	Type	Description
1	NAME(2)	CHAR4	Block Name

Record 1 - IDENT

Word	Name	Type	Description
1	ACODE(C)	I	Device code + 10*Approach code
2	TCODE(C)	I	Table code; always 4100
3	UNDEF	none	
4	RANDID	I	RANDOM set identification number
5	CCODE	I	Device code +10*function number
6	RCROID	I	RCROSS identification number
7	UNDEF(2)	none	
9	OCODE	I	Output code (2=real, 3=complex)
10	NUMWDE(C)	I	Length of entries in RECORD=DATA (always 3)
11	RTYPE1	CHAR4	Type of first response quantity
12	ID1	I	Element, grid or scalar point ID number
13	COMP1	I	Component code (item) ID number
14	RTYPE2	CHAR4	Type of second response quantity
15	ID2	I	Element, grid, or scalar point ID number
16	COMP2	I	Component code (item) ID number
17	CURID	I	Curve ID number
18	UNDEF(33)	none	
51	TITLE(32)	CHAR4	Title

Word	Name	Type	Description
83	SUBTITL(32)	CHAR4	Subtitle
115	LABEL(32)	CHAR4	Label

Record 2 - DATA

Word	Name	Type	Description
1	FREQ	RS	Frequency (Hz)
2	RPSDF	RS	Real part of cross-power spectral density function value
3	IPSDF	RS	Imaginary part of cross-power spectral density function value

Record 3 - TRAILER

Word	Name	Type	Description
1	WORD1	I	Number of records
2	WORD2	I	Number of data values in record 2.
3	UNDEF(5)	none	

Notes:

- Records repeat for each subcase having any output requests. Words 1-3 in record 2 repeat for each frequency. Records 1 and 2 repeat for each function generated.
- Device code:
 - 1 = print
 - 4 = punch
 - 5 = print and punch
- Approach code:
 - 1 = statics
 - 2 = reigen
 - 3 = ds0
 - 4 = ds1
 - 5 = freq

6 = bkl0

7 = bkl1

8 = ceigen

9 = pla

4. RTYPEi:

DISP = displacement

VELO = velocity

ACCE = acceleration

OLOA = applied loads

SPCF = single-point constraint force

MPCF = multi-point constraint force

STRE = stress

STRA = strain

FORC = force

APPENDIX

B

System Descriptions

- Supported System Configurations for NX Nastran 4.1

B.1 Supported System Configurations for NX Nastran 4.1

System Description – HP9000 – HP-UX

Item	Description
Supported Model(s)	PA-RISC
Configurations for Installed Timing Constants	250, 710, 712, 715, 720, 730, 735, 778, 800, 819, 889, 2200, 2600, 2733, 3700, 4000, 4900, 6750
Build Operating System	PA-RISC: HP-UX B.11.00
Other Supported Operating Systems	HP-UX B.11.11
Word Length	32 bits
Build Type	LP-64, LP-64 DMP

System Description – Intel – Linux

Item	Description
Supported Model(s)	Intel and Intel-compatible
Configurations for Installed Timing Constants	P4 2.8Ghz
Build Operating System	Red Hat 7.3
Other Supported Operating Systems	Suse 9.2, Red Hat EL 3.0, Red Hat EL 4.0
Word Length	32 bits
Build Type	ILP-32, ILP-32 DMP

System Description – Intel – Windows (32-bit)

Item	Description
Supported Model(s)	Intel and Intel-compatible
Configurations for Installed Timing Constants	Pentium II 400 MHz, P4 1.5GHz, Pentium Pro.
Build Operating System	Windows 2000, SP3
Other Supported Operating Systems	WXP SP1, WXP SP2, WXP-64 (on EM64T/Opteron), Win 2003 Server (on EM64T/Opteron)
Word Length	32 bits
Build Type	ILP-32

System Description – IBM RS/6000 – AIX (64 bit)

Item	Description
Supported Model(s)	Power, Power2, Power3, Power4
Installed Timing Constants	303,320H, 370, 375, 390, 397, 530, 530h, 550, 560, 570, 580, 590, 591, 950, 980, 990, 4316
Build Operating System	AIX 5.1
Other Supported Operating Systems	AIX 5.2, AIX 5.3
Word Length	LP-64: 32 bits; ILP-64: 64 bits
Build Type	LP-64, LP-64 DMP, ILP-64, ILP-64 DMP

System Description – SGI R8K, R10K, R12K – IRIX64

Item	Description
Supported Model(s)	R8K, R10K, R12K, R16K
Configurations for Installed Timing Constants	IP7, IP19, IP20, IP21, IP22, IP27, IP28, IP30, IP35, 240, 510
Build Operating System	IRIX 6.5.7

System Description – SGI R8K, R10K, R12K – IRIX64

Item	Description
Other Supported Operating Systems	IRIX 6.5.21, IRIX 6.5.24m, IRIX 6.5.27m
Word Length	32 bits
Build Type	LP-64, LP-64 DMP

System Description – Sun SPARC – Solaris

Item	Description
Supported Model(s)	UltraSPARC
Installed Timing Constants	UltraSPARC (75 & 95)
Build Operating System	UltraSPARC:Solaris 8
Other Supported Operating Systems	Solaris 9, Solaris 10
Word Length	32 bits
Build Type	LP-64

System Description – X86_64 Linux (AMD Opteron/EM64T)

Item	Description
Supported Model(s)	X86-64 Linux
Installed Timing Constants	8664
Build Operating System	SuSE 9.0
Other Supported Operating Systems	Suse 9.1, Suse 9.3, Suse SLES9, Red Hat EL 3.0, Red Hat EL 4.0
Word Length	LP-64: 32 bits; ILP-64: 64 bits
Build Type	LP-64, LP-64 DMP, ILP-64, ILP-64 DMP

System Description – Intel Itanium HP-UX

Item	Description
Supported Model(s)	Intel-Itanium-HP-UX
Installed Timing Constants	4900, 5300, 5400
Build Operating System	HP-UX B.11.22
Other Supported Operating Systems	HP-UX B.11.23
Word Length	LP-64: 32 bits; ILP-64: 64 bits
Build Type	LP-64, LP-64 DMP, ILP-64, ILP-64 DMP

System Description – Intel Itanium Linux

Item	Description
Supported Model(s)	Itanium II
Installed Timing Constants	IA-64 800 Mhz & 733Mhz
Build Operating System	Red Hat AW2.1
Other Supported Operating Systems	Red Hat EL3.0, Red Hat EL4.0
Word Length	LP-64: 32 bits; ILP-64: 64 bits
Build Type	LP-64, LP-64 DMP, ILP-64, ILP-64 DMP

System Description – SGI Altix

Item	Description
Supported Model(s)	SGI-ALTIX
Installed Timing Constants	6402
Build Operating System	SGI ProPack 3 sp3
Other Supported Operating Systems	SGI ProPack 4

System Description – SGI Altix

Item	Description
Word Length	LP-64: 32 bits; ILP-64: 64 bits
Build Type	LP-64, LP-64 DMP, ILP-64, ILP-64 DMP

System Description – Intel (EM64T/Opteron) – Windows (64 bit)

Item	Description
Supported Model(s)	Intel and Intel-compatible
Configurations for Installed Timing Constants	Pentium II 400 MHz, 733 MHz Itanium 32 Bit, P4 1.5GHz, Pentium Pro.
Build Operating System	Windows 2003 Server
Other Supported Operating Systems	WXP-64
Word Length	32 bits
Build Type	LP-64