MSC Nastran 2007 r1

Release Guide

Corporate
MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707 USA
Telephone: (800) 345-2078
Fax: (714) 784-4056

Europe
MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Asia Pacific
MSC.Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjyku-Ku
Tokyo 160-0023, JAPAN
Telephone: (03)-6911-1200
Fax: (03)-6911-1201

Worldwide Web
www.mscsoftware.com

Disclaimer

NA:2007R1:Z:Z:Z:DC-REL

Contents

MSC Nastran 2007 r1 Release Guide

Preface

- List of Books
- Technical Support
- Internet Resources

List of Books

Below is a list of some of the MD Nastran and MSC Nastran documents. You may order any of these documents from the MSC.Software BooksMart site at http://store.mscsoftware.com/.

Installation and Release Guides

- Installation and Operations Guide
- Release Guide

Reference Books

- Quick Reference Guide
- DMAP Programmer's Guide
- Reference Manual

User's Guides

- Getting Started
- Linear Static Analysis
- Basic Dynamic Analysis
- Advanced Dynamic Analysis
- Design Sensitivity and Optimization
- Thermal Analysis
- Numerical Methods
- Aeroelastic Analysis
- Superelement
- User Modifiable
- Toolkit
- Implicit Nonlinear (SOL 600)
- Explicit Nonlinear (SOL 700)
- MD User's Guide

Technical Support

For help with installing or using an MSC.Software product, contact your local technical support services. Our technical support provides the following services:

- Resolution of installation problems
- Advice on specific analysis capabilities
- Advice on modeling techniques
- Resolution of specific analysis problems (e.g., fatal messages)
- Verification of code error.

If you have concerns about an analysis, we suggest that you contact us at an early stage.

You can reach technical support services on the web, by telephone, or e-mail.

Web

Go to the MSC.Software website at **www.mscsoftware.com,** and click on **Support**. Here you can find a wide variety of support resources including application examples, technical application notes, training courses, and documentation updates at the MSC.Software Training, Technical Support, and Documentation web page.

Phone and Fax

United States
Telephone: (800) 732-7284
Fax: (714) 784-4343

Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Rome, Italy
Telephone: (390) (6) 5 91 64 50
Fax: (390) (6) 5 91 25 05

Moscow, Russia
Telephone: (7) (095) 236 6177
Fax: (7) (095) 236 9762

Frimley, Camberley Surrey, United Kingdom
Telephone: (44) (1276) 60 19 00
Fax: (44) (1276) 69 11 11

Tokyo, Japan
Telephone: (81) (03) 6911 1200
Fax: (81) (03) 6911 1201

Paris, France
Telephone: (33) (1) 69 36 69 36
Fax: (33) (1) 69 36 45 17

Gouda, The Netherlands
Telephone: (31) (18) 2543700
Fax: (31) (18) 2543707

Madrid, Spain
Telephone: (34) (91) 5560919
Fax: (34) (91) 5567280

Email

Send a detailed description of the problem to the email address below that corresponds to the product you are using. You should receive an acknowledgement that your message was received, followed by an email from one of our Technical Support Engineers.

Patran Support	mscpatran.support@mscsoftware.com
MSC Nastran Support	mscnastran.support@mscsoftware.com
Dytran Support	mscdytran.support@mscsoftware.com
MSC Fatigue Support	mscfatigue.support@mscsoftware.com
Marc Support	mscmarc.support@mscsoftware.com
MSC Institute Course Information	msctraining.support@mscsoftware.com

Training

The MSC Institute of Technology is the world's largest global supplier of CAD/CAM/CAE/PDM training products and services for the product design, analysis, and manufacturing markets. We offer over 100 courses through a global network of education centers. The Institute is uniquely positioned to optimize your investment in design and simulation software tools.

Our industry experienced expert staff is available to customize our course offerings to meet your unique training requirements. For the most effective training, The Institute also offers many of our courses at our customer's facilities.

The MSC Institute of Technology is located at:

2 MacArthur Place
Santa Ana, CA 92707
Phone: (800) 732-7211
Fax: (714) 784-4028

The Institute maintains state-of-the-art classroom facilities and individual computer graphics laboratories at training centers throughout the world. All of our courses emphasize hands-on computer laboratory work to facility skills development.

We specialize in customized training based on our evaluation of your design and simulation processes, which yields courses that are geared to your business.

In addition to traditional instructor-led classes, we also offer video and DVD courses, interactive multimedia training, web-based training, and a specialized instructor's program.

Course Information and Registration. For detailed course descriptions, schedule information, and registration call the Training Specialist at (800) 732-7211 or visit www.mscsoftware.com.

Internet Resources

MSC.Software (www.mscsoftware.com)

MSC.Software corporate site with information on the latest events, products, and services for the CAD/CAE/CAM marketplace.

1 Overview of MSC Nastran 2007 r1

- Overview

Overview

The MSC Nastran 2007 r1 release brings powerful new features and enhancements in the areas of high performance computing, nonlinear analysis, assembly modeling, optimization, rotor dynamics and aeroelasticity.

Implicit Nonlinear Analysis (SOL 600)

- Heat Transfer. Advanced thermal analysis is now available in SOL 600 including an efficient hemi-cube viewfactor calculation method and automated procedures for thermal stress analysis based upon the heat transfer simulation. Thermal analysis of composites may include accurate calculation of the thermal gradient through the thickness.
- Modeling Enhancements. Connector technology has been enhanced to include large deformation formulations of CFAST, CWELD and CBUSH. Enhancements for facture mechanics include the calculation of stress intensity factors using either the VCCT or Lorenzi method and the prediction of delamination.
- Performance Improvements. A new streaming input option is available that eliminates transfer files. The analysis of composite shells has been improved such that assembly time has been often reduced by a factor of 10 and memory requirements have also substantially been reduced. Other performance improvements include an out-of-core iterative and direct PLOAD4 support.

NVH & Acoustics

- Rigid Porous Absorber. A new capability for modeling basic rigid skeleton porous absorber characteristics in acoustic response analysis such as vehicle seats and absorber linings.

Numerical Enhancements

- Sparse Solvers. Two new sparse solvers have been introduced; TAUCS (statics) and UMFPACK (unsymmetric). UMFPACK solver provides scalable performance for unsymmetric frequency response problems. In addition, Lanczos has been enhanced to take advantage of available memory. Automatic optimal reordering selection has been implemented for solid models to eliminate having the user set flags.
- Iterative Solvers. Restrictions to the CASI iterative solver have been relaxed for statics of large solid models (engines). This includes an expanded list of supported element types.
- ACMS**.** Automated Component Modal Synthesis (ACMS) has been extended to External Superelements to provide significant reductions in compute time, I/O and scratch space. One typical case study demonstrates an order of magnitude improvement.
- Other HPC enhancements. MSC Nastran 2007 has been ported to Microsoft Compute Cluster. The compute kernels for x86_64 platforms have been optimized for both Intel and AMD based systems. Improved user diagnostics provides a pivot ratio bar chart to localize model singularities.

Elements & Connectors

- Connectors. A new seamweld (CSEAM) connector element is now available for assembly modeling. It features extended capabilities for connecting higher order elements, mesh independent connections to top/bottom shell patches defined either by property IDs or Element IDs, tailored parts connection, and support for anisotropic material properties. For spot weld elements (CWELD, CFAST) end point displacement output can be obtained to view the relationship between the spot weld and the connecting shells. A new connector type RBE2GS is introduced to optionally search and connect independent grids of the two closest RBE2 elements with a specified search radius.

Optimization

- Topology Optimization. This release provides combined topology, sizing and shape optimization simultaneously to find possible better designs. Different mass targets can now be applied on multiple design parts of the structure. Symmetry constrains have been extended to cyclical applications such as car wheels. An adjoint design sensitivity analysis method has been implemented for inertia relief sizing optimization. Significant performance enhancement with minimum member size control has been achieved particularly for large number of sizing design variables.
- Automatic External Superelement Optimization (AESO)**.** This new feature automatically partitions the model into a designed and non-designed part (external Superelement) for efficient optimization. Order of magnitude speedup can be achieved without requiring user knowledge of superlements.
- Randomization (Pre-release)**.** This randomization capability provides a way to stochastically introduce uncertainty into a model such as tolerances in connectivity, properties and loads. The user selects the outputs to monitor. This beta capability is a first step in developing a multi-run environment to spawn multiple jobs, collect the results and perform statistical post-processing.

Rotor Dynamics & Aeroelasticity

- Rotor Dynamics. Unbalance loading can now be used for frequency response with the rotor dynamics option. Frequency response case control can be used directly in SOL 146 since we can now handle multiple RGYRO subcases in rotor dynamics. Damping specification has been simplified and allow for new damping formulations such as hybrid damping. As a prerelease only capability: the effects of rotor stiffness, mass, and damping effects can be included in SOL 200 optimization.
- Aeroelasticity. Monitoring points can now be updated and summed. A new type of monitoring point (MONCNCM) has been introduced for monitoring stripwise aerodynamic results such as lift and pitching moments. Other enhancements include various splining techniques for aerodynamic structural applications.

List of MSC Nastran Documents Released with MSC Nastran 2007 r1

Along with this Guide, the following documents are updated for the MSC Nastran 2007 release.

- MSC Nastran 2007 Quick Reference Guide
- MSC Nastran Installation and Operations Guide
- MSC Nastran Implicit Nonlinear (SOL 600) User's Guide

2 Implicit & Explicit Nonlinear Analysis

- MSC Nastran Implicit Nonlinear - SOL 600

MSC Nastran Implicit Nonlinear - SOL 600

The MSC Nastran 2007 r1 release contains significant enhancements to functionality and performance. This includes the addition on heat transfer capability, support of Nastran fastener technology (CBUSH, CFAST and CWELD) in a native mode, improved support of PLOAD4, and an enhanced composite capabilities. Fracture mechanics may also be performed to obtain the stress intensity factors at a crack. A new mechanism has been added for the direct transfer of data that is known as streaming input.

Heat Transfer

For heat transfer, most of the capabilities in Nastran SOL 153 and 159 are supported by SOL 600 with the exception of CHBDYP and forced convection, the equivalents of which are not currently available in Marc. The main advantage of using SOL 600 for heat transfer over SOL 153 or 159 is that thermal contact is available directly and that radiation view factors may possibly be calculated faster. The user needs to weight the drawbacks of not having CHBDYP and forced convection. Because of these alternatives, SOL 600 offers two ways to perform a heat transfer analysis. The direct (new) method uses Marc to perform all of the calculations and can support thermal contact that varies during the run. The other (indirect) method is to calculate the thermal contact conditions (if they are needed) at the start of the run and perform the rest of the calculations using Nastran SOL 153/159. This option is addressed using a new option on the SOL 600 entry TSOLVE=M or TSOLVE=N respectively. A typical SOL 600 Executive Control statement for heat transfer using the direct method would be:

SOL 600,153 TSOLVE=M

A typical Executive Control statement for heat transfer using the indirect method would be:

SOL 600,153 TSOLVE=N

To use the "Thermal Contact" capability released with the Nastran 2005 version either TSOLVE=N should be used or the TSOLVE option should be left blank (which will support most existing input decks).

SOL 600 heat transfer addresses conduction, free convection, radiation to space, cavity radiation, thermal contact and latent heat. Steady state or transient heat transfer calculation may be obtained. All material properties may be temperature dependent, and the material may be isotropic, orthotropic or anisotropic. For the direct method, Marc's table input is used for all applicable input items. The direct method requires postprocessing using the Marc t16 file. All standard output forms (op2, xdb, f06 and/or punch) are available using the indirect method.

The temperature history obtained may then be used in a subsequent thermal stress simulation by using the MINSTAT and MCHSTAT bulk data options to read the temperatures off the t16 file. When used in conjunction with the MTHERM bulk data option the time steps will be either subdivided or merged to satisfy the accuracy and convergence requirements of the nonlinear mechanical analysis.

For the directly solutions, when CTRIA3 or CQUAD4 elements are used, the thermal conduction can be based upon either two methods which is selected on the PSHELL option. Similar to conventional Nastran, the thermal behavior may be membrane like only, in which case there is no thermal gradient

through the thickness. To support this, new heat transfer elements have been added and are used when appropriate. These new elements are:

Element 196 — Three-node, Bilinear Heat Transfer Membrane), 904

Element 197 — Six-node, Biquadratic Heat Transfer Membrane), 907

Element 198 — Four-node, Isoparametric Heat Transfer Element), 911

Element 199 — Eight-node, Biquadratic Heat Transfer Membrane, 915

The second method is that the element has a thermal gradient through the thickness, which may be required for composite simulation or thermal shock type problems. This is activated by specifying a nonzero MID2 entry. The MPHEAT options is used to specify, whether the temperature gradient is linear or quadratic through the total thickness of the shell, or linear or quadratic variation is specified per layer basis. In the later case if a composite shell has n layers the number of degrees of freedom per grid is n+1 or 2*n+1 for the quadratic case.

The MHEATSHL parameter may also be used to control this behavior.

Additions/changes to a standard Nastran SOL 153 or SOL 159 heat transfer input file are as follows:

Executive Control

Change SOL statement as described above.

Case Control

No changes

Bulk Data

BCBODY, BCTABLE (BCPARA if necessary to change defaults) if there is thermal contact

MPHEAT – New entry that maps to Marc's HEAT "parameter". See, "MPHEAT (SOL 600)" on page 1738 of the MSC Nastran Quick Reference Guide for more information.

NLHEATC - Defines numerical analysis parameters for SOL 600 Heat Transfer Analysis. For more information please see, "NLHEATC (SOL 600)" on page 1756 of the MSC Nastran Quick Reference Guide.

MCHSTAT - Option to change state variables for SOL 600 – Used in SOL 600 only. For more information please see, "MCHSTAT (SOL 600)" on page 1685 of the MSC Nastran Quick Reference Guide.

MINSTAT - Option to define initial state variables for SOL 600 – Used in SOL 600 only. Please see, "MINSTAT (SOL 600)" on page 1700 of the MSC Nastran Quick Reference Guide for more information.

Bulk Data Parameters

PARAM,MARCHEAT is obsolete starting with this release.

PARAM,MARHTPRT	(Integer) Control heat transfer output in the Marc .out file 0 = Do not print any output except for summary tables 1 = Print the nodal temperatures 2 = Print all possible nodal heat transfer output
PARAM,MRADUNIT	(Integer) Controls the units used in radiation heat transfer for SOL 600 1 = Degrees Celsius 2 = Degrees Kelvin (default if parameter not entered) 3 = Degrees Fahrenheit Remark: Degrees Rankin are not available
PARAM,MHEMIPIX	(Integer) Controls the number of pixels used in radiation heat transfer for SOL 600 using the hemi-cube method. The default, if this parameter is not entered is 500.
PARAM,MARVFCUT	(Real) Controls the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored. Default is 0.0001 if this parameter is not entered (Used in SOL 600 radiation heat transfer only)
PARAM,MRVFIMPL	(Real) Controls the fraction of the maximum view factor that is to be treated implicitly (contribute to operator matrix). View factor values smaller than this cutoff are treated explicitly. Default is 0.01 if this parameter is not entered using this parameter reduces the size of the heat transfer operator matrix, which reduces the computational costs associated with decomposition. (Used in SOL 600 radiation heat transfer only)
PARAM,MRSTEADY	(Integer) Controls the solution method for SOL 600 steady state heat transfer 1 Marc STEADY STATE is used with TIME STEP of 1.0 (default if parameter not entered) The specific heat matrix is not formed. 2 AUTO STEP is used.
Remark:	Requires that a sufficiently large time period to be simulated for the solution to reach steady state.

Bulk Data

BCBODY, BCTABLE (BCPARA if necessary to change defaults) if there is thermal contact. Please see the *MSC Nastran Quick Reference Guide* for more details on entries, DMIGOUT, MCHSTAT, MINSTAT, MTHERM, MPHEAT, NLHEATC.

Heat Transfer Examples

The following heat transfer examples are located in the tpl1 directory:

Conduction	mhqbd1, mhqbd1a, mhqbd1c, mhqbd1s, mhqbd2, mhqbd2c, mhqbd2s, mhbc01, mhbc02, mhtepe
Free Convection	mhcbv1, mhcbv1a, mhcbv1b, mhcbv1c, mhcbv1d, mhcbv1e
Radiation to Space	mhrad1, mhrad2, mhrad3
Cavity Radiation	mhrcv1, mhrcv1a, mhrcv2, mhrhx0, mhrdhx, mhrhx4, mhrc1t
Thermal Contact	mhcnoc, mhtc07, mhtc7a
Latent Heat	mtlat1, mtlat2

Creep Simulations

Creep is an important phenomena in high temperature applications. To facilitate this type of analysis several new options have been added into SOL 600 in the MSC Nastran 2007 r1 release.

The MPCREEP allows you to select the procedure used to perform the time integration.If the explicit method is used, then the time step must be small, but is the material is elastic and small deformation, no reassembly of the stiffness matrix is required. If the implicit method is chosen, then larger time steps may be used, but reassembly occurs at every increment.

In thermal creep simulations it is necessary that the time step be chosen to satisfy accuracy of both the rate independent thermal stress problem and the rate dependent creep problem. To insure that this occurs the MTCREEP bulk data option has been introduced.

Element Selection

To increase the flexibility in performing simulations the MRALIAS param has been augmented by the ALIASM bulk data entry. This allows one to map the Marc element type to be used for a selection of elements. All Marc element types may be used if they are topologically similar to the Nastran element type. See Marc Volume B for greater details.

Other SOL 600 Items

Membrane Elements

SOL 600 now directly supports membrane-only shells without the user having to add any alias bulk data entries or parameters. This was made possible due to a new element which was added in Marc:

Element 200 — Six-node, Biquadratic Isoparametric Membrane), 920

This new element completes the set of shell elements necessary to support membranes. Users should be careful when using membranes with nonlinear analyses as they are unstable under certain conditions because of a lack of bending stiffness.

Fracture Mechanics

New capabilities have been added to SOL 600 to allow greater insight in determining the fracture resistance of your designs. These methods complement the material damage models that existed previously and are entered through the MATHED option. The new capabilities include:

- Calculation of Energy Release Rates and Stress Intensity Factors – VCCT and LORENZI
- Option
- Crack Propagation – VCCT
- Delamination – COHESIV
- New Failure Criteria – MATF
- Birth and death of elements

Virtual Crack Closure Technique (VCCT)

Marc's revised VCCT capability is fully supported by this release of SOL 600. It involves a anew VCCT Case Control entry and a new VCCT Bulk Data entry (see the MSC Nastran Quick Reference Guide for details). This option defines that the virtual crack closure technique is to be used for evaluating energy release rates. The user defines the node (in 2-D or for shells) or nodes (in 3-D) that define each crack tip. The supported elements are lower- and higher-order 2-D solids and 3-D shells, lower- and higher-order 3-D hexahedral solids and lower order 3-D tetrahedral solids. For 3-D solids it is important that a regular mesh around the crack front is used.

Multiple cracks can be defined and results obtained for each crack separately. Each crack consists of a crack tip node in 2-D and for shells and a list of nodes along the crack front for 3-D solids. Shell elements can be used for defining a 2-D style line crack and also be connected to the face of another shell or 3-D solid to form a 3-D style surface crack. The different cases are automatically identified.

The VCCT method is advantageous because it may be used with any material model including orthotropic or anisotropic behavior, and because it automatically obtains the mode I, II, and III stress intensity factors. This makes is applicable to composite structures.

For crack propagation, there are two modes of growth: fatigue and direct growth. For fatigue style, the user specifies a load sequence time period. During the load sequence, the largest energy release rate and the corresponding estimated crack growth direction is recorded. At the end of the load sequence, the crack is grown using the specified method. For direct growth, the crack grows as soon as the calculated energy release rate is larger than the user-specified Gc. Note that Gc can be made a function of the accumulated crack growth length to model a crack growth resistance behavior. This release does not support large crack propagation which requires remeshing.

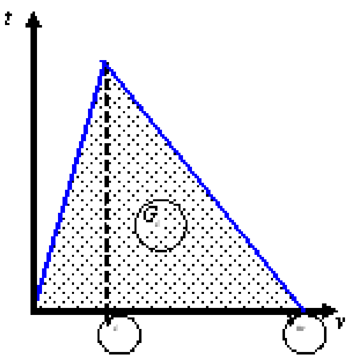
Fracture Mechanics J-Integral (LORENZI)

This option gives an estimation of the J-Integral for a crack configuration using the domain integration method. The domain integration method has the advantage that it can also be used for problems with thermal behavior and for dynamic analysis. This procedure is only available for continuum elements. Only the nodes defining the crack front (crack tip in two dimensions) need to be defined. The program automatically finds integrations paths according to the format below. The complete J-Integral is evaluated and output. For the case of linear elastic material with no external loads on the crack faces, the program automatically separates mode I, mode II, and mode III (3-D only) stress intensity factors from the J-Integral. for isotropic materials.

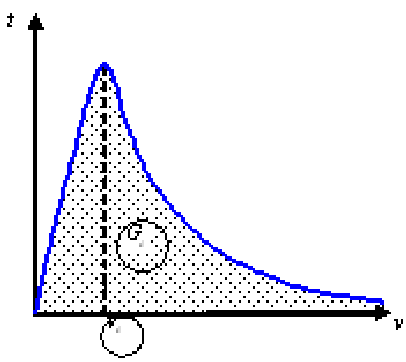
A new Bulk Data entry, LORENZI, is necessary to activate this capability and if entered applies to all subcases in the analysis. For more details the *MSC Nastran Quick Reference Guide*.

Delamination

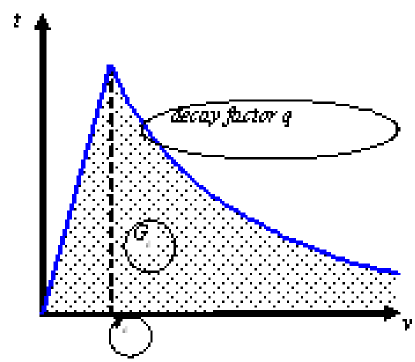
An alternative method to model failure is to use the COHESIV bulk data option in conjunction with special delamination or interface elements. Three different models are available along with a user subroutine. The user defines the traction versus the relative separation. The area under the curve is the cohesive energy, often known as the critical energy release rate.



Bilinear model



Exponential Model



Linear-Exponential Model

Element Type	Number of Nodes	Characteristic
186	4	Planar
187	8	Planar
188	8	3-D
189	20	3-D

Element Type	Number of Nodes	Characteristic
190	4	Axisymmetric
191	8	Axisymmetric
192	6	3-D
193	15	3-D

MATEP Extensions

Material description MATEP was extended to add Chaboche, Power Law, Kumar, Johnson Cook and other options. For more details the *MSC Nastran Quick Reference Guide*.

SOL 600 Failure Description – MATF

For SOL 600 failure indices or actual material failure is only described using the MATF entry. For this release, MATF has been totally revised to accommodate additional types of failure and improve the input and user understanding of the input. The user should be aware that other Nastran solutions can specify failure index calculation on various MAT entries. These specifications are not available in SOL 600 – only MATF may be used. To activate the new entries, the PARAM,MRMATFSB, 1 must also be included. For more details on the MATF entry, see the *MSC Nastran Quick Reference Guide*.

Primary Format (SOL 600)

1	2	3	4	5	6	7	8	9	10
MATF	MID	ITYPE	SB						
	"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	1st
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	
	"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	2nd
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	
	"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	3rd
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Primary Format Example:

1	2	3	4	5	6	7	8	9	10
MATF	100	0							
+	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
+	4500.	4500.	4500.						
+									
+	CRI	2							2nd
+									
+	.11	.06	.1	.05	.075	.03	.03	.03	
+	CRI	4	2500.	4000.	2500.	4000.	2000.	3000.	3rd
+	4500.	4500.	4500.	0.90					

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criteria has been omitted)

Alternate Format (SOL 600 Only)

1	2	3	4	5	6	7	8	9	10
MATF	MID	CRI67	Xt	Xc	Yt	Yc	Zt	Zc	
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Example Alternate Format

1	2	3	4	5	6	7	8	9	10
1	2	3	4	5	6	7	8	9	10
MATF	100	1	2500.	4000.	2500.	4000.	2000.	3000.	
	4500.	4500.	4500.						

Field	Contents
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry (Integer>0, no default). See Remarks 1 and 2.
ITYPE	Flag to invoke progressive failure (Integer, default=1 for SOL 600)
	0 No progressive failure compute failure indices only (Default)
	2 Progressive failure (presently available only for Hashin and Puck methods)
SB	Allowable shear stress of the bonding material between layers (composites only) (Real, No default)
"CRI"	For the alternate format, enter the character string "CRI" to start each criteria (Character, Required)
CRI67	Used with the alternate format only (Integer, no default, required). It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows:
	1000000*ITYPE+10000*C3+100*C2+C1
	Where C1, C2, C3 are integer values for the various Criteria listed below.
Criteria	Select an integer corresponding to the failure criteria to be applied (integer, no default) Up to three failure criteria may be specified for each MID for 600. Only one failure criteria may be used for the primary format.

Field	Contents
	1. Maximum stress criterion. (SOL 600 only, See Remark 3) 2. Maximum strain criterion. (SOL 600 only, See Remark 4) 3. Hill failure criterion. (SOL 600 only, See Remark 5) 4. Hoffman failure criterion. (SOL 600 only, See Remark 6)Tsai-Wu failure criterion. (SOL 600 only, See Remark 7) 5. Hashin failure criteria (SOL 600 only, remark 8) 6. Puck failure criteria (SOL 600 only, remark 11) - must not be combined with any other method 7. User defined failure criteria (SOL 600 only, remark 12) 8. Hashin-Tape (SOL 600 only, remark 13) 9. Hashin-Fabric (SOL 600 only, remark 14)
Xt	Maximum tensile stress in x-direction (Real>0. or blank)
Xc	Maximum compressive stress (absolute value) in x-direction (Real>0., default=Xt)
Yt	Maximum tensile stress in y-direction (Real>0., default=Xt)
Yc	Maximum compressive stress (absolute value) in y-direction (Real>0., default=Yt)
Zt	Maximum tensile stress in z-direction (Real>0., default=Xt)
Zc	Maximum compressive stress (absolute value) in z-direction (Real>0., default=Zt)
Sxy	Maximum shear stress in xy-plane (Real>0. or blank)
Syz	Maximum shear stress in yz-plane (Real>0., default=Sxy)
Szx	Maximum shear stress in zx-plane (Real>0., default=Sxy)
Find	Failure index (Real>0., default=1.) See Remarks 5-7.
Fxy	Interactive strength constant for xy-plane (Real<0., default=). $-\frac{1}{2}\sqrt{\frac{1}{X_t X_c}\frac{1}{Y_t Y_c}}$
Fyz	Interactive strength constant for yz-plane (Real<0., default=). $-\frac{1}{2}\sqrt{\frac{1}{Y_t Y_c}\frac{1}{Z_t Z_c}}$
Fzx	Interactive strength constant for zx-plane (Real<0., default=). $-\frac{1}{2}\sqrt{\frac{1}{Z_t Z_c}\frac{1}{X_t X_c}}$
Ext	Maximum tensile strain in x-direction (Real>0. or blank)
Exc	Maximum compressive strain (absolute value) in x-direction (Real>0., default=Ext)

Field	Contents
Eyt	Maximum tensile strain in y-direction (Real>0., default=Ext)
Eyc	Maximum compressive strain (absolute value) in y-direction (Real>0., default=Eyt)
Ezt	Maximum tensile strain in z-direction (Real>0., default=Ext)
Ezc	Maximum compressive strain (absolute value) in z-direction (Real>0., default=Ezt)
Gxy	Maximum shear strain in xy-plane (Real>0.)
Gyz	Maximum shear strain in yz-plane (Real>0., default=Gxy)
Gzx	Maximum shear strain in zx-plane (Real>0., default=Gxy)

Element Birth and Death

Starting with this release it is possible to deactivate and re-activate elements in the model that have failed or for some other reason needs to be deactivated or re-activated. This is accomplished using Case Control commands DEACTEL and ACTIVAT as well as matching Bulk Data entries DEACTEL and ACTIVAT. Once an element is deactivated or activated it stays that way during the entire subcase case unless it fails due to a MATF criteria. Please see the *MSC Nastran Quick Reference Guide* for further details.

Unglue

Frequently in contact analysis it is known beforehand that two surfaces will never separate once they contact. To prevent numerical chattering contact between these surfaces is frequently described using glued contact. In order to perform VCCT analysis of such surfaces it might be necessary to unglue those nodes near a crack. A new Bulk Data entry, UNGLUE, is available for such purposes. Please see the *MSC Nastran Quick Reference Guide* for further details.

Composite Element Numerical Analysis

In previous versions, SOL 600 provided two options for composite analyses (1) complete through the thickness integration at every iteration and (2) the “smeared” approach as used in other Nastran solution sequences. The first approach is more accurate particularly for nonlinear analyses where local buckling takes place and the analysis needs to extend well into the post-bucking regime. The second approach is usually satisfactory for small deformation linear static and dynamic analyses. Method 1, complete integration through the thickness has been modified such that the accuracy has been retained, but the computational times and memory requirements have been significantly reduced. These are known as “fast integration” techniques and are described by the new Bulk Data entry PCOMPF. The limitation is that using these fast integration procedures the material may not exhibit any nonlinear behavior. Large deformation and buckling is supported using these procedures.

The following table indicates typical performance improvements with this release when using this option.

Model	Number of Element	Maximum Number of Layers	Improvement in CPU for Stiffness Matrix	Memory Reduction
1	10000	46	1425 %	1550 %
2	88854	182	2013 %	957 %
3	48858	33	1085 %	795 %
4	420015	48	1263 %	886 %

Please see the *MSC Nastran Quick Reference Guide* for further details.

PLOAD4 Extensions

SOL 600 versions previous to this one did not support the PLOAD4 continuation line. In addition, if corner pressures with different values were entered, they were averaged. SOL 600 now fully supports different corner pressures, pressures specified by the CID, N1, N2, and N3 fields on the continuation line and line loads specified by the CID, N1, N2, and N3 fields. The SOLR field is fully supported. The LDIR field is not supported. Line loads must be specified using the CID, N1, N2, and N3 fields rather than LDIR. For SOL 600, the SORL field applies to CQUAD4 and CTRIA3 as well as CQUAD4R and CTRIAR elements. The CID field may reference an ID of any CORD1R, CORD1C, CORD1S, CORD2R, CORD2C or CORD2S entry but not CORD3G.

All of these new PLOAD4 extensions are activated by entering PARAM,MRPLOAD4,2 in the bulk data or by placing this parameter in one of the RC files. For this release these capabilities are not the default and must be activated using this parameter. For the next release, it is anticipated that MRPLOAD4=2 will become the SOL 600 default.

Large Rotation RBE

Improved large motion RBE capabilities have been added to SOL 600, however it was decided to retain the small rotation formulation as the default. This will allow models used with previous versions to obtain the same results. To activate the large RBE rotation capability, add the following parameter to the bulk data:

```
PARAM,marc7601,1
```

Streaming Input

A new capability available in this release is known as “streaming input”. Normally SOL 600 will form a Marc input file from the Nastran input file, then execute Marc to compute the results. With “streaming input” Marc is not executed. Instead, the Marc subroutines which are now in Nastran are used directly and called by the main SOL 600 routine. The Marc input file is still formed and saved on disk for possible

future use by the customers, but the same information is passed in memory from the main SOL 600 routine to the Marc initiation routine, thus saving computer time because a physical Marc input file on disk does not need to be opened and read. Streaming input is activated using the Bulk Data entry:

```
PARAM,MRSTREAM,1
```

Streaming input is not available with DDM (parallel processing) or if user subroutines are necessary. For those cases, do not include param,mrstream and run SOL 600 as before. Streaming input should also not be used if a special version of Marc is necessary for your particular application. In that case, specify one of the PATH options on the SOL 600 Executive statement in conjunction with a file to point the analysis to the location of the version of Marc that is desired.

CONNECTOR TECHNOLOGY

CBUSH, CWELD and CFAST have been added to Marc as nonlinear (large deformation and rotation) elements. These formulations are now available in SOL 600 using PARAM,MARCWELD (see the QRG for additional details). The user should beware that CBUSH, CWELD, CFAST, RBE2, RBE3 in SOL 600 and Marc are truly nonlinear elements while in Nastran they are linear small deformation elements. Thus different results will sometimes be obtained. In addition, solutions that converge with SOL 106, or 129 may not converge with SOL 600 (the opposite may also occur). The user should be careful when using such elements to make sure they are applicable to both linear and nonlinear solutions when constructing the model.

The new formulation of the CBUSH is activated using the MARCBUSH,-1, param. If CBUSH entries are used with a nonzero CID, then the MRCOORDS,0, param should also be included.

The new formulation of the CFAST and CWELD is activated using the MARCWELD,1 param.

The other new parameters that control CWELD behavior are:

- MARIPROJ
- MRCWANGL
- MRFACEA
- MRFACEB
- MRHERRMN
- MRITTYPE

The following CWELD parameters are not supported in SOL 600

- CWDIAGP
- CWRANDEL

The following CWELD parameters are supported in SOL 600

- CWLDIGNR

If the new CWELD is used, then MSPEEDCW is ignored

The following CFAST parameters are not supported in SOL 600

- CFDIAGP
- CFRANDELT

COMPUTATIONAL ENHANCEMENTS

The Direct Iterative Solver (MARCSOLV=2) can now be used with out-of-core assembly to allow larger models to be analyzed. It should be noted that there is a decrease in performance. The program will automatically use the out-of-core option if necessary or you can select it by using the MARCOOCC param.

SUPER ELEMENTS / DMIG

The use of super elements in conjunction with SOL 600 has increased over the last year, and several changed have been made in this area. In addition to the MDMIOUT that may be used to create a reduced stiffness matrix (super element) or an Adams MNF file, the DMIGOUT option has been added.

The DMIGOUT option may be used to output either the complete global matrices or individual element matrices at the element levels. These global matrices include:

- Stiffness matrix – This is the total stiffness matrix, including geometric stiffness, follower force and friction contributions.
- Differential stiffness – This is only available in a buckle sub case
- Mass matrix – Available in dynamics
- Damping matrix – Available in dynamics
- Conductivity matrix – Available in heat transfer
- Specific Heat matrix – Available in transient heat transfer

The element matrices correspond to the above and can be output in either the basic system or in the transformed system. The global stiffness matrices are always output in the transformed system.

It should be noted that the output of these matrices may be huge. To reduce the size of the DMIG files it is possible to filter out small values. It should be noted that doing this may influence subsequent calculations.

CONTACT ENHANCEMENTS

The spline option used with deformable bodies via the BCBODY option can now be used with higher order elements. This improves the calculation of when contact is to occur and the normal to the surface.

New SOL 600 Parameters

The following new SOL 600 parameters have been introduced as described below. Please see the MSC Nastran Quick Reference Guide for more details:

PARAM,MARMTLCK	Determines whether a check of various property-material combinations for SOL 600 will be made or not. This slightly slows down the input processing.
PARAM,MARLDCMB	Determines whether extraneous loads in the input file will be combined to save computer time.
PARAM,MARLDRMV	Determines whether extraneous FORCE, MOMENT and/or PLOAD4 entries in the input file will be filtered out at an early stage to save computer time.
PARAM,MARNOCID	SOL 600 by default does not support MCID defined by cylindrical or spherical coordinate systems. This parameter determines whether MCID defined by cylindrical or spherical coordinate systems will be ignored or "fataled out" for shell and solid elements depending on this parameter. Inclusion of this parameter overcomes this problem.
PARAM,MRCPENTA	This entry determines how CPENTA will be mapped to Marc degenerate solid elements. Marc does not presently have wedge elements, so CPENTA elements must be mapped to degenerate hexa elements such as type 7.
PARAM,MRHERRMN	This entry controls whether extra grids created for such items as hyperelastic Herrmann elements, CWELD, etc. are output or not in the op2, f06, punch and/or xdb files. When Herrmann grids are output, the displacement value is actually pressure which might be confusing when looking at an f06 file.
PARAM,MARCMID3	This entry controls whether MID3 will be set to the same value as MID2 when the Marc PSHELL option is used (designated by PARAM,MRPSHELL,1 or when the SMEAR option is used on the SOL 600 Executive Control statement)
PARAM,MFORDUPE	This entry controls how duplicate forces encountered for the same load case are handled in SOL 600.
PARAM,MARBK105	This entry controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,105.
PARAM,MRCTRIA3	This entry controls the "default" Marc element type for CTRIA3 elements (75 or 138) in SOL 600
PARAM,MRCQUAD4	This entry controls the "default" Marc element type for CQUAD4 elements (75 or 139) in SOL 600

PARAM,MARCDUPE	This entry controls whether SOL 600 will check for duplicate entries for most every type of bulk data card. SOL 600 does not allow duplicate entries, but the portion of IFP that runs prior to spawning Marc does not usually check for duplicate entries.
PARAM,MARCSTOP	This entry controls whether a check model run will be performed, no actual simulation will occur, and the analysis will stop with a Marc Exit 7.
PARAM,MRRSTOP2	Normally op2, xdb, punch and f06 output is not available for SOL 600 restart analyses. Setting this parameter to 1 will allow the program to attempt to create one or more of these files. Only in limited cases will the job be successful.
PARAM,MRBDYCVT	Determines if CHBDYG is converted to CHBDYE for SOL 600 heat transfer
PARAM,MRMATFSB	Determines if the version of Marc being used supports the new MATF SB field or not
PARAM,MRDELTTT	Determines how delta time is set for each "step" of a SOL 600 transient nonlinear analysis.
PARAM,MARCFEAT,N	If entered will add FEATURE,N to the Marc input file in the parameters section
PARAM,MRCOORDS	Determines whether Marc COORD SYSTEM will be added if any CORD1i or CORD2i entries are in the model and if CBUSH elements are present in the model
PARAM,MARCWELD	Determines how CWELD/PWELD and CFAST/PFAST elements will be translated to Marc
PARAM,MRITTYPE	Type of “constraint” used to connect the auxiliary nodes in all CWELDs
PARAM,MRCWANGL	Angle in degrees over which to rotate the CWELD cross-section about the beam axis to obtain its final orientation.
PARAM,MARIPROJ	Flag to determine if auxiliary nodes of a CWELD will be projected on the model
PARAM,MARFACEA	Face number for “A” side of weld if welds are made of solid elements
PARAM,MARFACEB	Face number for “B” side of weld if welds are made of solid elements

Platform Specific Notes

For Linux IA64 and EM64T platforms:

The default MPI for these platforms is HP MPI.

Intel MPI is also supported and can be used by switching to it using the maintain script under the tools directory.

To use the Intel MPI (iMPI), please observe the following.

a. Create a .mpd.conf file in your home directory that contains the following line.

secretword=<your mpd password>

where <your mpd password> can be any arbitrary string.

Change mode of the .mpd.conf to 600, i.e. do a

```
chmod 600 $HOME/.mpd.conf
```

b. Setup a mpd.hosts file in your home directory consists of the names of nodes in your cluster (it can have only 1 node, i.e. 1 line):

clusternode1

clusternode2

clusternode3

The rest is taken care of by the run_marc script.

However, if your cluster requires password to perform ssh or rsh between nodes, you may need to enter your password every time you are running a parallel job. You can disable the password requirement in the use of ssh or rsh. Please consult your system administrator.

For 64-bit Windows EM64T platforms:

The default MPI for this platform is MPICH2. MS MPI is not supported for this release of MSC Nastran for SOL 600.

To install MPICH2, go into the mpichx64\bin directory and type

```
smpd -install
```

For 32-bit Windows platforms:

The default MPI for this platform is Argonne National Lab's MPICH2.

To use the MPICH2 please observe the following:

a. Go into the mpich2\bin directory and type

```
smpd –install
```

If smpd could not be installed, you may need to reboot your system.

b. The first time your run a parallel job, you will be required to enter your login ID and password.

If your system is rebooted or your command prompt is closed, you will need to repeat step b.

Supported Systems for SOL 600 in MSC Nastran 2007

Vendor	OS	Hardware	FORTRAN Version	C Version	Parallel Enabled	Default MPI	Also Works On
HP-Alpha (DEC)[4]	Tru64 5.1	Alpha Server 4100	f90 5.5	cc 6.4	yes	HP MPI 2.0[1]	
HP (64-bit)[2,4] HP (64-bit)[2,4]	HPUX 11.0 HPUX 11.23	PA2.0 Itanium 2	f90 2.9.2 f90 2.8.7	C.03.50 A.06.02	yes yes	HP MPI 2.0 HP MPI 2.2	
IBM (64-bit)[4]	AIX 5.2	RS/6000 & RS/6000 SP	xlf 8.1.1	cc 6.0.0	yes	MPICH[1]	IBM POE 4.1
SGI (mips4 64-bit)[2,3,4] SGI (Altix 64-bit)[2, 4]	IRIX 6.5 Linux 2.4.21-sgi303r2	R12000 Itanium 2 (Propack 3.0)	f90 7.4 Intel 8.1	cc 7.4 Intel 8.1	yes yes	MPICH[1] SGI MPT 1.10.1	 Propack 4.0
Sun (64-bit)[4]	Solaris 2.8	UltraSPARC III	f90 8.1	cc 5.7	yes	MPICH[1]	
Linux (32-bit)	RedHat 9	Intel Pentium or equiv.	Intel 8.1	Intel 8.1	yes	HP MPI 2.2.5	RedHat AS 3.0
Linux (64-bit)[4,5]	RedHat AS 3.0	Itanium 2	Intel 8.1	Intel 8.1	yes	HP MPI 2.2.5[5]	
Linux (64-bit)[4,5]	RedHat WS 3.0	Intel EM64T	Intel 8.1	Intel 8.1	yes	HP MPI 2.2.5[5]	AMD Opteron, RedHat WS 4.0
Intel (32-bit) Intel (64-bit)[4]	Windows 2000 Windows Server 2003 x64	Intel Pentium or equivalent Intel EM64T	Intel 8.1 Intel 8.1	Intel 8.1 Intel 8.1	yes yes	MPICH2 MPICH2	Windows XP, Intel 9.1 MPICH2, Windows XP 64

1 Hardware MPI version also available (via maintain in /tools directory).

2 Supports Solver 6.

3 Supports multi-threading.

4 Supports true 64-bit version.

5 Supports the Intel MPI 3.0

3 NVH & Acoustics

- Rigid Porous Absorber

Rigid Porous Absorber

Introduction

A new capability to model basic rigid skeleton porous absorber characteristics in acoustic response analysis is now available. The capability allows some types of absorbent material to be modelled, such as vehicle seat structures or lining materials which exhibit stiff carcasses. The absorber material is considered using an equivalent fluid analogy and so is modelled in the same manner as a typical fluid, using solid CHEXA, CPENTA or CTETRA elements, the GRID points for which have their CD field set to -1. The porous absorber elements reference a PSOLID property entry with field 8 set to PFLUID in the usual way, with field 3 of the PSOLID entry referencing a MAT10 entry which has been modified with an additional field 7 dedicated to porous absorber materials.

Porous Materials

If a material is not completely solid, but contains voids or air pockets, then it is said to be porous. There are a great many materials which exhibit porosity, the term given to the degree of openness of the material, including materials generally considered “solid” like brick or stone. If the voids in the substance are large enough, they may form an interconnected maze of passage ways allowing air (or any other fluid) to pass through the material. However, depending on the degree of convolution in the passage ways (known as tortuosity), the air will encounter some resistance as it passes through the substance, requiring pressure to be exerted to push the air against the resistance. Sound waves striking the material do not cause air to flow through the material, but they do cause local perturbations that exert pressure and cause the air to move in the vicinity of the material; the oscillating movement of air caused by the sound waves encounters resistance (called impedance) which uses up some of the sound energy and damps the level of sound. This energy is eventually dissipated as heat.

If the porous material is enclosed in a frame which is considered as rigid, as will be the case for example for a porous medium which has a high skeleton density or very large elastic modulus or weak fluid-structure coupling, the porous material can be considered as an equivalent fluid with both density and bulk modulus being complex frequency dependent parameters. It is possible to obtain values for these parameters by empirical methods as introduced, for example, by Delany and Bazley[1], methods which have been widely used to describe sound propagation in fibrous materials.

In the equivalent fluid approach, the equation of motion reads

$$\frac{1}{\rho_e}\nabla^2 P + \frac{\omega^2}{B_e}P = 0$$

where ρ_e is the equivalent density, B_e the equivalent bulk modulus, P the complex pressure amplitude and ω the circular excitation frequency, in which it can be shown

$$\frac{1}{\rho_e} = \frac{1}{\rho}(1 + iGE)$$

and

$$\frac{1}{B_e} = \frac{1}{B} - i\frac{A}{\omega} = \frac{1}{B}\left(1 - \frac{i\alpha}{\omega}\right)$$

if it is assumed that the parameters are not frequency dependent, a reasonable assumption for the study of frequencies in a narrow band. Here, ρ , B and GE are the values of RHO, BULK and GE respectively of the MAT10 entry for the porous absorber material. A new field 7 has been added to the MAT10 entry to allow the value of α , the normalized admittance coefficient[2], to be entered.

Inputs

MAT10 Fluid Material Property Definition

Defines material properties for fluid elements in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C	GE	ALPHA			

The following may be used to calculate the equivalent fluid property values to be entered on the MAT10 entry starting from the complex density and complex bulk modulus describing the rigid porous absorber. If a complex density and complex speed of sound are determined for the porous material, the complex bulk modulus must first be calculated.

MAT10 density ρ

$$\rho = \frac{\rho_r^2 + \rho_i^2}{\rho_r}$$

ρ_r complex density, real part

ρ_i complex density, imaginary part

MAT10 damping coefficient GE

$$GE = -\frac{\rho_i}{\rho_r}$$

MAT10 bulk modulus B

$$B = \frac{B_r^2 + B_i^2}{B_r}$$

B_r complex bulk modulus, real part

B_i complex bulk modulus, imaginary part

MAT10 normalized admittance coefficient

$$\alpha = \omega\frac{B_i}{B_r}$$

Discussion

The implementation implies that if the complex density and bulk modulus are constant, the normalized admittance coefficient is a function of frequency. However, frequency dependent α is not supported, so a reference frequency must be selected. Typically, this frequency will be either in the mid-range of the desired frequency range to be studied, or will correspond to the frequency at a response peak of interest. As frequencies progressively further away from the reference frequency are considered, the response becomes increasingly subject to the limitations of the frequency independent formulation; the extent will depend somewhat on the nature of the problem, and it may be necessary to study discrete frequency bands in order to mitigate against this effect.

The use of a non-zero value in field 7 of the MAT10 entry causes the generation of a damping matrix because the normalized admittance coefficient is multiplied by the imaginary operator *i*. Consequently, the use of modal methods on the fluid are not appropriate and frequency response analysis must be carried out using the direct method, at least for the fluid.

References

1. M.E. Delany & E.N. Bazley, *Acoustical Characteristics of Fibrous Absorbent Materials*, National Physics Laboratory, Aerodynamics Division, NPL Aero Report Ac 37, March 1969.
2. J. Wandinger, *Possible Implementations of Porous Absorbers in Nastran*, MSC internal memo, April 2006.

Example

Consider the following unbounded fluid (air) and porous absorber medium domains as in Figure 3-1. An acoustic source is placed at the location indicated and the acoustic response (pressure) at the centre of the fluid is monitored.

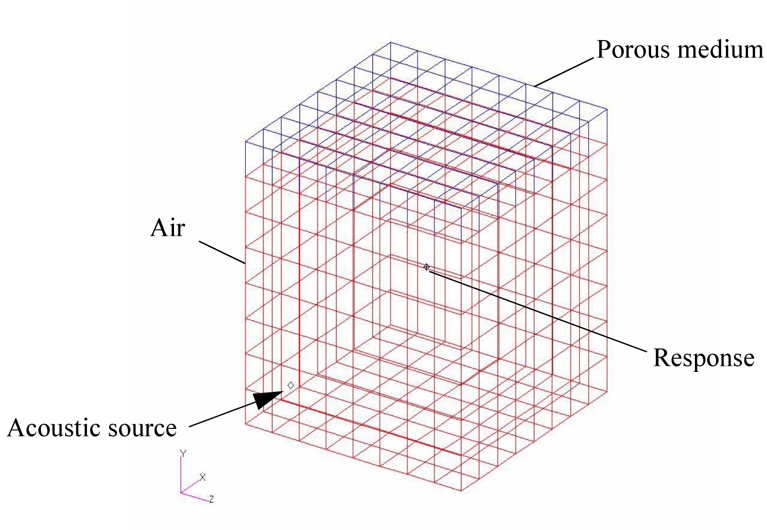


Figure 3-1 Porous medium

The following properties were determined using experimental methods.

Air

Density	1.225 + *i*0
Speed of sound	340.0 + *i*3.4

Porous Absorber

Density	3.8663 + *i*14.2204
Speed of sound	92.7076 + *i*70.2854

From which the following equivalent bulk moduli were obtained

Air

Bulk modulus	141595.8 + *i*2832.2

Porous Absorber

Bulk modulus	-171190.0 + *i*102356.3

A frequency of 250 Hz was selected to calculate the values of alpha for air and the porous absorber. Using the equations illustrated above, the following data is entered on the MAT10 entries.

MAT10 for air

	MID	BULK	RHO	C	GE	ALPHA
MAT10	1	141652.5	1.225		0.0	31.41907

MAT10 for the porous absorber material

	MID	BULK	RHO	C	GE	ALPHA
MAT10	2	-232389.	56.16948		-3.67804	-939.196

Notice that the values of bulk modulus, GE damping coefficient and alpha are all negative; this is a normal characteristic of the implementation.

The response at the centre of the air domain is calculated using the new capability and the results compared with the same model run in Actran. Both HEXA-20 (Figure 3-2) and HEXA-8 (Figure 3-3) elements are compared.

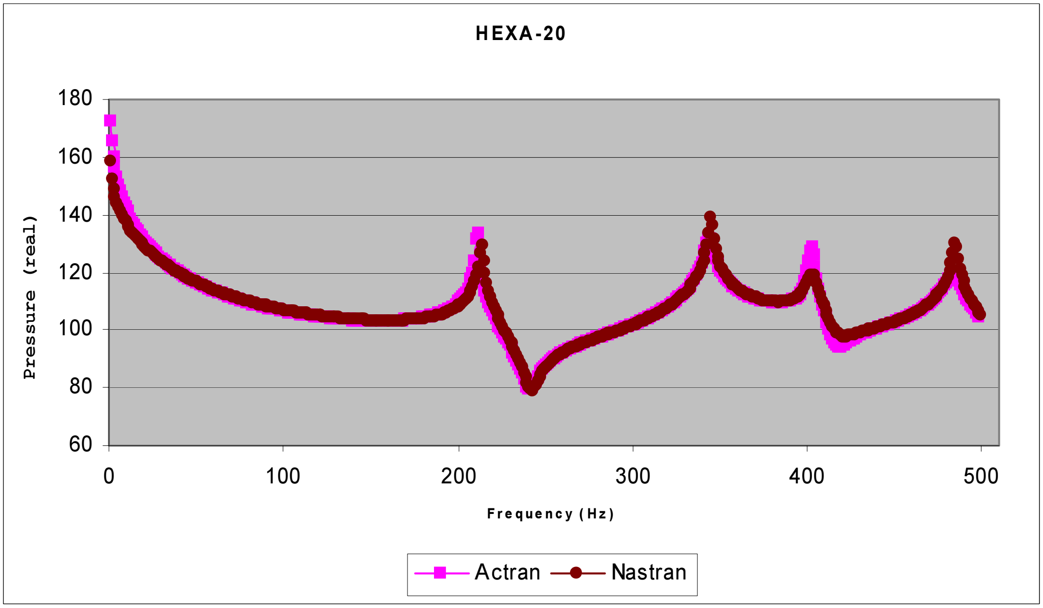


Figure 3-2 HEXA-20

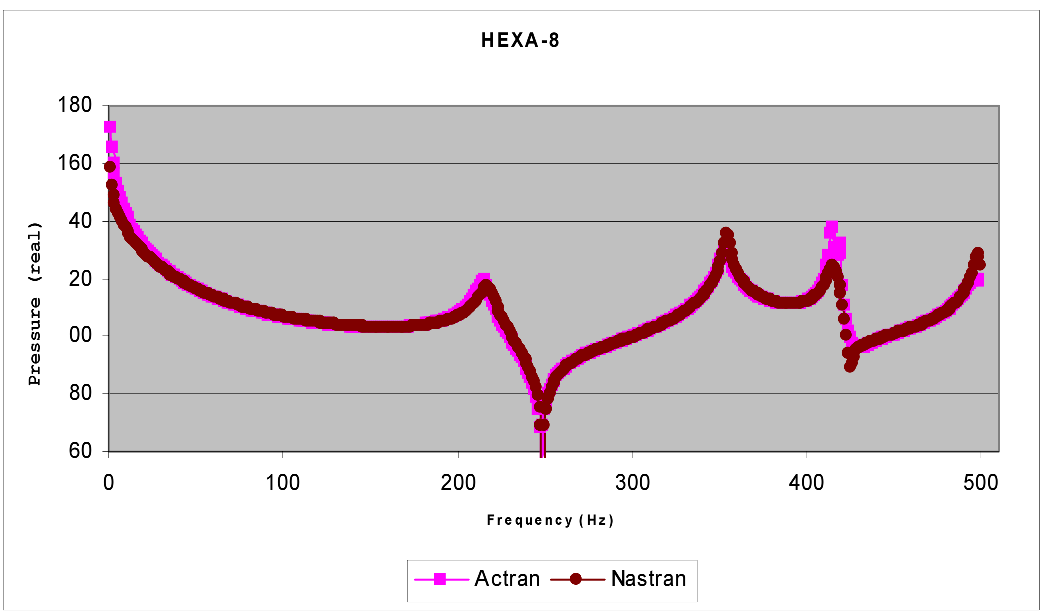


Figure 3-3 HEXA-8

The results compare very favorably with those from Actran. Notice the gradual departure from the expected Actran response for frequencies progressively further away from the reference frequency of 250 Hz, as the response becomes increasingly subject to the limitations of the frequency independent formulation.

4 Numerical Enhancements

- New SPARSESOLVER Executive Statement
- Improved Performance with New Sparse Solvers
- Improved Memory Usage in Lanczos (Pre-release)
- New MAXRATIO Information Output (Pre-release)
- Performance Improvements and Expanded Capabilities for ACMS
- Relaxed Restrictions for CASI Solver Usage
- System Dependent Performance Improvements
- Improved Selection of Reordering Methods

New SPARSESOLVER Executive Statement

Introduction

A new interface, the **SPARSESOLVER** Executive Statement, is now available for analysts to better control the options and processes associated with sparse matrix solution methods. The interface permits specification of the solver to be used as well as matrix reordering methods and compression techniques to be used. Furthermore, improved control of matrix diagonal term ratio output is provided.

Benefits

The new interface provides analysts more succinct control over the process than the existing method selection process that usually involves defining values for some SYSTEM cells via the NASTRAN statement. In addition, a new output data option is available for matrix diagonal term ratios in the form of a simple bar chart that provides a more comprehensive view of the ratio data.

Method and Theory

No new theory is involved. The method involves simply the specification of solver options to be used by various DMAP modules during the solution process. The specified options are checked against feature availability tables to ensure that they do not conflict with any limitations posed by the specification of feature combinations. For example, the specification of a particular ordering method may not be available for a particular solver specification. The options are available only for the DCMP, DECOMP, FRRD1, READ, SOLVE and TRD1 modules.

Inputs

The sparse solution options are controlled by keywords specified on the SPARSESOLVER Executive statement. See the *MSC Nastran Quick Reference Guide* for a complete description of this statement.

Outputs

There are no new outputs associated with this feature other than informational and diagnostic messages.

Guidelines and Limitations

The ability to specify particular sparse matrix solution options is sometimes useful in determining whether one method is more effective than another in obtaining the solution. Other features can also be useful as in obtaining diagnostic data output. For example, one might be interested in reviewing matrix diagonal term ratios. In general, high ratios indicate a loss of accuracy. The feature can be used by taking all of the program defaults for the various control variables. These defaults produce both the table and bar outputs. The table is limited to 25 ratios that exceed 1.0E+05. The bar chart uses powers of ten for segment widths. This can be done by simply adding:

```
SPARSESOLVER  DCMP ( MDTRATIO )
```

to the Executive Section of the input data file. The use of this new feature is currently limited to sparse symmetric matrix operations in the DCMP module.

Several different sparse matrix factorization methods are available. Specification of a particular method should be done only after thoughtful consideration. The following table summarizes the advantages and disadvantages of the various factorization methods.

Method	Advantages	Limitation
MSCLDL	Small memory requirements; handles indefinite matrices	None
MSCLU	Small memory requirements	None
TAUCSCHL	Generally superior performance compared to MSCLDL	High memory requirements; real positive definite matrices only
UMFLU	Generally superior performance compared to MSCLU	High memory requirements

Similarly, re-ordering methods can also be specified, but only should be done after consideration of the potential effects. The following table summarizes the advantages and disadvantages of the various re-ordering methods.

Method	Advantage	Limitation
AMF	Similar to BEND but with smaller memory requirements	Produces less optimal reordering compared to BEND
BEND	Optimal for small matrices and for large matrices from models dominated by 1- and 2-dimensional finite elements	None
MD	Very low memory requirements	Produces sub-optimal reordering
METIS	Good for large matrices dominated by 3-dimensional finite elements	Large memory requirements
MMD	Low memory requirements	Produces sub-optimal reordering

Similarly, compression methods can also be specified, but only should be done after consideration of the potential effects. The following table summarizes the advantages and disadvantages of the various compression methods.

Method	Advantage	Limitation
GRID	Utilizes USET and SIL information from the Nastran Database	Relies on USET and SIL
SUPER	Does not rely on USET and SIL tables; can produce better orderings for models dominated by 3-dimensional elements	None
GRDSUPER	Can produce better orderings for models dominated by 3-dimensional finite elements	Requires USET and SIL tables

Improved Performance with New Sparse Solvers

Introduction

As part of MSC Software HPC development, surveys of solver technology from industry and academia are conducted. This has lead to the integration of two solvers developed outside of MSC Software: TAUCS and UMFPACK.

In addition, MSC has enhanced the Lanczos eigensolver to take better advantage of available memory.

The TAUCS Sparse Solver

A new sparse Cholesky solver is available. The solver is derived from the TAUCS package of linear solvers.

User Interface

The new solver can be selected for the DCMP, DECOMP, and SOLVE modules through the SPARSESOLVER Executive statement:

```
SPARSESOLVE DCMP (FACTMETH=TAUCSCHL)
```

It may also be selected by setting system cell 166 to 8192.

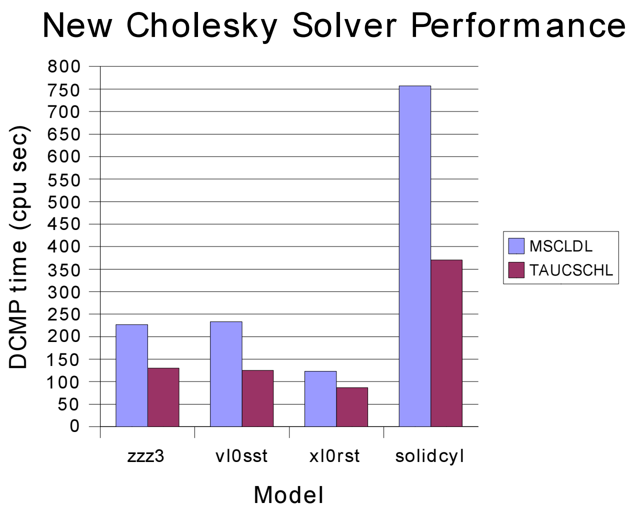
Use Cases

The new solver keeps the matrix data in memory, and may show improved performance over the default solver for models which are dominated by three dimensional elements.

The four models described below demonstrate the potential performance enhancement with the new solver.

Model Name	DOF	Elements
zzz3	537876	108675 TETRAs
vl0sst	408999	710768 TETRAs
xl0rst	739815	113217 HEXAs
Solidcyl	604800	176400 HEXAs

Each model was run on one node with 2 dual core 2.4GHz Opteron processors, 8Gb of memory and 56Gb of scratch space. Each job was run with mem=7gb, and the CPU time of the DCMP module is displayed in the following chart.



Limitations

The new solver only works for real, symmetric matrices which are positive definite. Modeling techniques which lead to indefinite matrices, such as Lagrange multipliers, are not supported by the new solver. If the new solver detects an indefinite matrix, or has insufficient memory to perform the factorization, or encounters any other error, the out-of-core LDL^T solver is used.

The new solver is not recommended for multiple superelement analysis.

The TAUCS code is used by MSC under the following license agreement. Please note that in the agreement, “this program” and “this software” refers only to the TAUCS code, available at http://www.tau.ac.il/~stoledo/taucs.

The UMFPACK Sparse Solver

UMFPACK is a set of routines for solving unsymmetric sparse linear systems using an unsymmetric multi-frontal method. The UMFPACK factorization has been implemented in Nastran as a licensed software product from the University of Florida. It operates on both real and complex matrices.

UMFPACK has been implemented for solving unsymmetric frequency response problems. In addition, it is implemented in the SOLVE and DECOMP modules. It is activated by an explicit user request via the SPARSESOLVER Executive statement. For example:

```
SPARSESOLVER  FRRD1  (FACTMETH=UMFLU)
```

This command specifies that UMFPACK is to be run in the FRRD1 module, which is used for frequency response analysis problems. UMFPACK may also be selected by setting system cell 209 to 16.

The UMFPACK solver has shown to be very useful for problems in Exterior Acoustics, which generates UMFPACK Memory Guidelines

The UMFPACK solver operates entirely in memory. Therefore, memory requirements for UMFPACK may be considerably higher than for the default Nastran sparse direct unsymmetric solver.

Additionally, it is important to note that UMFPACK operates entirely outside the control of Nastran memory management. Each time it is invoked, UMFPACK will obtain memory from the operating system, use the new memory, and then return the new memory back to the operating system. If there is insufficient memory to complete the UMFPACK factorization, Nastran terminates with a Fatal Error message, and it attempts to inform the user of memory requirements. It is not possible to allocate additional memory for UMFPACK via the *nastran* command line option "*mem=*".

It is the user's responsibility to ensure that sufficient unsubscribed memory is available from the operating system for UMFPACK to complete successfully. This may mean lowering the amount of memory requested at job submittal with the "*mem=*" option. For example, suppose there is 8GB of memory on your computer, and you want to use Nastran with UMFPACK. If the memory requirement for UMFPACK to complete is 4GB, then you should not submit your Nastran job with more than 4GB. In fact, UMFPACK will be the most memory intensive portion of your Nastran analysis, so that the actual Nastran memory could be set much lower by comparison.

This memory limitation will be removed in a future release, so that all UMFPACK memory operations will be controlled by Nastran memory management.

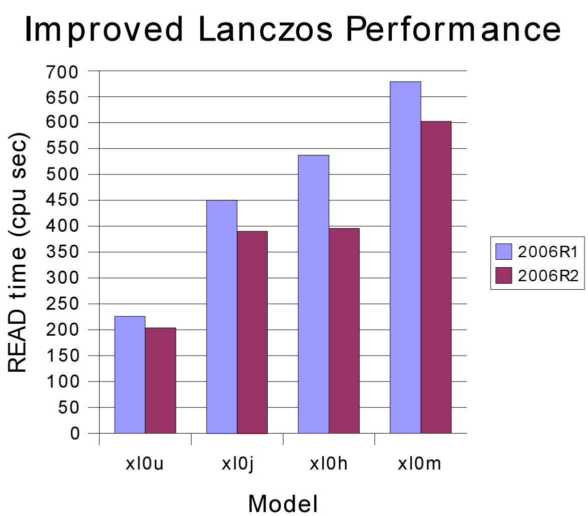
Improved Memory Usage in Lanczos (Pre-release)

The Lanczos method has been improved to better take advantage of the given memory.

This beta feature may be selected by setting system cell 146 to -1.

To illustrate the performance improvements, the following models were run on an SGI Altix with four 1.4GHz itanium2 processors, 20Gb of ram and 470Gb of disk space. Each model was run with mem=4200mb.

Model Name	DOF	Elements
xl0u	595303	55925 HEXAs, 18483 PENTAs
xl0h	335680	54509 QUAD4s, 15523 TRIA3s
xl0j	590474	94979 QUAD4s, 13406 TRIA3s
xl0m	529199	96185 QUAD4s



New MAXRATIO Information Output (Pre-release)

Introduction

A new interface is now available for analysts to better control the generation of matrix diagonal term ratio statistics produced by the sparse symmetric matrix decomposition process in the DCMP module. The matrix diagonal term ratio statistics are sometimes useful in determining the quality of the matrix decomposition process. In general, for linear static analysis, high or negative ratios indicate a loss of accuracy and could be indicative of a modeling error.

Benefits

The new interface provides analysts more control over the process than the existing method of supplying a value for the MAXRATIO DMAP parameter. In addition, a new output data option is available in the form of a simple bar chart that provides a more comprehensive view of the ratio data.

Method and Theory

No new theory is involved. The method involves simply the computation of a ratio defined to be the original matrix diagonal term divided by the decomposed matrix diagonal term. These ratios are placed in a table together with the external identifier associated with the row/column of the term. This table is then processed according to the options requested by the user.

Inputs

The matrix diagonal term ratio output options are controlled by keywords specified on the SPARSESOLVER Executive statement. See "New SPARSESOLVER Executive Statement" on page 34 for a complete description of this statement.

Outputs

The matrix diagonal term ratios can be presented in two different views. The first view is the table view in which each ratio is listed together with the external identifier of the row/column of the matrix as well as the original input matrix diagonal term. This format is virtually identical to that produced now when any ratio exceeds the value of the MAXRATIO input parameter. The second view of the ratios is statistical in nature. It is similar to a bar chart. A series of bar segments are generated. There are two options for specifying the segment widths of the bars. The default option uses powers of 10 as the widths (e.g. 10.0 to 100.0 and 100.0 to 1000.0). The second option allows the user to specify how many segments are desired. The program will compute the segment width using the maximum and minimum ratios. For each bar in the chart, the total number of terms in the range is tabulated together with a visual indication of the percentage number of terms in that particular bar.

Note that when negative matrix diagonal term ratios are detected, they will always be output if the TABLE option is specified.

These new views of the ratios do not replace any existing diagnostics generated by the DCMP module if a problem is detected. Under these conditions, output from the table view may duplicate previous output generated by DCMP module error processing.

Guidelines and Limitations

The matrix diagonal term ratio statistics are sometimes useful in determining the quality of the matrix decomposition process. In general, high ratios indicate a loss of accuracy. The feature can be used by taking all of the program defaults for the various control variables. These defaults produce both the table and bar outputs. The table is limited to 25 ratios that exceed 1.0E+05. The bar chart uses powers of ten for segment widths. This can be done by simply adding

SPARSESOLVER DCMP (MDTRATIO)

to the Executive Section of the input data file.

The use of this new feature is currently limited to sparse symmetric matrix operations in the DCMP module.

If there are scalar-type points present in the problem, the degrees of freedom associated with these points will be grouped into the results for the translational degrees of freedom output.

Demonstration Example

A simple example is presented that demonstrates the use of some of the new features available for output of the matrix diagonal term ratios. The SPARSESOLVER Executive statement is used to specify the desired features. The example problem is used for demonstration purposes only and is not representative of anything in particular. The model data consists of a simple plate structure subject to an end load.

Example Input Data

```
$
$ Example problem to demonstrate matrix diagonal term ratios
$
id test,case
sol 101
SPARSESOLVER  DCMP  (MDTRATIO) $
cend
spc=100
load=1000
disp=all
begin bulk
grdset,,,,,,,6
cquad4,101,101,1,2,52,51
cquad4,102,101,2,3,53,52
cquad4,103,101,3,4,54,53
cquad4,104,101,4,5,55,54
cquad4,105,101,5,6,56,55
```

```
cquad4,106,101,6,7,57,56
cquad4,107,101,7,8,58,57
cquad4,108,101,8,9,59,58
cquad4,109,101,9,10,60,59
cquadr,1101,101,1,2,52,51
cquadr,1102,101,2,3,53,52
cquadr,1103,101,3,4,54,53
cquadr,1104,101,4,5,55,54
cquadr,1105,101,5,6,56,55
cquadr,1106,101,6,7,57,56
cquadr,1107,101,7,8,58,57
cquadr,1108,101,8,9,59,58
cquadr,1109,101,9,10,60,59
grid, 1,, 0.0,0.0,0.0
grid, 2,, 1.0,0.0,0.0
grid, 3,, 2.0,0.0,0.0
grid, 4,, 3.0,0.0,0.0
grid, 5,, 4.0,0.0,0.0
grid, 6,, 5.0,0.0,0.0
grid, 7,, 6.0,0.0,0.0
grid, 8,, 7.0,0.0,0.0
grid, 9,, 8.0,0.0,0.0
grid,10,, 9.0,0.0,0.0
grid,51,, 0.0,1.0,0.0
grid,52,, 2.4,1.0,0.0
grid,53,, 3.5,1.0,0.0
grid,54,, 4.6,1.0,0.0
grid,55,, 5.7,1.0,0.0
grid,56,, 6.8,1.0,0.0
grid,57,, 7.9,1.0,0.0
grid,58,, 9.0,1.0,0.0
grid,59,,10.1,1.0,0.0
grid,60,,11.2,1.0,0.0
$
ctria3,201,101,101,102,151
ctria3,202,101,102,152,151
ctria3,203,101,102,103,152
ctria3,204,101,103,153,152
ctria3,205,101,103,104,153
ctria3,206,101,104,154,153
ctria3,207,101,104,105,154
ctria3,208,101,105,155,154
ctriar,1201,101,101,102,151
ctriar,1202,101,102,152,151
ctriar,1203,101,102,103,152
ctriar,1204,101,103,153,152
ctriar,1205,101,103,104,153
ctriar,1206,101,104,154,153
ctriar,1207,101,104,105,154
ctriar,1208,101,105,155,154
grid,101,, 0.0,0.0,0.0
grid,102,, 1.0,0.0,0.0
grid,103,, 2.0,0.0,0.0
grid,104,, 3.0,0.0,0.0
grid,105,, 4.0,0.0,0.0
grid,151,, 0.0,1.0,0.0
grid,152,, 3.4,1.0,0.0
grid,153,, 4.5,1.0,0.0
grid,154,, 5.6,1.0,0.0
grid,155,, 6.7,1.0,0.0
```

```
$
pshell,101,1,0.05,1
mat1,1,10.+6,,0.33
spc1,100,123,1,101
spc1,100,3,5,55,105,155
spc1,100,1,55,155
spc1,100,2,1,101
force,1000,10,,1000.0,1.0,0.0,0.0
force,1000,60,,1000.0,1.0,0.0,0.0
force,1000,105,,1000.0,1.0,0.0,0.0
force,1000,155,,1000.0,1.0,0.0,0.0
enddata
```

Example Output

The output generated by the previous example is shown following. Notice that there are two separate sections of output: one for translational degrees of freedom and one for rotational. Within each section, both a bar chart and table of matrix diagonal term ratios are output.

```
 TRANSLATIONAL DOF DIAGONAL TERM RATIO STATISTICS CHART FOLLOWS FOR THE DECOMPOSITION OF MATRIX KLL
  ------------------------------------------------|---------------------------------------------------------------------------
     DIAGONAL TERM RATIO RANGE    #TERMS    % TOT  |MAXIMUM RATIO =   6.90963E+02         MINIMUM RATIO =   1.00000E+00
  ------------------------------------------------|---------------------------------------------------------------------------
    1.0000E+00  TO  1.0000E+01        62   79.49   |***************************************************************************>
    1.0000E+01  TO  1.0000E+02        12   15.38   |***************
    1.0000E+02  TO  1.0000E+03         4    5.13   |*****

0
0        MATRIX/FACTOR DIAGONAL TERMS RATIO SUMMARY TABLE FOR TRANSLATIONAL DOF SORTED ON DIAGONAL RATIO
         GRID POINT ID       DEGREE OF FREEDOM   MATRIX/FACTOR DIAGONAL RATIO        MATRIX DIAGONAL
                                    (TOP 1 RATIOS>MAXRAT= 6.90963E+02)
              58 T3 6.90963E+02            5.65535E+04

       ROTATIONAL DOF DIAGONAL TERM RATIO STATISTICS CHART FOLLOWS FOR THE DECOMPOSITION OF MATRIX KLL
  ------------------------------------------------|---------------------------------------------------------------------------
     DIAGONAL TERM RATIO RANGE    #TERMS    % TOT  |MAXIMUM RATIO =   3.35974E+02         MINIMUM RATIO =   1.00000E+00
  ------------------------------------------------|---------------------------------------------------------------------------
    1.0000E+00  TO  1.0000E+01        38   63.33   |***************************************************************
    1.0000E+01  TO  1.0000E+02        18   30.00   |******************************
    1.0000E+02  TO  1.0000E+03         4    6.67   |*******

0
0        MATRIX/FACTOR DIAGONAL TERMS RATIO SUMMARY TABLE FOR    ROTATIONAL DOF SORTED ON DIAGONAL RATIO
         GRID POINT ID       DEGREE OF FREEDOM   MATRIX/FACTOR DIAGONAL RATIO        MATRIX DIAGONAL
                                    (TOP 1 RATIOS>MAXRAT= 3.35974E+02)
              58              R2        3.35974E+02                 2.14135E+04
```

Performance Improvements and Expanded Capabilities for ACMS

Introduction

Automated Component Modal Synthesis (ACMS) is a powerful tool for a variety of large modal based analyses. ACMS functions in two domains, the Matrix Domain and the Geometric Domain. Since its introduction in 2005, Matrix Domain ACMS (MDACMS) has become the default ACMS method. Developments described in this section apply to MDACMS.

MDACMS has been extended to more thoroughly interact with the Nastran External Superelement capability. See "MDACMS for Upstream Superelements" below.

An automatic decision logic has been implemented to select which path to take for frequency response calculations. See "Automatic FASTFR Decision Logic" below.

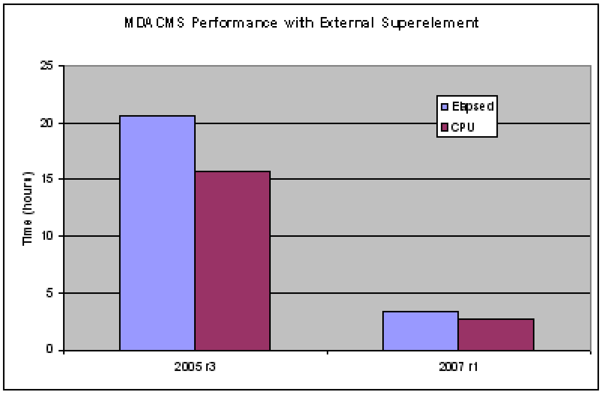
For other enhancements, see "Miscellaneous Performance Improvements" below.

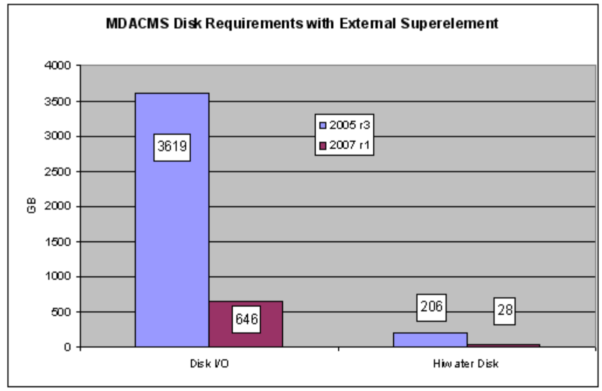
MDACMS for Upstream Superelements

MDACMS has been extended to more thoroughly interact with the Nastran External Superelement capability. Specifically, MDACMS reduction calculations have been expanded to include all quantities being reduced to the residual, namely stiffness, mass, and damping. (Previously, only the component modes were computed by MDACMS.) By reducing these quantities while simultaneously computing component modes, a significant reduction in compute time, disk I/O, and scratch disk space is realized.

One typical case study is presented here. Note that in this example, the resource requirements for MSC Nastran R1 exceeded availability and the job did not complete, even though it had run five times longer than the current release at the time of its failure.

No. of Grid Points	No. of global DOF	No. of O-set DOF	No. of A-set DOF	Analysis Description
375,502	2.25 million	1.9 million	3526	Normal modes, frequency range 0-550Hz (1627 modes)





The above jobs were run on an IBM Power4 computer running the AIX operating system.

Automatic FASTFR Decision Logic

In MSC.Nastran 2004, the FASTFR method was introduced for modal frequency response analysis. It can be selected via the Bulk Data entry *PARAM,FASTFR,YES* and shows significant performance improvement for certain models in the mid-frequency range. However, the user has to make the decision as to whether to use the new method or whether to run with the standard FRRD1 method with or without the iterative solver before starting to run the job.

With this release, automatic decision logic has been implemented which eliminates the need for the user to make that decision. The program will decide automatically which solution method will be most efficient for the frequency response portion in a SOL 111 analysis. Based on the size of the modal space and some other heuristic criteria, either the FASTFR solution method will be run, or the FRRD1 module with or without the iterative solver will be used.

Currently, the FASTFR method is selected or deselected in the Bulk Data Section using *param,fastfr,yes* or *param,fastfr,no*. To activate the automatic decision logic, specify

```
PARAM, FASTFR, AUTO
```

in the Bulk Data Section. This will cause the program to run as if the FASTFR method was selected until it reaches the point where the decision will be made. If it is decided that it would be faster to *not* run with the FASTFR method, the following message will be printed in the F06 file, and the program will continue with the standard frequency response method.

Also, if the FASTFR method is deselected for a different reason, a similar message will be printed. If the

```
^^^ SYSTEM INFORMATION MESSAGE 9157 (GMA)
^^^ FASTFR OPTION REQUESTED, BUT THE MODEL DID NOT  MEET THE FOLLOWING CRITERIA:
^^^  THE FASTFR OPTION MAY BE TOO EXPENSIVE.
^^^ USER INFORMATION:   STANDARD FREQUENCY RESPONSE METHOD WILL BE USED
```

FASTFR method will actually be used, then no message will be printed.

This new option for *PARAM,FASTFR* is helpful for all modal frequency response analysis jobs, especially in the mid to upper frequency range.

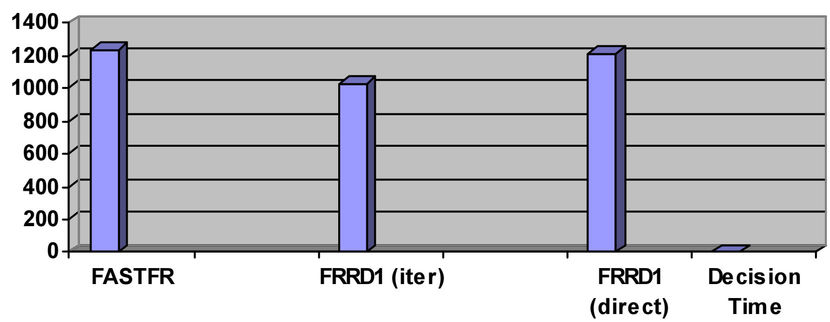
Examples

The following charts show the benefit of the automatic decision logic for three examples. Please note that the times given are the elapsed minutes for the complete job.

Example 1

For this automotive example, the automatic decision logic decided that the FASTFR method might be too expensive. So it switched to the FRRD1 module with the iterative solver.

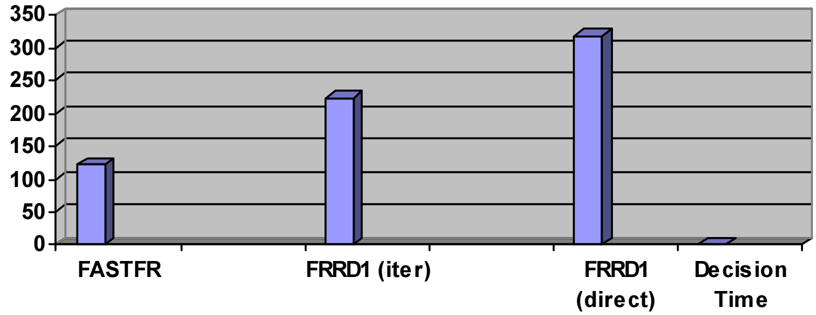
Job Statistics: 1,208,135 grid points;
matrix size in FRRD1 (i.e. H-size) is 9,974;
modes below 1,000 Hz;
2 frequencies.



Example 2

For this automotive example, the automatic decision logic decided to go with the FASTFR method, and rightly so. Both, the FRRD1 module with and without the iterative solver proved to be much slower.

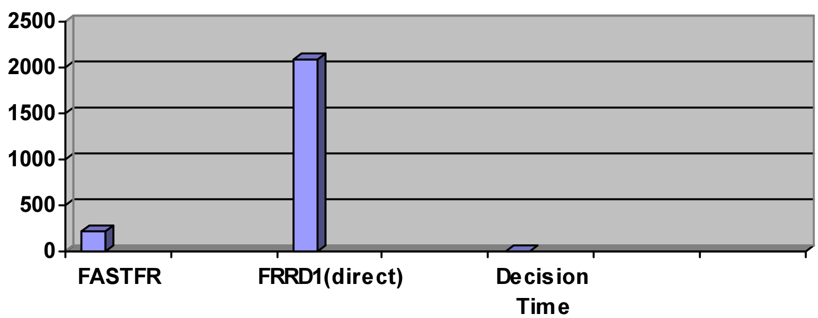
Job Statistics: 233,094 grid points;
matrix size in FRRD1 is 4,350;
modes below 600 Hz;
260 frequencies.



Example 3

For this automotive example, the automatic decision logic decided to go with the FASTFR method which proves to be significantly faster than the conventional method (i.e. FRRD1 with the direct method). The iterative method inside FRRD1 is not practical for this job because there are 147 loads which significantly slow down the iterative solver.

Job Statistics:	321,597 grid points;
	matrix size in FRRD1 is 5,424;
	modes below 600 Hz;
	501 frequencies.



Limitations for the FASTFR method

1. Fluid Damping can only be specified via the param,gfl entry and/or the sdamping(fluid) command.
   a. The CAABSF element (acoustic absorbers) is not supported;
   b. The FASTFR method will be turned off automatically for fluid K4 and for fluid viscous damping.

2. All matrices must be symmetric
   a. Unsymmetric formulation for acoustic coupling is not supported;
   b. EPOINT Bulk Data entries are not supported.
3. The FASTFR method work only for SOL 111 and for SOL 200 with ANAL=MFREQ.
4. SESDAMP and FASTFR are not allowed in the same run.

Miscellaneous Performance Improvements

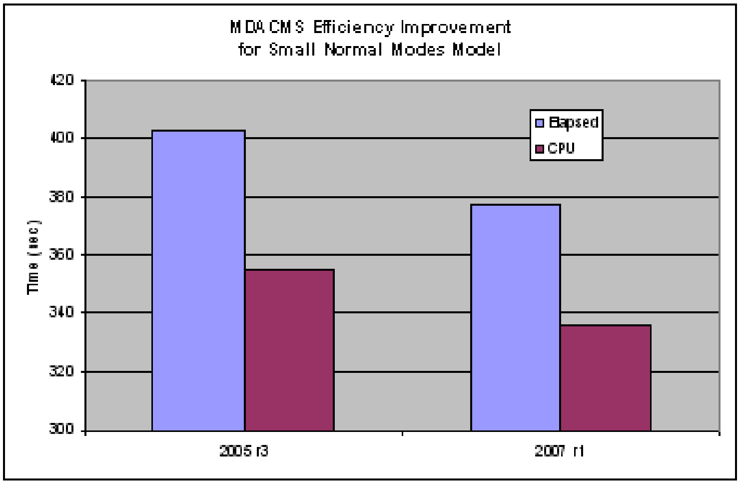
MDACMS has been enhanced to reduce the amount of disk I/O required for a successful analysis. This happens automatically and there is no user action required to realize the reduction in I/O. Typically, reduced I/O requirements results in reduced elapsed time and greater CPU utilization, which increases the effectiveness of Shared Memory Parallel (SMP) computations.

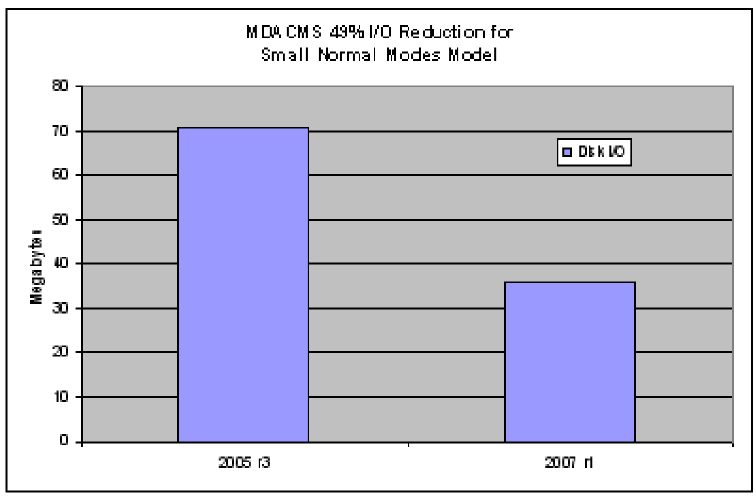
To demonstrate some of these performance improvements, three example jobs were run and the performance results were compared to the previous version. The results are displayed below.

Model Description:

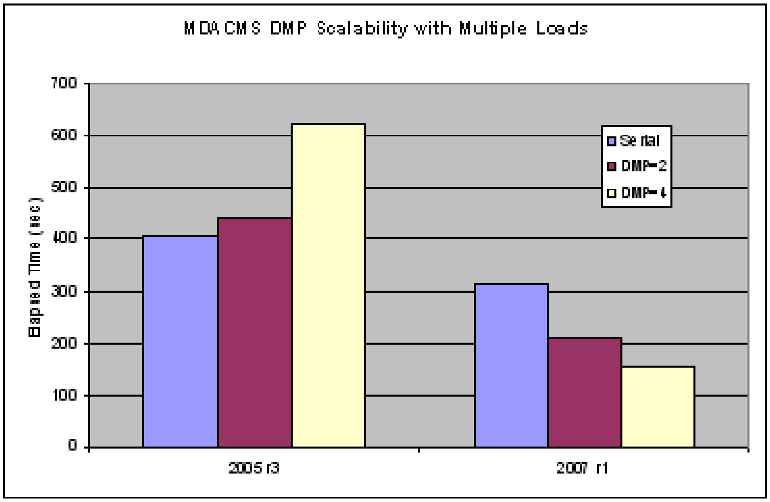
Model No.	No. of grid Points	No. of global DOF	Analysis Description
1	44,314	265884	Auto body normal modes run w/ACMS. Frequency range 0-400Hz (360 modes)
2	44,314	265884	Model (1) run in SOL 111 w/ACMS; 75 load cases, 250 forcing frequencies
3	1,243,651	7.4 million	Mid frequency acoustic analysis of automotive trimmed body. 9389 structure modes below 750Hz; 1486 fluid modes below 1200Hz; 4 load cases, 290 forcing frequencies

Example 1 shows time and I/O resource requirements for model one run on an IBM Power4 system under AIX.

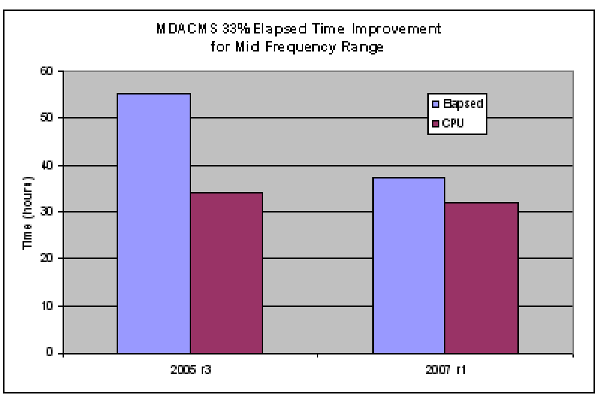


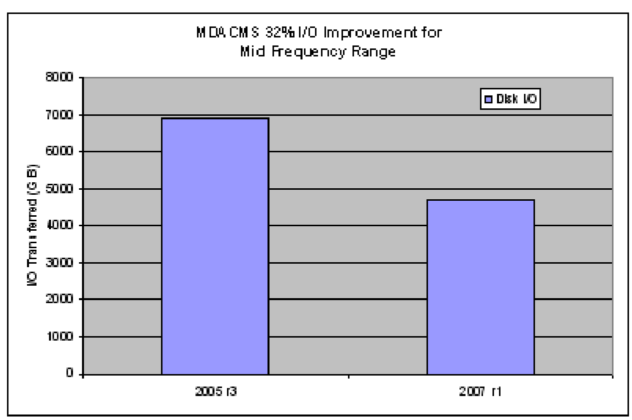


Example 2 shows DMP scalability improvement for model two. These jobs were run on a Altix machine with IA64 processors under Linux.



Example 3 shows I/O and elapsed time improvement for model three. These jobs were run on an IBM Power4 system under AIX.





Relaxed Restrictions for CASI Solver Usage

The use of the element-based CASI iterative solver is limited by several restrictions noted in the remarks for the ITER Bulk Data entry. Two of these restrictions have been somewhat relaxed so that the solver is usable over a wider range of problems.

The first is a reduction in the number of element types that the solver does not recognize. The solver recognizes only a sub-set of all of the element types available, but the list has been expanded to include the FAST, SEAM and WELD element types as supported element types.

The second is a relaxation of the restriction associated with the processing of direct input matrices via the K2GG Case Control command. K2GG matrix input is now allowed by the CASI solver interface. However, the matrix size is limited to 100 grid/scalar points, since it must be treated as an unknown finite element type. This should be sufficient for most cases where the input K2GG matrix represents some form of attachment to, or base for, the structural model being analyzed.

System Dependent Performance Improvements

Introduction

MSC has implemented new versions of the Basic Linear Algebra Subroutine library (BLAS) for Nastran, on platforms supported by the "x86-64" architecture.

Version 9 of Intel's Math Kernel Library (MKL) provides optimal performance for analysis solutions that make good use of the BLAS. This includes the ACMS solution for automotive NVH and acoustic analysis, and the Exterior Acoustics capability in Nastran. In addition, MKL provides automatic multithreading support. Shared memory parallel speedup is available via the *smp=n* command supplied to the *nastran* command processor.

The ATLAS project (Automatically Tuned Linear Algebra Software) provides a portable, efficient version of the BLAS. An ATLAS version of the BLAS library has been implemented for use on AMD Opteron processors. ATLAS is public domain software available on the internet at

`http://math-atlas.sourceforge.net/.`

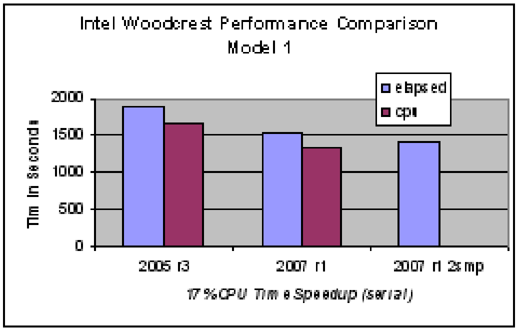
For demonstration purposes, two analysis model were selected and run with the current and prior release of MSC Nastran. Performance results are shown on the following page.

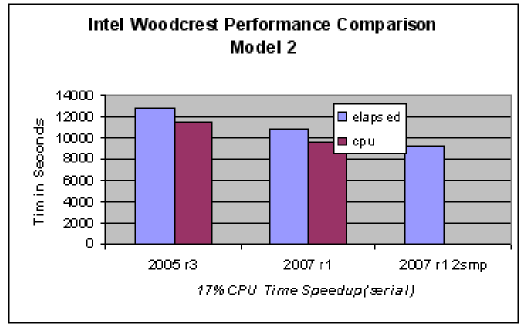
Model Description

Model No.	No of Grid Points	No. of Global DOF	Analysis Description
1	268,486	1.6 million	Auto body normal modes run w/ACMS. Frequency range 0-200Hz (1043 modes)
2	603,266	2.8 million	Low frequency acoustic analysis of automotive trimmed body

Performance Improvements for Intel x86_64

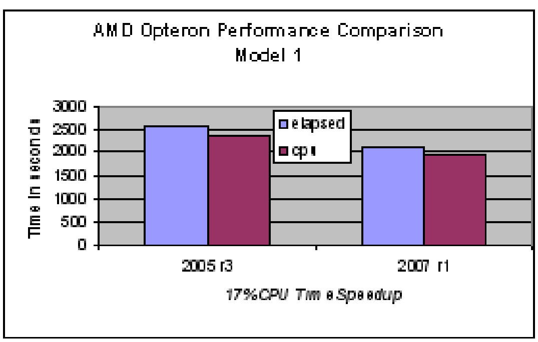
Processor	Clock Speed	OS	Real Memory	Scratch Filesystem
Woodcrest	2666MHz	Linux RHWS4	16GB	ext2 – not striped

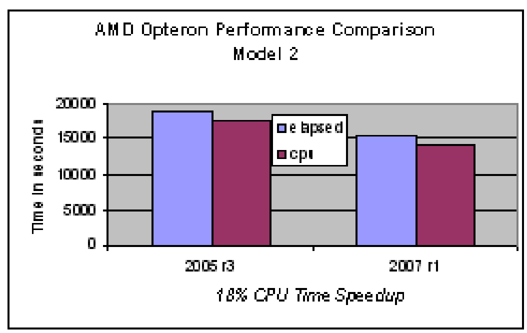




Performance Improvements for AMD due to Atlas

Processor	Clock Speed	OS	Real Memory	Scratch Filesystem
Opteron	2400MHz	Linux SuSE 10	8GB	xfs – not striped





Improved Selection of Reordering Methods

Nastran contains five methods for reordering (permuting) sparse matrices in preparation for a sparse symmetric factorization. The five methods are minimum degree (MD), multiple minimum degree (MMD), and three nested dissection/minimum degree hybrid methods, called BEND, AMF, and METIS. A judicious choice of a reordering method can lead to a dramatic improvement in the performance of the sparse factorization.

The hybrid methods (BEND, AMF, and METIS) are generally superior to MD and MMD. The previous default behavior was to use BEND unless the USET and SIL tables were not available, in which case MD was used. If the BEND algorithm failed, MD was also used in this case. The new default behavior is to use BEND except for large (> 50,000 DOF) models, which are dominated by three-dimensional elements; for those models METIS is the default. If the default method fails, the program will attempt to use another hybrid method (for example, if BEND fails, the program tries METIS). The minimum degree algorithms are selected only as a last resort.

To illustrate the performance improvements, the following models were run on an SGI Altix with four 1.4GHz itanium2 processors, 20Gb of ram and 470Gb of disk space.

Model Name	Solution	DOF	Elements
xx0e	Normal Modes	3308298	656570 TETRAs
xx0d	Normal Modes	1920855	402441 TETRAs
xx0f	Statics	1237999	246066 TETRAs
lg0q	Direct Freq. Response	93375	28140 HEXAs

The chart below shows the factorization time for each job. The improvement for the xx0e, xx0d, and xx0f jobs stems from the automatic selection of METIS rather than BEND reordering. The improvement in lg0q is due to the selection of METIS rather than MD.


Reordering Selection Performance Improvement
Factorization time (cpu sec)
2750
2500
2250
2000
1750
1500
1250
1000
750
500
250
0
xx0e
xx0f
xx0d
lg0q
Model
2006R1
2006R2

5 Elements & Connectors

- Enhancements to Connector Elements
- Nonhomogeneous Multipoint Constraint

Enhancements to Connector Elements

Introduction

In MSC Nastran 2007, a new seam weld element is introduced to allow for the definition of a seam line. This new CSEAM element replaces the existing CWSEAM element and is modeled by the new CSEAM and PSEAM Bulk Data entries. In addition, the SWLDPRM Bulk Data entry is enhanced to support element type specific control parameters and different diagnostic output format.

For spot weld elements CWELD and CFAST, the displacements of the projected grids GA and GB are now computed and output in this release. These data will facilitate users to view the relationship between the spot weld and the connecting shells.

CSEAM Elements

The new CSEAM element provides the following key features to extend the analysis capabilities, provide more flexibility in modeling, and improve the robustness of the analysis results.

- A seam line is considered continuous between two CSEAM elements that have a common face based on either common GS/GE points or XYZ coordinates. Note that the SMLN label on a CSEAM element does not determine the definition of a seam line. It is only intended for ease of seam line visualization.
- This element can connect up to 64 shell grids, which allows the connection of higher order shell elements.
- Besides selecting the connected surface patches by property IDs, the user may define the connection by specifying shell element IDs directly.
- Each of the eight auxiliary points must have a projection onto the connected shell element. This requirement prevents the generation of ill conditioned stiffness matrix in down stream processing.
- The user can model tailored blank tapering by specifying different property IDs at the start and end points.
- This element type supports the MAT9 anisotropic material properties.

Inputs

The seam connection is modeled by the new CSEAM and PSEAM Bulk Data entries and the modified SWLDPRM Bulk Date entry. The details of these entries are described in the *MSC Nastran Quick Reference Guide*.

Outputs

The connecting record based on seam line label is written as a SEAMLN record in GEOM2 data block, which is included in OP2 file for post-processing. The contents of the SEAMLN record for each seam line label are listed:

Word	Type	Description
1-2	C	Seam line ID
3	I	Total number of elements for this seam line
4	I	CSEAM element ID
5	I	GS grid ID for the start point
6	I	GE grid ID for the end point
:	I	Repeat words 4-6 for each CSEAM element
:	I	End the seam line data with -1

Limitations

- Each CSEAM element can connect a maximum of three shell elements on patch A and three shell elements on patch B.
- Only line type of seam is supported.
- Super-element modeling is not supported.
- FORCE, STRESS and STRAIN output requests are not supported.

Example – A Symmetric Hat Profile (cseam_hut.dat)

This example demonstrates the application of CSEAM elements to analyze an unconstrained symmetric hat profile model, see Figure 5-1. Each edge of the hat is connected by 27 CSEAM elements. The grids with identification numbers 6000 to 6028 and 8000 to 8028 are used as the piercing points to define the seams, see Figure 5-2.

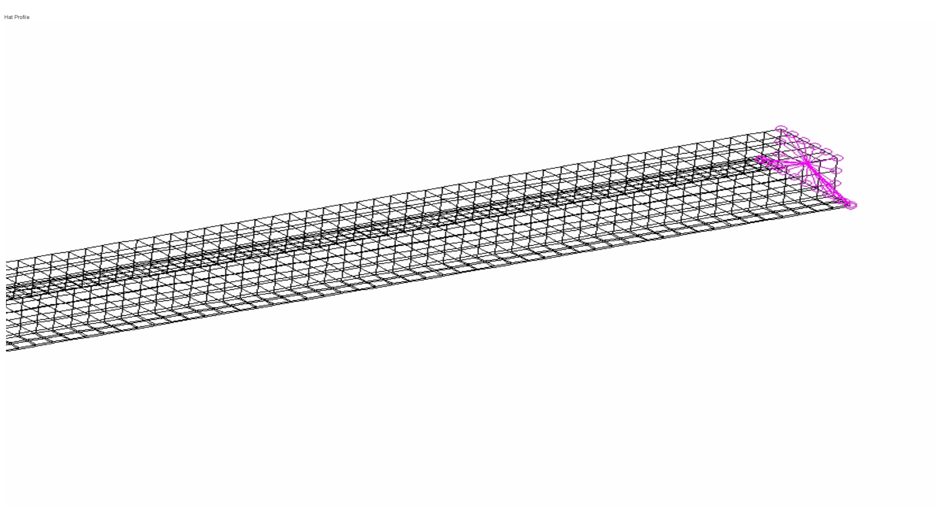


Figure 5-1 Hat Profile

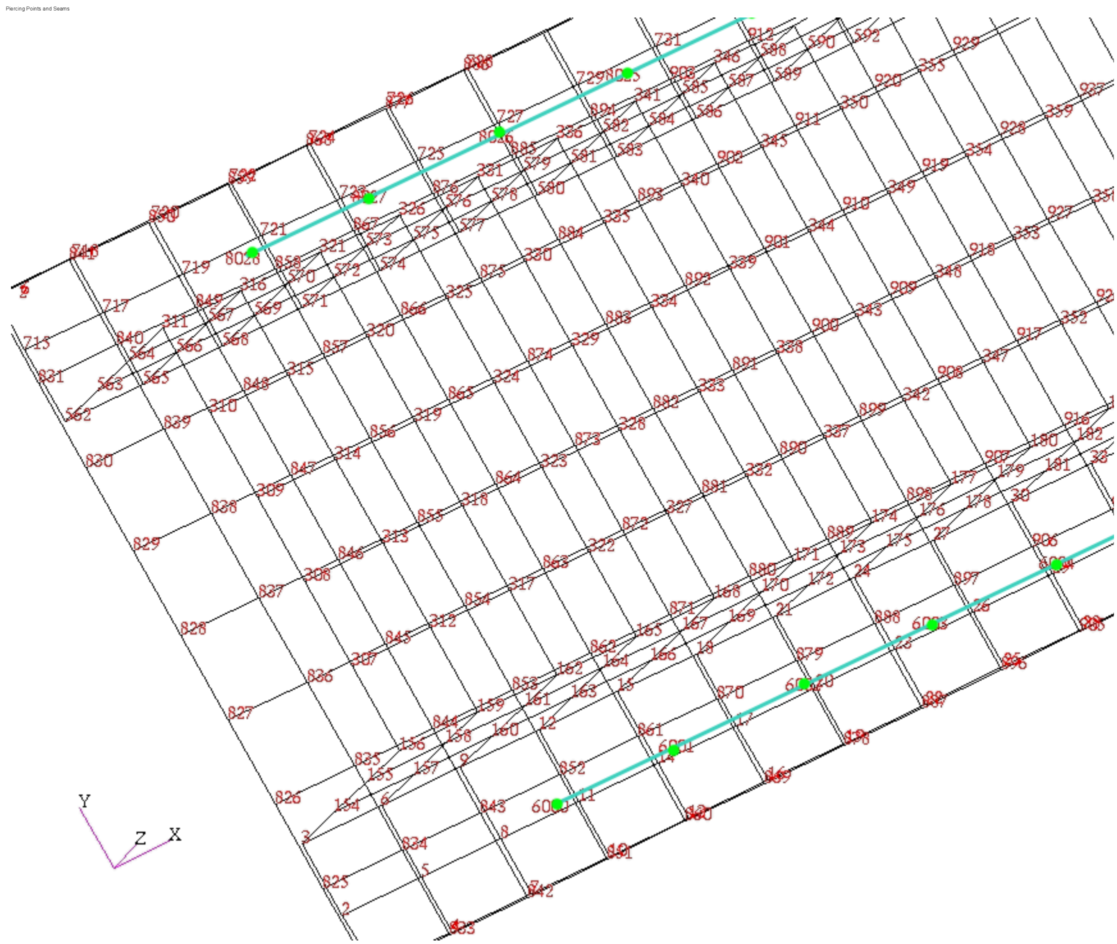


Figure 5-2 Piecing Points and Seams

The input for the seam welds is listed:

```
CSEAM      7000       500   SMLN_A   PSHELL      100      200
           6000      6001
CSEAM      7001       500   SMLN_A   PSHELL      100      200
           6001      6002
:
CSEAM      7026       500   SMLN_A   PSHELL      100      200
           6026      6027
CSEAM      7027       500   SMLN_A   PSHELL      100      200
           6027      6028
$
CSEAM      9000       500   SMLN_B   PSHELL      100      200
           8000      8001
CSEAM      9001       500   SMLN_B   PSHELL      100      200
           8001      8002
:
CSEAM      9026       500   SMLN_B   PSHELL      100      200
```

```
                8026      8027
CSEAM           9027       500    SMLN_B   PSHELL      100       200
                8027      8028
$
PSEAM            500         1     LINE      1.0
```

The normal mode analysis results are shown:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	-5.927181E-05	7.698819E-03	1.225305E-03	1.000000E+00	-5.927181E-05
2	2	-2.941204E-05	5.423287E-03	8.631429E-04	1.000000E+00	-2.941204E-05
3	3	2.051645E-05	4.529509E-03	7.208938E-04	1.000000E+00	2.051645E-05
4	4	2.372748E-05	4.871086E-03	7.752574E-04	1.000000E+00	2.372748E-05
5	5	5.330599E-05	7.301095E-03	1.162005E-03	1.000000E+00	5.330599E-05
6	6	1.118197E-04	1.057448E-02	1.682981E-03	1.000000E+00	1.118197E-04
7	7	1.154947E+07	3.398451E+03	5.408803E+02	1.000000E+00	1.154947E+07
8	8	1.585229E+07	3.981493E+03	6.336742E+02	1.000000E+00	1.585229E+07
9	9	2.653947E+07	5.151647E+03	8.199102E+02	1.000000E+00	2.653947E+07
10	10	2.959784E+07	5.440389E+03	8.658648E+02	1.000000E+00	2.959784E+07
11	11	3.002028E+07	5.479077E+03	8.720222E+02	1.000000E+00	3.002028E+07
12	12	3.041736E+07	5.515193E+03	8.777703E+02	1.000000E+00	3.041736E+07
13	13	3.328896E+07	5.769658E+03	9.182696E+02	1.000000E+00	3.328896E+07
14	14	3.757468E+07	6.129819E+03	9.755909E+02	1.000000E+00	3.757468E+07
15	15	4.259932E+07	6.526814E+03	1.038775E+03	1.000000E+00	4.259932E+07
16	16	4.738684E+07	6.883810E+03	1.095592E+03	1.000000E+00	4.738684E+07

SWLDPRM Enhancements

Element Specific Control Parameters

The SWLDPRM Bulk Data entry is enhanced to support element type specific control parameters. Two new parameters, CNRAGLI and CNRAGLO, are introduced to define the angle limits for checking the geometry of seam elements. These parameters replace the GSPROJ parameter to define the allowable angles for corner check. The GSPROJ parameter specified in the previous seam weld models must be replaced by CNRAGLI and CNRAGLO to obtain identical results.

For example, if the SWLDPRM Bulk Data entry is defined as

```
SWLDPRM,PROJTOL,0.0,GSMOVE,0,GSPROJ,-1.,NREDIA,4,
,GMCHK,1,CHKRUN,1
```

then this entry must be changed into

```
SWLDPRM,PROJTOL,0.0,GSMOVE,0,GSPROJ,-1.,NREDIA,4,
,GMCHK,1,CHKRUN,1,cnraglo,-1.
```

to get the same results as the results running from previous version.

Real Format of Diagnostic Output

For the PRTSW parameter of SWLDPRM Bulk Data entry, two options are added to support real format of diagnostic output so that more significant digits will show (quality 1-25861871). This parameter now has five options.

PRTSW	Output
PRTSW = 0	No Diagnostic Output
PRTSW = 1	Print diagnostic output in exponential format to .f06 file
PRTSW = 2	Punch diagnostic output in exponential format to .pch file
PRTSW = 11	Print diagnostic output in real format to .f06 file
PRTSW = 12	Punch diagnostic output in real format to .pch file

Displacement Output of GA and GB for CWELD and CFAST Elements

The displacements of the projected grids GA and GB for CWELD elements with GRIDID, ELEMID, ELPAT or PARTPAT format and CFAST elements are computed to display the relationship between these elements and their connecting shell elements. As a result, the displacement output of GA and GB for CWELD element with GRIDID or ELEMID format and MSET=OFF are no longer dummy zero values. These displacements are calculated from the constraint equations described in the *Basic Theory and Methods* section.

If GA or GB is not specified for CFAST elements or CWELD elements with ELPAT or PARTPAT option, the program will create a grid internally, with the grid ID number starting from OSWPPT+1 (OSWPPT is a parameter specified by PARAM Bulk Data entry). The user may request a positive PRTSW parameter (1, 2, 11, or 12) in SWLDPRM Bulk Data entry to view the grid ID of GA or GB in the diagnostic output.

Input

The displacements of GA and GB are requested using the CONNECTOR keyword of the DISPLACEMENT Case Control command. See *MSC Nastran Quick Reference Guide* for a detail description of this command.

Outputs

The output is integrated with the displacements of the general grid points.

Basic Theory and Methods

For CWELD elements with GRIDID or ELEMID format, the displacements of the projected grids GA and GB in basic coordinate are computed by the following equations:

$$d_A = \sum_I A_I u_I$$

$$d_B = \sum_J B_J u_J$$

Where d_A and d_B are displacements at GA and GB. A_I and B_J are constraint matrices. u_I and u_J are displacements of the connected shell grids.

For CWELD elements with ELPAT or PARTPAT format, the displacements of the auxiliary points in basic coordinate are calculated first.

$$u_I = \sum_K G_{IK} u_K$$

$$u_J = \sum_L G_{JL} u_L$$

Where u_I and u_J are displacements of the auxiliary points. G_{IK} and G_{JL} are RBE3 type constraint matrices. u_K and u_L are displacements of the connected shell grids.

Then the displacements of the projected grids GA and GB in basic coordinate are computed by the same constraint equations used for GRIDID and ELEMID options.

$$d_A = \sum_I A_I u_I$$

$$d_B = \sum_J B_J u_J$$

Where d_A and d_B are displacements at GA and GB. A_I and B_J are constraint matrices. u_I and u_J are displacements of the connected auxiliary points.

Current Limitation

The displacement output of GA and GB are only available in Solution Sequences 101 and 103.

Example

This example demonstrates the various displacement output requests and their results for a small model with two CWELD elements.

The input file follows:

```
nastran mesh
SOL 101
TIME 60
CEND
TITLE= two elements, identical location for GA, GB, GS
```

```
OLOAD= ALL
FORCE = ALL
SUBCASE 1
   SUBTITLE= shear the weld
   SPC= 1
   LOAD= 1
   DISP= ALL
SUBCASE 2
   SUBTITLE= in plane twist
   set 21 = 1002,1003,2011,thru,2014
   spc= 1
   LOAD= 2
   DISP(CONN=ALL)=21
SUBCASE 3
   SUBTITLE= upper bending
   set 32 = 4
   set 33 = 1012,1013,2001,thru,2004
   spc= 1
   LOAD= 3
   DISP(CONN=32)=33
BEGIN BULK
$
$ Grids of lower shell
grid, 1001, , 0., 0., 0.
grid, 1002, , 20., 0., 0.
:
grid, 1013, , 20., 10., 5.
grid, 1014, , 0., 10., 5.
$ Grids of upper shell
grid, 2001, , 0., 0., 0.0
grid, 2002, , 20., 0., 0.0
:
grid, 2013, , 20., 10., 6.0
grid, 2014, , 0., 10., 6.0
$ spot weld grid
grid, 3001, , 10.0, 5.0, 0.0
grid, 3011, , 10.0, 5.0, 10.0
$ quad4s
cquad4, 4001, 10, 1001, 1002, 1003, 1004
cquad4, 5001, 10, 2001, 2002, 2003, 2004
cquad4, 4011, 10, 1011, 1012, 1013, 1014
cquad4, 5011, 10, 2011, 2012, 2013, 2014
:
$ property and material
pshell, 10, 10, 1.0, 10
mat1, 10, 2.e+5 , , 0.3, 0.785e-8
$
$ spot welds
$
cweld, 4, 4, 3001, elemid, , , , , +cw4
+cw4, 4001, 5001
cweld, 5, 4, 3011, elemid, , , , , +cw5
+cw5, 4011, 5011
pweld, 4, 10, 5.0
$
enddata
```

The displacement results are shown as follows:

0 SUBCASE 1

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1001	G	0.0	0.0	0.0	0.0	0.0	0.0
1002	G	4.887498E-03	5.624979E-04	0.0	0.0	0.0	1.406245E-05
1003	G	4.887498E-03	-5.624979E-04	0.0	0.0	0.0	-1.406245E-05
1004	G	0.0	0.0	0.0	0.0	0.0	0.0
1011	G	0.0	0.0	0.0	0.0	0.0	0.0
1012	G	4.887498E-03	5.624979E-04	-7.685000E-01	-6.750000E-03	5.864999E-02	1.406245E-05
1013	G	4.887498E-03	-5.624979E-04	-7.685000E-01	6.750000E-03	5.864999E-02	-1.406245E-05
1014	G	0.0	0.0	0.0	0.0	0.0	0.0
2001	G	6.058339E-04	3.750000E-04	0.0	0.0	0.0	7.733938E-19
2002	G	5.605834E-03	3.750000E-04	0.0	0.0	0.0	1.084202E-1
2003	G	5.605834E-03	-3.750000E-04	0.0	0.0	0.0	1.191827E-18
2004	G	6.058339E-04	-3.750000E-04	0.0	0.0	0.0	7.623363E-19
2011	G	3.907157E-02	3.750000E-04	8.148733E-04	-2.562901E-16	3.850649E-02	2.836059E-17
2012	G	4.407157E-02	3.750000E-04	-7.693148E-01	-5.232793E-15	3.850649E-02	2.851452E-17
2013	G	4.407157E-02	-3.750000E-04	-7.693148E-01	-5.356609E-15	3.850649E-02	2.956961E-17
2014	G	3.907157E-02	-3.750000E-04	8.148733E-04	-2.636780E-16	3.850649E-02	2.803595E-17
3001	G	0.0	0.0	0.0	0.0	0.0	0.0
3011	G	0.0	0.0	0.0	0.0	0.0	0.0
101000001	G	2.443749E-03	8.917563E-18	0.0	0.0	0.0	8.402567E-19
101000002	G	3.105834E-03	8.944668E-18	0.0	0.0	0.0	9.774760E-19
101000003	G	2.443749E-03	2.517788E-16	-3.842500E-01	-2.775558E-15	3.842500E-02	2.303930E-17
101000004	G	4.157158E-02	3.029749E-15	-3.842500E-01	-2.789435E-15	3.850649E-02	2.786400E-17

0 SUBCASE 2

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1002	G	1.499616E-02	3.324066E-02	0.0	0.0	0.0	2.330633E-03
1003	G	-1.499616E-02	3.324066E-02	0.0	0.0	0.0	2.330633E-03
2011	G	6.107051E-03	-8.968769E-03	-1.542189E-16	3.303767E-17	1.479144E-16	1.890160E-03
2012	G	2.110705E-02	4.220943E-02	-3.132043E-15	5.117434E-17	1.498367E-16	3.390160E-03
2013	G	-2.110705E-02	4.220943E-02	-2.615099E-15	5.202362E-17	1.383761E-16	3.390160E-03
2014	G	-6.107051E-03	-8.968769E-03	1.645747E-16	3.118436E-17	1.398878E-16	1.890160E-03
101000001	G	6.505213E-18	1.662033E-02	0.0	0.0	0.0	1.580825E-03
101000002	G	6.722053E-18	1.662033E-02	0.0	0.0	0.0	2.640160E-03
101000003	G	3.035766E-18	1.662033E-02	-1.434168E-15	4.168935E-17	1.434168E-16	1.580825E-03
101000004	G	1.461505E-16	1.662033E-02	-1.434197E-15	4.178690E-17	1.439374E-16	2.640160E-03

0 SUBCASE 3

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1012	G	3.460202E-16	9.473882E-16	-1.537000E+00	-1.350000E-02	1.173000E-01	6.294140E-17
1013	G	-4.478185E-16	9.592263E-16	-1.537000E+00	1.350000E-02	1.173000E-01	6.130147E-17
2001	G	0.0	0.0	8.021409E-04	-1.965454E-14	7.693148E-02	0.0
2002	G	0.0	0.0	-1.537827E+00	6.397660E-15	7.693148E-02	0.0
2003	G	0.0	0.0	-1.537827E+00	6.758049E-15	7.693148E-02	0.0
2004	G	0.0	0.0	8.021409E-04	-2.021495E-14	7.693148E-02	0.0
101000001	G	0.0	0.0	-7.685000E-01	-6.591949E-15	7.685000E-02	0.0
101000002	G	0.0	0.0	-7.685127E-01	-6.664110E-15	7.693148E-02	0.0

Nonhomogeneous Multipoint Constraint

Introduction

In the past, if a nonhomogeneous multipoint constraint was desired then the method of "SLACK" variable was required. That is, the nonhomogeneous right hand side of the MPC equation was written using a scalar or grid point and an SPC or SPCD specifying the right hand side value. In this release a MPCY Bulk Data entry is introduced that allows the user to enter in a right hand side value directly into the MPC equation. The entry defines an equation of the form

$$A_m u_m + \sum_i A_i u_i = Y_m$$

Associated with the MPCY entry is another new Bulk Data entry MPCD used to define a load selectable value for Y_m of nonhomogeneous multipoint constraint.

Benefits

The user can define a nonhomogeneous multipoint constraint directly.

Input

1. The MPCY Bulk Data entry is used to define a nonhomogeneous multipoint constraint directly. This entry can also be used to define a standard homogenous multipoint constraint as well.
2. The MPCD Bulk Data entry is used to define a load selectable value for Y_m of nonhomogeneous multipoint constraint.

Output

Standard MSC Nastran MPCF output is available.

Guidelines and Limitations

Currently this method is not supported in the Dynamic solution sequences because the dynamic load generation modules have not yet been taught to automatically expand Y_m into the required number of load columns.

Theory

The basic relations are as follows:

The unconstrained stiffness matrix equation in MSC Nastran is

$$[K_{gg}]\{U_g\} = \{P_g\} \tag{5-1}$$

If we apply a nonhomogeneous MPC constraint

$$[R_{\mu g}]\{U_g\} = \{Y_\mu\} \tag{5-2}$$

Then Eq. (5-1) becomes

$$\begin{bmatrix} \bar{K}_{NN} & K_{NM} \\ K_{MN} & K_{MM} \end{bmatrix} \begin{Bmatrix} U_N \\ U_M \end{Bmatrix} = \begin{Bmatrix} \bar{P}_N + Q_N \\ P_M + Q_M \end{Bmatrix} \tag{5-3}$$

where Q_N and Q_M are the forces of constraint necessary to impose Eq. (5-2).

Partition Eq. (5-2) as

$$\begin{bmatrix} R_{\mu N} & R_{\mu M} \end{bmatrix} \begin{Bmatrix} U_N \\ U_M \end{Bmatrix} = \{Y_\mu\} \tag{5-4}$$

solve for U_M to get

$$\{U_M\} = \{\theta_M + G_{MN}U_N\} \tag{5-5}$$

where $\theta_M = R^{-1}_{\mu M}Y_\mu$ and $G_{MN} = -R^{-1}_{\mu M}R_{\mu N}$

Substitute Eq. (5-5) into Eq. (5-3) to get

$$[K_{MN} + K_{MM}G_{MN}]\{U_N\} = \{P_M + Q_M - K_{MM}\theta_M\} \tag{5-6}$$

or

$$\{Q_M\} = \underbrace{[K_{MN} + K_{MM}G_{MN}]\{U_N\} - \{P_M\}}_{\text{current}} + \underbrace{\{K_{MM}\theta_M\}}_{\text{new}}$$

(5-7)

Since the θ of Eq. (5-5) are prescribed, any virtual variation of Eq. (5-5) the results by definition of $\delta\theta_M = 0$. Hence the standard conjugate force transformation holds. Or

$$\{Q_N\} = \left\{-G_{MN}^T Q_M\right\} \tag{5-8}$$

where the minus sign comes from the fact we are imposing constraints and that Q_N forces are reactive to the Q_M constraint forces.

Then the upper Eq. (5-3) with Eq. (5-8) becomes

$$\{\bar{K}_{NN}U_N + K_{NM}[G_{MN}U_N + \theta_M]\} = \left\{\bar{P}_N - G_{MN}^T Q_M\right\}$$

and using Eq. (5-7) to expand the result to get

$$[\bar{K}_{NN} + K_{NM}G_{MN} + G_{MN}^T K_{MN} + G_{MN}^T K_{MM}G_{MN}]\{U_N\} = \{\bar{P}_N + G_{MN}^T P_M\} - [K_{NM} - G_{MN}^T K_{MM}]\{\theta_M\}$$

or

$$\underbrace{[K_{NN}]\{U_N\} = \{P_N\}}_{\text{current}} \quad \underbrace{-[K_{NM} + G_{MN}^T K_{MM}]\{\theta_M\}}_{\text{new}} \tag{5-9}$$

Examples

As a simple example consider a horizontal rod structure. The left and right ends are clamped. However as Figure 5-3 shows, there is a break in the structure of 0.1 units of length. If the grid id at the left end of the break is A and the grid id at the right end of the break is B, we can tie the structure by the MPC equation

$$U_B - U_A = 0.1$$

In the example, we show how to write the MPC equation in two ways. The first way uses the standard “SLACK” variable method. The second way uses the new MPCY entry.

Figure 5-3 Example Rod Structure for MPCY

```
SOL 101
CEND
TITLE = DEMONSTRATE USE OF MPCY
  SPC   = 300
  MPC   = 300
  LOAD= 300
  DISPL= ALL
  OLOAD= ALL
  MPCFO = ALL
  SPCFO= ALL
```

```
  ELFOR = ALL
BEGIN BULK
$
PROD   1      1      1.
MAT1   1      1.+7          0.
GRDSET                                                 3456
$
$
$                   STANDARD "SLACK" VARIABLE METHOD
$
$ |1               2               3   4               5|
$ |O---------------O---------------O   O---------------O|
$ |       1               2                   3         |
$
$         GRIDs 3 and 4 have a 0.1 unit gap between them
$         We wish to impose relationship in x-direction
$                     U4 = U3 + 0.1
$         When GRID 3 has a x-direction load of 1.+5
$
GRID    1               0.      0.      0.              123456
GRID    1               100.    0.      0.              23456
GRID    1               200.    0.      0.              23456
GRID    1               200.1   0.      0.              23456
GRID    1               300.1   0.      0.              123456
$
SPOINT 6
$
CROD   1      1      1      2
CROD   2      1      2      3
CROD   3      1      4      5
$
FORCE  300    3              1.+5   1.
$
SPC    300    6      1      0.1
MPC    300    4      1      -1.    3      1      1.
              6      1      1.
$               ^
$               |
$               |
$               |
$             SPOINT with 0.1 in displacement gap
$
$
$ ----------------> NEW MPCY METHOD <----------------
$
$ |11             12              13   14             15|
$ |O---------------O---------------O   O---------------O|
$ |      11              12                  13         |
$
$         GRIDs 13 and 14 have a 0.1 unit gap between them
$         We wish to impose relationship in x-direction
$                     U14 = U13 + 0.1
$         When GRID 13 has a x-direction load of 1.+5
$
GRID    11              0.      1.      0.              123456
GRID    12              100.    1.      0.              23456
GRID    13              200.    1.      0.              23456
GRID    14              200.1   1.      0.              23456
GRID    15              300.1   1.      0.              123456
$
```

```
CROD    11      1       11      12
CROD    12      1       12      13
CROD    13      1       14      15
$
FORCE   300     13              1.+5    1.
$
MPCY   300      14      1        1.     0.1
                13      1       -1.
ENDDATA
```

The results are:

```
D I S P L A C E M E N T   V E C T O R

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G      0.0            0.0            0.0            0.0            0.0            0.0
            2      G      3.000000E-01   0.0            0.0            0.0            0.0            0.0
            3      G      6.000000E-01   0.0            0.0            0.0            0.0            0.0
            4      G      7.000000E-01   0.0            0.0            0.0            0.0            0.0
            5      G      0.0            0.0            0.0            0.0            0.0            0.0
            6      S      1.000000E-01
           11      G      0.0            0.0            0.0            0.0            0.0            0.0
           12      G      3.000000E-01   0.0            0.0            0.0            0.0            0.0
           13      G      6.000000E-01   0.0            0.0            0.0            0.0            0.0
           14      G      7.000000E-01   0.0            0.0            0.0            0.0            0.0
           15      G      0.0            0.0            0.0            0.0            0.0            0.0


                                                 L O A D   V E C T O R

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            3      G      1.000000E+05   0.0            0.0            0.0            0.0            0.0
            6      S      0.0
           13      G      1.000000E+05   0.0            0.0            0.0            0.0            0.0


                              F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            1      G     -3.000000E+04   0.0            0.0            0.0            0.0            0.0
            5      G     -7.000000E+04   0.0            0.0            0.0            0.0            0.0
            6      S      7.000000E+04
           11      G     -3.000000E+04   0.0            0.0            0.0            0.0            0.0
           15      G     -7.000000E+04   0.0            0.0            0.0            0.0            0.0


                                         F O R C E S   I N   R O D   E L E M E N T S     ( C R O D )
      ELEMENT             AXIAL                                                 ELEMENT             AXIAL
        ID.               FORCE          TORQUE                                   ID.               FORCE          TORQUE
            1         3.000000E+04    0.0                                           2          3.000000E+04    0.0
            3        -7.000000E+04    0.0                                          11          3.000000E+04    0.0
           12         3.000000E+04    0.0                                          13         -7.000000E+04    0.0


                                F O R C E S   O F   M U L T I P O I N T   C O N S T R A I N T

     POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
            3      G     -7.000000E+04   0.0            0.0            0.0            0.0            0.0
            4      G      7.000000E+04   0.0            0.0            0.0            0.0            0.0
            6      S     -7.000000E+04
           13      G     -7.000000E+04   0.0            0.0            0.0            0.0            0.0
           14      G      7.000000E+04   0.0            0.0            0.0            0.0            0.0


                                         * * * END OF JOB * * *
```

6 Optimization

- Topology Optimization Enhancements
- Automatic External Superelement Optimization (AESO)
- Randomization of a User’s Input Data File (Pre-release)
- Random Elimination of Element Types (Pre-release)

Topology Optimization Enhancements

Introduction

Topology optimization capability was first released in 2005 and a number of manufacturability constraints were added to MSC Nastran 2005 r3. New features have been added to this release based on feedback from clients. With these enhancements, MSC Nastran 2007 SOL200 is able to support: combined topology, sizing, and shape optimization, multiple mass reduction targets, cyclical symmetry constraints, and adjoint design sensitivity analysis for inertia relief of static analyses. A major performance enhancement of module DOPR1 has been made to speed up minimum member size control and sizing optimization with many thousands variables.

Benefits

Combined Topology, Sizing, and Shape Optimization

It is often recommend that topology optimization is first used to find efficient design concepts at the early design stage whereas sizing and/or shape optimization is used for detail design based on the topology design proposals at a later design stage. The use of topology, sizing, and shape optimization simultaneously may find possible better design since the interaction of sizing and/or shape variables with topology optimization is considered during a single design optimization process. Another benefit of this feature is that the DRESP2 BETA function (minimize the maximum responses) is now available to topology optimization.

Cyclical Symmetry Constraints

A mirror symmetry constraint was added to MSC Nastran 2005 r3. It is also desirable to design a rotational symmetric component or system. By using cyclical symmetry constraints in topology optimization, a rotational symmetric design can be obtained regardless of the boundary conditions or loads. This cyclical symmetric constraint capability can be used for irregular finite element meshes.

Adjoint Design Sensitivity Support for Inertia Relief

A direct design sensitivity analysis method is only available for inertia relief of static analyses in previous versions. The direct method is not affordable computationally for topology optimization since many thousands of variables are often involved. The adjoint design sensitivity analysis method has been developed to benefit not only inertia relief topology optimization but also inertia relief sizing optimization with many design variables. The adjoint method is automatically selected when it is more efficient computationally.

Multiple Mass Target

Type one response FRMASS (DRESP1=FRMASS) used to be total fraction mass of topological designed properties. Feedback from industrial users have shown that it is desirable to set up different mass

reduction targets on multiple designed parts for a built-up structures. This enhancement reflects this requirement.

Module DOPR1 Performance Enhancements

When a large value is given for minimum member size TDMIN, a significant CPU time is spent on module DOPR1 in previous versions. With this enhancement, a substantial performance speedup (3-100 times) is achieved for minimum member size control. Module DOPR1 is also enhanced to efficiently support optimization problems with many thousands sizing design variables (for example, a sizing optimization deck with 320,000 design variables, the DOPR1 module in previous MSC Nastran versions and other Nastran products requires 16,000 seconds, MSC Nastran 2007 DOPR1 module requires only 10 seconds).

Input

The TOPVAR Bulk Data entry has been enhanced to provide cyclical symmetry constraints. To select a topologically designable region, the user needs to specify a group of elements by using a Bulk Data entry, TOPVAR. The cyclical symmetry constraints are then applied on all elements referencing the given property on TOPVAR entry.

The enhanced TOPVAR format is:

1	2	3	4	5	6	7	8	9	10
TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	“SYM”	CID	MSi	MSi	MSi	CS	NCS		
	“CAST”	CID	DDi	DIE					
	“EXT:	CID	EDi						
	“TDMIN”	TV							

Field	Contents
PID	Property entry identifier (Integer > 0). This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1 and DVPREL2 entries. Topology and sizing variables cannot share the same properties. (Integer > 0)
“SYM”	Indicates that this line defines symmetry constraints.
CID	Rectangular coordinate system ID used for specifying manufacturing constraints. See Remark 2. (Blank or Integer > 0; Default = 0)
CS	Cyclical symmetry axis (character X, Y, Z). See Remark 3.
NCS	Number of cyclical symmetric segments in 360 degrees. (Integer > 0). The angle for one segment is calculated by 360/NCS. See Remark 3..

New Remarks:

1. The topologically designable element properties include PROD, PBAR, PBARL, PBEND, PBEAM, PBEAML, PSHELL, PSHEAR, PSOLID, and PWELD. Multiple TOPVARs are allowed in a single file. Combined topology, sizing, and shape optimization is supported in a single file. However, TOPVAR cannot be used with DVMREL1 and DVMREL2 entries.
2. Only CORD1R and CORD2R can be used as a referenced coordinate system to specify topology manufacturing constraints. Only one reference coordinate system CID is allowed for each TOPVAR entry.
3. The first cyclical symmetry segment starts at the X-axis when CS=Z (at Z-axis when CS = Y, and at the Y-axis when CS = X). One cyclical symmetry constraint can be combined with one mirror symmetry constraint as long as the axis of cyclic symmetry is normal to the plane of mirror symmetry. For example, MSi = YZ and CS = X, MSi = XZ and CS = Y, and MSi = XY and CS = Z. This feature can also be used for rotational parts with < 360 degrees but NCS must be given in 360 degrees.
4. For “EXT” constraints, possible combinations are (ED=X, MSi=XY, and/or ZX or CS=X), (ED=Y, MSi=YZ, and/or XY or CS=Y), (ED=Z, MSi=ZX, and/or YZ or CS=Z).
5. For “CAST” constraints, possible combinations are (DD=X or X-, MSi=XY and/or ZX or CS=X), (DD=Y or Y-, MSi=YZ and/or XY or CS=Y), (DD=Z or Z-, MSi=ZX and/or YZ or CS=Z).

Modified Type One Responses - Fractional Mass

To allow the user to put different mass reduction constraints on multiple designed properties, RTYPE=FRMASS entry has been extended to provide property ID (PID) at attribute field ATTi. The format for this extension is shown in Table 6-1.

Table 6-1 Modified RTYPE=FRMASS

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
FRMASS Remark 1	BLANK	BLANK	BLANK or Property ID (PID)

Remark:

1. RTYPE= FRMASS (mass fraction of topological designed elements) entries are used for topology optimization or combined topology, sizing and or shape optimization. ATTi=Blank is for total mass fraction of all topological designed properties. ATTi=PID is the mass fraction of topological designed property PID.

Guidelines and Limitations

- Although combined topology and sizing optimization is supported, TOPVAR and DVPREL1/2 entries cannot reference the same property ID (PID).
- While FRMASS is calculated for topological designed properties only, RTYPE=WEIGHT computes total weight including all designed and non-designed parts. For combined topology and sizing/shape optimization problems, it is recommended that RTYPE=FRMASS is used for topological designed property mass reduction constraints and RTYPE=WEIGHT is used for total mass reduction constraints.

Example 1 – Wheel (wheeltop.dat)

A wheel model shown in Figure 6-1 is used to demonstrate MSC Nastran topology optimization cyclical symmetry capabilities. The wheel is modeled with six-sided solid elements (CHEXA). The wheel outer layers and bolts are non-designable. One load case is considered. The structural compliance is minimized (i.e., minimize the total strain energy of the structure) with a mass target 0.1 (i.e., remove 90% of the material). Although the load is not cyclically symmetric about the Y-axis, the design is required to be cyclically symmetric about the Y-axis with five segments.

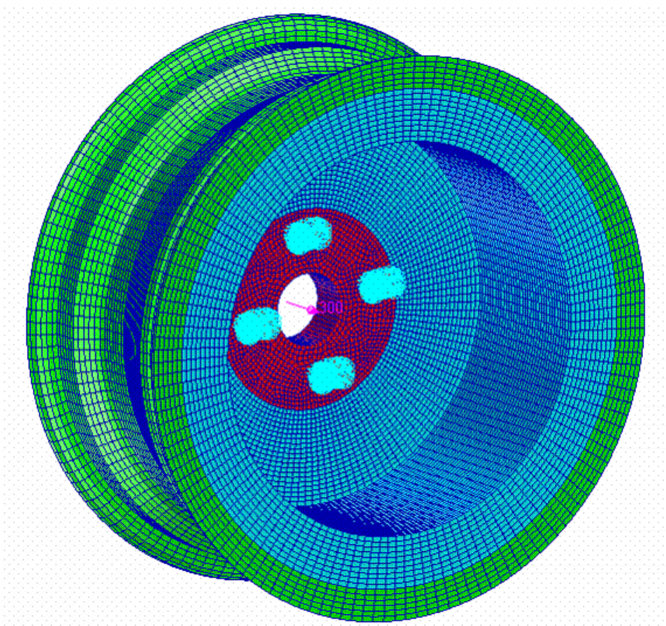


Figure 6-1 Wheel

Input

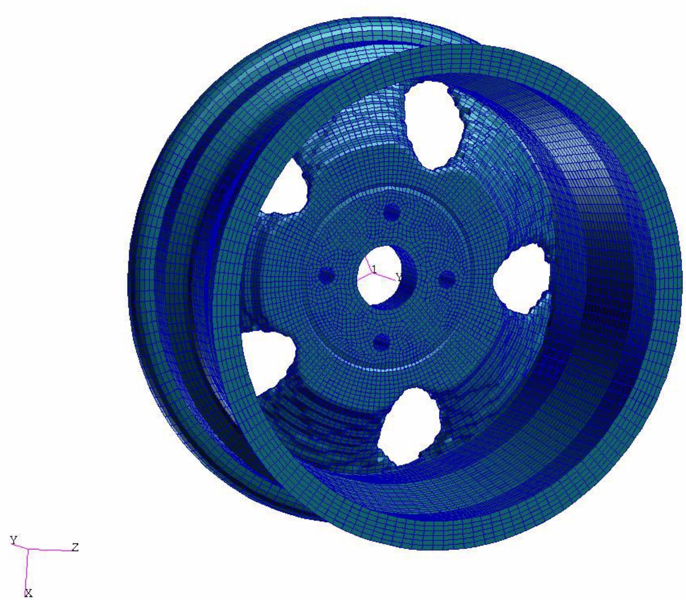
The input data for this example related to topology optimization model is given in Listing 6-1. The coordinate system CORD2R = 1 is created to be used to specify cyclical symmetric constraints. The field CS on the SYM line is Y-axis with NCS=5.

Listing 6-1 Input File for Example 1

```
DESOBJ = 10
DESGLB = 1
ANALYSIS = STATICS
SMETHOD = ELEMENT
SUBCASE 1
   SPC = 2
   LOAD = 2
BEGIN BULK
CORD2R   1            10.512  33.3312 12.9921 -22.209833.3312 4.88385
        28.388  33.3313 -19.7297
DCONSTR  1       2               .1
TOPVAR   1       PSOLID  PSOLID  .1                               2
         SYM        1                             Y        5
DRESP1   2       FRM     FRMASS
DRESP1   10      COMP    COMP
```

Output

Figure 6-2 shows the topology optimized result that is smoothed by using Patran. It is noticed that cyclical symmetry is obtained even though the loading is not cyclically symmetric.



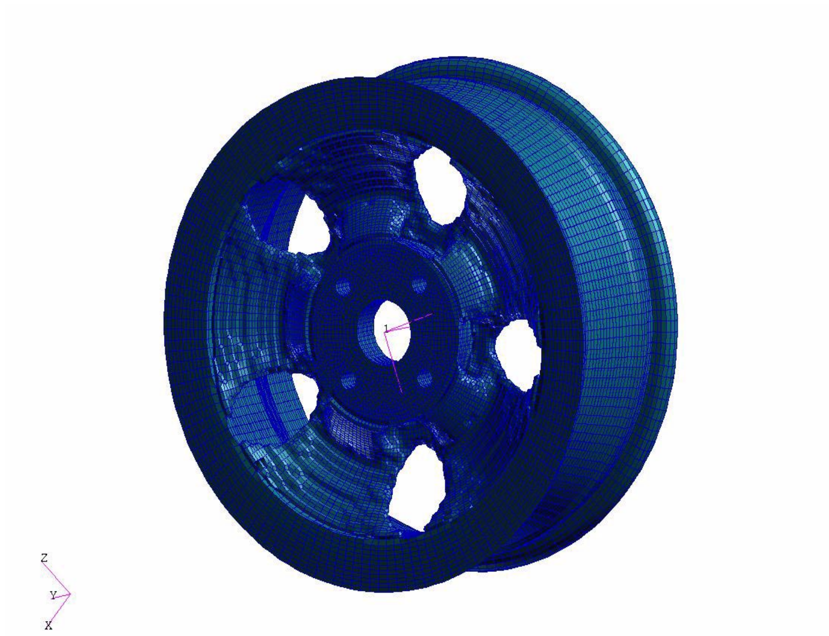


Figure 6-2 Wheel Topology Design

Automatic External Superelement Optimization (AESO)

Introduction

The Manual External Superelement Optimization capability (MESO) has been available since the MSC.Nastran 2004. In this technique, the user manually partitions the analysis model into two parts: a designed part and a non-designed part. The latter is treated as one external part superelement while the former is defined as a residual structure. A creation run is performed which applies Component Mode Synthesis (CMS) or Static Condensation to the part superelement and stores the resulting boundary matrices in a database or a punch file. The original optimization task is performed on the assembly run that assembles those boundary matrices into the residual model for solving system solutions. The strategy is most efficient when the size of the design model (or residual model) is much smaller than the size of the original analysis model. However, although the feature is efficient in CPU time, since both files of both creation and assembly runs must be created by the user, significant effort in manual partitioning the model might outweigh the performance gain.

The AESO capability presented here extends the MESO in an important way: rather than requiring the user to segregate his large model into a designed and non-designed part, the process does this automatically by identifying which parts of the finite element model are affected by the design task. In essence, the new AESO capability provides an efficient and accurate solution in a user friendly way.

Benefits

Several major benefits are:

1. Removes a tedious and error prone task from the user in preparing the user input data.
2. Does not require the user to be knowledgeable in the specialized area of superelements in general and external superelements in particular.
3. Provides an efficient and accurate approach for large-scale design optimization tasks.
4. Enable the performance of various design studies rapidly once the model has been divided into a designed and non-designed part. Examples of this are the setting up of different design constraints and objective in the studies to gain insight into the design and the available trade-offs or the applying of various frequency excitation loadings in the frequency response analyses.

Methodology

A complete AESO task involves two separate MSC Nastran job runs: 1) the first run is an AESO creation run (or simply creation run) whose logical flow is described in Figure 7-3 and 2) the second run is an AESO assembly run (or simply assembly run) whose logical flow is described in Figure 7-5.

As shown in Figure 7-3, the creation run automatically partitions the original analysis model into the residual (the designed part) and external SE (the non-designed part). This automatic partition procedure will assign the following grid points to the residual:

1. All grid points that belong to a design model consisting of DRESP1, DVGRID, DVPRELi, DVMRELi and DVCRELi entries;
2. All grid points that are referenced on all static or dynamic loading entries such as DAREA, DPHASE, FORCE, MOMENT, PLOADi, TEMP entries;
3. All grid points for a rigid element that has one or more connecting grid belonging to the residual;
4. Any grid point that is a dependent grid on an MPC entry

After the automatic partition procedure, a new user input file is created from the residual (Figure 7-4). Then, the remainder of the creation run is the application of Static Condensation and/or Component Modes Synthesis procedures to produce stiffness, mass, damping boundary matrices. After the creation run is complete, a Nastran database is saved to store the boundary matrices and a .asm file is also created to include superelement boundary connection information.

The assembly run is similar to a conventional SOL 200 task as shown in Figure 7-5 by utilizing all three types of data generated from the creation run (Figure 7-4). The original optimization problem is solved by assembling the boundary matrices into the residual for the system solutions.

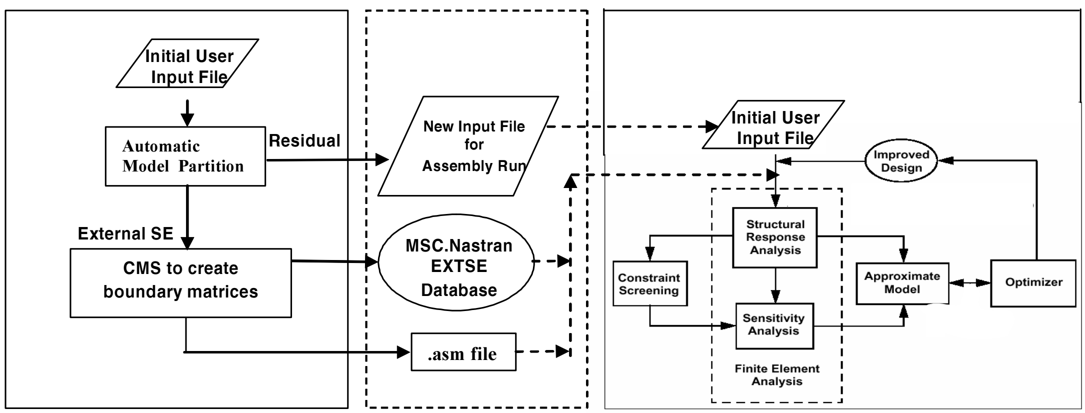


Figure 7-3 AESO Creation Run

Figure 7-4 Output of Creation Run and Input of Assembly Run

Figure 7-5 AESO Assembly Run

Input

1. Two parameters are added on the DOPTPRM entry for an AESO job:

   AUTOSE - flag to request an AESO job (integer 0, 1, Default = 0). AUTOSE = 1 activates an AESO creation run.

   DRATIO - the threshold value that is used to turn off an active AESO job if the ratio of the size of the design model to that of the analysis model is greater than DRATIO (Real > 0.; Default = 0.1).

2. An ASSIGN statement with a logical key name 'AESO' is placed in the FMS to specify an input file name for the assembly run.

To illustrate ideas behind the input and output for an AESO task, a test problem (aeso1.dat) is used here.

Figure 6-6 shows a sample model whose upper left portion covering elements 18 to 42 (SE 1) is the non-designed part while the rest of structure is the residual structure.

The listing below is a condensed version of the creation run file (aeso1.dat) that only shows the required user input to invoke an AESO creation run. The **assign aeso='aeso1_2.dat'** statement is specified in the FMS section and **autose 1** and **dratio 0.9** are requested on the DOPTPRM entry. The DRATIO=0.9 here overrides the default.

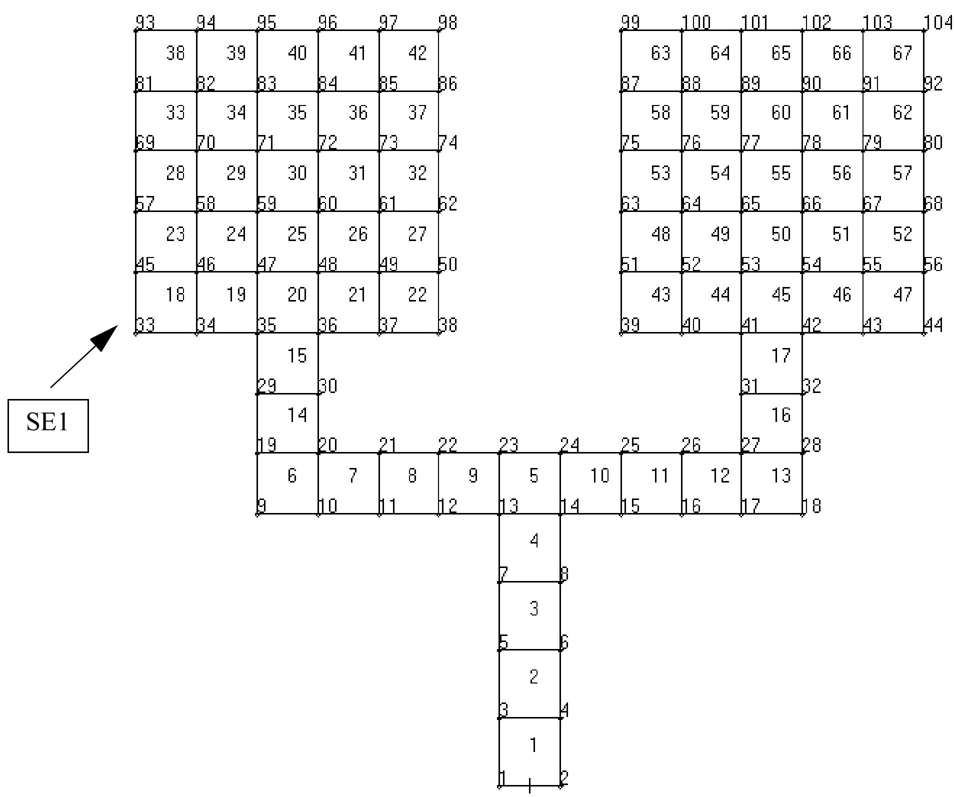


Figure 6-6 A Sample Model

```
assign aeso='aeso1_2.dat'
SOL 200
CEND
desobj(max) = 1
analysis = modes
…..
Begin Bulk
……
doptprm  desmax 10      autose  1       dratio  0.9
enddata
```

Figure 6-7 Condensed Version of the Creation Run File, dseoptl.dat

Outputs

As shown in Figure 7-4, three types of data generated from the creation run are saved in the working directory: a Nastran database, a .asm file and a new input file for the assembly run (or an assembly file). This section describes each of these items and explains how they are used in the assembly run. In addition, some special print outputs from the creation run are shown that display the model partition information.

Nastran Database Files

Two Nastran database files: aeso1.MASTER and aeso1.DBALL are automatically saved when the creation run is submitted with SCR=NO option. Notice the size of the database is much smaller than the regular Nastran database because only boundary matrices are stored. The Master file will be referenced by an ASSIGN statement in the new input file described below.

The .asm File

The 'aeso1.asm' file that includes the boundary connection information follows. As shown in Figure 6-6, this problem has only two boundary points, 35 and 36 between the residual and the external superelement 1. This file is accessed through an INCLUDE command in the assembly file.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$ ASSEMBLY PUNCH (.ASM) FILE FOR EXTERNAL SUPERELEMENT        1
$
$ THIS FILE CONTAINING BULK DATA ENTRIES PERTAINING TO
$ EXTERNAL SUPERELEMENT        1 IS MEANT FOR INCLUSION
$ ANYWHERE IN THE MAIN BULK DATA PORTION OF THE ASSEMBLY RUN
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
SEBULK         1EXTERNAL          MANUAL
$
SECONCT        1       0 5.0E-05
              35      35      36      36
$
$ BOUNDARY GRID DATA
$
GRID    35              -3.6    6.      0.
GRID    36              -2.8    6.      0.
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

New Input File for the Assembly Run

Notice that the name of this file, aeso1_2.dat is specified on the ASSIGN AESO statement in the creation run (Figure 6-7). It is a standard Nastran input file. The AESO specific contents in the Executive Control, Case Control and Bulk Data Sections are listed in Figure 6-8 and are described below.

```
nastran rseqcont=1
assign sel=   './aeso1.MASTER'
dblocate datablk(EXTDB) logical=se1,
CONVERT(SEID=1)
SOL 200
CEND
```

Executive Control Section

```
desobj(max) = 1
analysis = modes
…….
subcase 10
method = 1
spc = 10
$
```

Case Control Section

```
begin bulk
include './aeso1.asm'
…….
doptprm  desmax 10
enddata
```

Bulk Data Section

Figure 6-8   Highlights of the Assembly Run File, aeso1_2.dat

1. Executive Control Section

The Nastran rseqcont=1 statement instructs the input file processor to ignore all continuation fields. This statement is automatically created in this file regardless of whether the creation run has it or not.

The next two statements assign the Nastran Master database file and locate the EXTDB datablock that stores various boundary matrices.

Notice that the other statements in Executive Control Section of the creation run are not retained.

2. Case Control Section

   The whole Case Control section of the creation run is retained in the assembly file.

3. Bulk Data Section

   This section completely defines the residual structure. The include './aeso1.asm' command allows the assembly run to access the .asm file created from the creation run. In addition, the autose 1 and dratio 0.9 have been removed from the doptprm entry of the creation run.

Special Print Outputs from Creation Run

The following output is taken from the aeso1.f06 file. It displays detailed information about the model partition. You may use Figure 6-6 to help read the printout here. Notice that the Superelement 1 covers the non-designed part while the residual (or Superelement 0) covers the designed part.

Listing 6-2 Printout Showing Model Partition of Designed and Non-Designed Parts

```
BOUNDARY SEQUENCE ASSIGNMENT TABLE
      BOUNDARY
      SEQUENCE ID  -------------------------- ASSIGNED TO POINT ID (SUPERELEMENT) ---------------------------------
         1B        35 (        0)        35 (        1)
     2B      36 (      0)      36 (      1)

                                              SUPERELEMENT        0
                                           LIST OF  INTERIOR POINTS        ( TOTAL NO. OF INTERIOR POINT = 70 )
       INDEX      -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-      -10-
           1        1         2         3         4         5         6         7         8         9        10
          11       11        12        13        14        15        16        17        18        19        20
          21       21        22        23        24        25        26        27        28        29        30
          31       31        32        39        40        41        42        43        44        51        52
          41       53        54        55        56        63        64        65        66        67        68
          51       75        76        77        78        79        80        87        88        89        90
          61       91        92        99       100       101       102       103       104        1B        2B
                                             SUPERELEMENT        0
                                           LIST OF  ELEMENTS              ( TOTAL NO. OF ELEMENTS       =  42 )
      INDEX       -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-      -10-
          1         1         2         3         4         5         6         7         8         9        10
         11        11        12        13        14        15        16        17        43        44        45
         21        46        47        48        49        50        51        52        53        54        55
         31        56        57        58        59        60        61        62        63        64        65
        41        66       67
                                             SUPERELEMENT        1
                                          LIST OF  EXTERIOR POINTS        ( TOTAL NO. OF EXTERIOR POINT =   2 )
      INDEX       -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-      -10-
      1           1B       2B
                                             SUPERELEMENT        1
                                           LIST OF  INTERIOR POINTS        ( TOTAL NO. OF INTERIOR POINT = 34 )
      INDEX       -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-      -10-
          1        33        34        37        38        45        46        47        48        49        50
         11        57        58        59        60        61        62        69        70        71        72
         21        73        74        81        82        83        84        85        86        93        94
        31        95        96        97        98
                                             SUPERELEMENT        1
                                           LIST OF  ELEMENTS              ( TOTAL NO. OF ELEMENTS       = 25 )
       INDEX      -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-      -10-
          1        18        19        20        21        22        23        24        25        26        27
         11        28        29        30        31        32        33        34        35        36        37
         21        38        39        40        41        42
```

Guidelines and Limitations

- You may adjust DRATIO to allow an assembly run with larger or smaller residual model. The UIM 7824 provides brief information about the sizes of your analysis model and design model in terms of number of the grid points.

```
*** USER INFORMATION MESSAGE 7824 (DSGRDM)
    THE NUMBER OF GRID POINTS IN THE ANALYSIS MODEL =        104.
    THE NUMBER OF GRID POINTS IN THE DESIGN MODEL =         70.
    THE DESIGN MODEL COMPRISES     67.3 PERCENT OF THE ANALYSIS MODEL.
```

- For an AESO job with Analysis=MODES or MFREQ, it is recommended to activate the matrix domain based decomposition with ‘domainsolver acms(partopt=dof)’ in the Executive Control section to speed up the CMS procedure.
- Always specify the ASSIGN AESO=’filename.ext’ statement in the creation run to define the name of the assembly file. Directly assigning the original job name to filename should be avoided and will cause the assembly run to fail with User Fatal Message 713. A good practice is to add some suffix to the original file name such as myjob_2nd.dat where myjob is the original file name.

```
*** USER FATAL MESSAGE 732 (OPFUNT)
     LOGICAL NAMES 'INPUT    ' AND 'AESO     ' ARE ASSIGNED TO THE SAME PHYSICAL FILE.
     USER INFORMATION: PHYSICAL FILE NAME 1: ./abc.aeso
                       PHYSICAL FILE NAME 2: ./abc.aeso
     USER ACTION:  CHANGE FILE NAME ON ONE OF THE ASSOCIATED ASSIGN STATEMENTS.
```

- When submitting the AESO creation run, use SCR=NO option. Otherwise, the Nastran database will not be retained after the creation run is done. However, it is optional for submitting an assembly run.
- After the creation run is complete, check the following user information message in the f06 file to ensure the job is terminated successfully.

```
^^^
 ^^^ USER INFORMATION MESSAGE 9181 (FEA)
 ^^^ THE JOB IS TERMINATED FOR AN AUTO   EXTERNAL CREATION RUN
 ^^^
```

- The assembly input file may be modified to perform various design studies as long as the changes do not affect the boundary matrices stored in the database.
- If the AESO creation run includes a GRAV entry, it will be terminated with the following message. The same applies to TEMPD entry. They should be removed from the file if they are temporary inactive for the current task. Gravity and TEMPD loads are not supported with AESO since they apply loads to the entire structure and therefore block any partitioning.

```
*** USER FATAL MESSAGE 7699 (DSGRDM)
        A GRAV Bulk Data entry is specified in an AESO creation run.
        USER INFORMATION: The AESO run does not support the GRAV entry.
```

- Duplicate GRID entries are allowed in the .asm file and in the assembly input file. They will be automatically removed during the assembly run within the location tolerance specified by the TOL field on the SECONCT entry. For the AESO jobs, the default of the location tolerance has been increased to 5.E-5 from the original 1.E-5. However, due to numerical imperfection, this tolerance may need to be adjusted particularly for the cases in which the boundary grid points are defined in one or multi-levels of coordinate systems.

- Bulk Data parameter SEMAPPRT can be used to control the printout of the model partition information shown in Listing 6-2. For example, Setting SEMAPPRT to 0 will turn off the printout.
- If the AESO task includes DVGRID entries, make sure that the grid points referenced by DVGRID entries are inside the residual. Since the grid points on DVGRID entries vary during the design process, including them as part of boundary grid points will invalidate the invariance of those boundary matrices. Currently, the grid points on the DVGRID entries will not be automatically assigned to be inside the residual. ASET and ASET1 entries can be used to create an enclosure or a barrier to ensure the grid points referenced on the DVGRID entries are always placed inside the residual.
- Since all the CORD1i entries are automatically converted to CORD2i entries during the AESO creation run. The DVGRID entry should not reference the grid points that define the CORD1i entry.
- The AESO tasks do not support acoustics response.

Examples

Road Response Optimization with CAMARO Model with Analysis=MFREQ

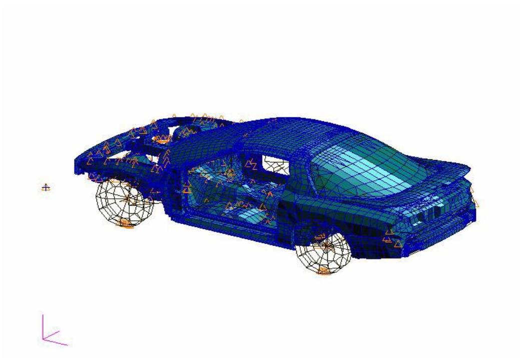


Figure 6-9 CAMARO Model

A CAMARO model provided by GM as shown in Figure 6-9 is modeled with 23K grid points, 37K elements and 137K DOFs, random inputs applied on left and right suspension including cross-correlation, to simulate road conditions.

The design task is to vary 9 spring constants of engine mounts modeled by elastic elements in two design cases:

Case A: minimize the sum of RMS acceleration at Driver’s seat and passenger’s seat and limiting the PSD response at steering column and

Case B: minimize the RMS acceleration at Driver’s seat and maintain frequency dependent limits on driver’s seat.

Because these design variables and responses are limited to a small part of the total vehicle, the size ratio of design model vs. analysis model is <1.% (the residual structure has 298 elements, 163 interior points and 130 boundary points while the external superelement has 33345 elements and 22761 interior points).

To demonstrate the efficiency of the AESO capability, the tables below present the results from a regular run and the AESO job. First, the same final designs are achieved by both jobs. However, the AESO job takes 1/5 of the time to complete Case A. Furthermore, since Case B just changes the design objective and constraint formulation, the boundary matrices from the creation run are invariant. Therefore, only the assembly run for Case B is required. Comparing the time spent on this assembly run with the single run as shown in Case B in the bottom half of the table, the speed up is 16X.

Case A	Initial OBJ	Final OBJ	Init Max Const.	Final Max Const.	# Design Cycle	Clock Time (Minutes)
Full Model Run	0.1534	-0.0639	0.1329	-0.2102	9	76
AESO Creation Run	N/A	N/A	N/A	N/A	N/A	10
AESO Assembly Run	0.1534	-0.0639	0.1329	-0.2102	9	7
Total time of two AESO runs						16
Performance Ratio						5

Case B	Initial OBJ	Final OBJ	Init Max Const	Final Max Const	# Design Cycle	Clock Time (Minutes)
Full Model Run	0.0713	0.0586	0.2855	0.0081	14	110
AESO Creation Run	N/A	N/A	N/A	N/A	N/A	0
AESO assembly Run	0.0713	0.0584	0.2855	-0.0108	14	7

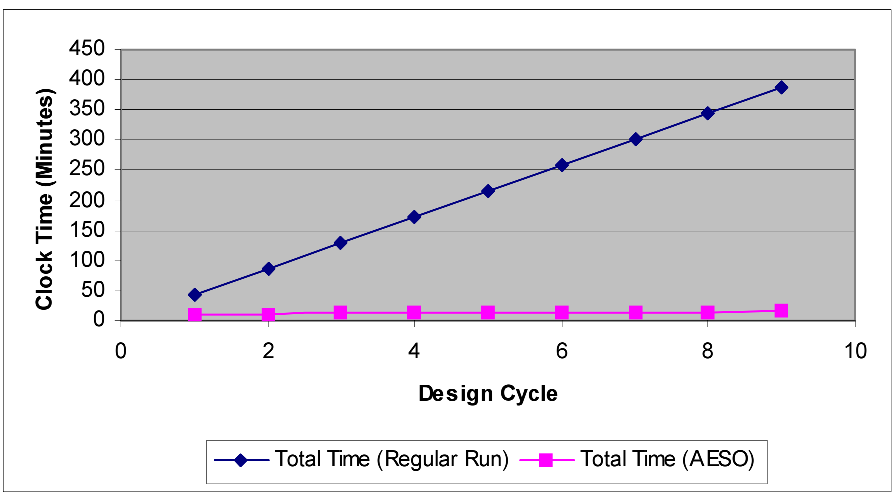
Road Response Optimization with CAMARO Model with Analysis=DFREQ

In general, direct frequency approach is more expensive than the modal approach. However, when the size of the residual model is small, the Direct Frequency approach can also be performed efficiently as shown here. Case A of the same road response optimization problem are solved with both a regular direct frequency response optimization and the direct frequency AESO. The table below presents the results.

Again, the same final designs are achieved. However, the speed up obtained by the AESO over the regular job is 27X.

Case A	Initial OBJ	Final OBJ	Init Max Const	Final Max Const	# Design Cycle	Clock Time (Minute)
Full Model Run	0.1535	0.1327	-0.0631	-0.2073	9	388
AESO Creation Run	N/A	N/A	N/A	N/A	N/A	10
AESO Assembly Run	0.1534	0.1327	-0.0636	-0.2062	9	5
Total time of two AESO runs						15
Performance Ratio						27

The following figure shows the plot of the total clock time spent by two runs vs. design cycle and it clearly shows that the AESO is much more efficient than the regular run because the cost per design cycle for AESO (or the slope of the curve for the AESO run) is much smaller than that for a regular run (the slope of the curve for the regular run). Although for this example, the one time cost for the AESO due to the creation run is smaller than that for a regular run, in general, this one time cost may be larger because the time spent by CMS procedure on mass matrix reductions could be expensive.



Miscellaneous

Prior to this release, SOL200 supports a limited Part optimization capability while it fully supports the traditional Superelement optimization. Notice that the traditional SE is defined with SESET Bulk Data entry while the Part is defined with Begin Super Bulk Data entry. In addition, the Part may be defined either as External or Internal.

The enhancement added in this release provides a robust Part optimization capability as long as the design model is in the residual (or an upstream Part can not be designed or constrained). One example of its application would be: an airplane wing is modeled as an internal Part while the engine is modeled as an external Part. Then these two are attached to the fuselage, a residual that will be designed.

The Table below summarizes the features of Superelement Optimization and Part Optimization in SOL200 in this new release.

Item	Feature	Support in SOL 200	Design Model in Upstream SE or Part
1	Traditional SE	Yes	Yes
2	Ext. Part	Yes	No
3	Int. Part	Yes	No
4	2+3	Yes	No
	1+2,1+3,1+2+3	No	No

To illustrates how to create a Part optimization job, the sample model as shown in Fig 7-6 is used here. The top left squared region is modeled as Part 1 (elements 18 to 42) while the top right squared region is modeled as Part 2 (elements 43 to 67). The rest is modeled as a residual (elements 1 to 17). Grid points 1 and 2 are fully constrained. The design task is to maximize the first natural frequency by varying the thickness of elements 1 to 17 within a given range.

The following listing is a condensed version of dseopt18.dat that shows three major Bulk Data sections. Its full version can be obtained from the Test Problem Library. The unique feature of a Part optimization job is the multiple Bulk Data sections and how the design model definition is placed. Three sections in this job are:

1. Main section (between Begin Bulk and Begin Super = 1) that defines the residual model. Notice that all the designed entries such as DESVAR, DVPREL1, DRESP1 entries are defined in this section because the design model must be within the residual.
2. Section for Part 1 (between Begin Super = 1 and Begin Super =2). This section defines Part 1 model.
3. Section for Part 2 (between Begin Super = 2 and Enddata). It defines Part 2 model.

Notice that the design model must be placed in the main Bulk Data section because the design model is required to be in the residual. The detailed steps to define Parts and residual model are not skipped here.

```
SOL 200
diag 8,15,56
CEND
TITLE = Test for Internal Part Optimization
echo = sort
desobj(max) = 1
analysis = modes
subcase 10
spc = 10
method = 1
$
$ residual structure model
$
BEGIN BULK
eigrl,1,,,9
param,post,0
PARAM,GRDPNT,0
PARAM,WTMASS,.00259
CQUAD4  5      1      13     14     24     23

$
$GRDSET                                                         6
GRID    13              -.4     3.6     0.
GRID    14              .4      3.6     0.
GRID    23              -.4     4.4     0.
GRID    24              .4      4.4     0.
$
MAT1,1,30.+6,,.3,.283
PSHELL,1,1,.05,1,,1
$
CQUAD4  14      1       19      20      30      29
CQUAD4  15      1       29      30      36      35
$$
spc1,10,123456,1,2
$
$ design model definition must be in the main BULK Data section
$    2      3    4     5
desvar  1       X1      0.05    0.01    2.0
dvprel11       pshell      1    T
      1     1.0
DRESP1  1       F1      FREQ                    1
```

Main Bulk Data Section

```
$
$ Part 1
$
begin super = 1
$
$ define modal coordinates for CMS
$
$ define q-set for component modes and residual vectors
SPOINT   11001   THRU    11020
QSET1    0       11001   THRU    11020
$
$ define which dofs will be retained (i.e. which dofs
$ will form the attachment to the system model when we
$ create SE10 in se10.dat)
$
ASET1     123456  35      36
$
$     part1.dat
$
CQUAD4  18      9       33      34      46      45
CQUAD4  42      9       85      86      98      97
$
param,k6rot,100.
$
$     boundary grids
$
GRID    35              -3.6    6.      0.
GRID    36              -2.8    6.      0.
$
GRID    97              -2.     10.     0.
GRID    98              -1.2    10.     0.
$
MAT1,1,30.+6,,.3,.283
PARAM,WTMASS,.00259
PARAM,AUTOSPC,YES
PSHELL,9,1,.05,1,,1
$
$ plotels to outline component in assembly run
$
plotel,101,33,35
plotel,102,33,93
plotel,103,93,98
plotel,104,98,38
plotel,105,38,36
plotel,106,35,36
eigrl,1,,,9
```

Section for
Part 1 Model

```
$ Part 2
$
begin super = 2
$
$ define modal coordinates for CMS
$
$ define q-set for component modes and residual vectors
SPOINT   21001   THRU    21020
QSET1    0       21001   THRU    21020
$
$ define which dofs will be retained (i.e. which dofs
$ will form the attachment to the system model when we
$ create SE10 in se10.dat)
$
ASET1     123456  4142
$
$
$    part2.dat
$
CQUAD4  43      2       39      40      52      51

CQUAD4  66      2       90      91      103     102
CQUAD4  67      2       91      92      104     103
$
param,k6rot,100.
$
$     boundary grids
$
GRID    41              2.8     6.      0.
GRID    42              3.6     6.      0.
$

GRID    103             4.4     10.     0.
GRID    104             5.2     10.     0.
$
$
$
MAT1,1,30.+6,,.3,.283
PARAM,WTMASS,.00259
PARAM,AUTOSPC,YES
PSHELL,2,1,.05,1,,1
$
$ plotels to outline component in assembly run
$
plotel,201,39,41
plotel,202,39,99
plotel,203,99,104
plotel,204,104,44
plotel,205,44,42
plotel,206,41,42
$
eigrl,1,,,9
enddata
```

Section for
Part 2 Model

Randomization of a User's Input Data File (Pre-release)

Introduction

The stochastic capability in MSC Nastran is the first step towards a complete and automatic self-randomization of a Finite Element model. The capability currently offers the user the possibility to automatically distribute tolerances and uncertainties with minimum effort. This reduces dramatically the burden on a user wishing to perform large-scale stochastic simulations. In fact, once the stochastic option is triggered, the entire Bulk Data Deck is randomized automatically and without further user intervention. The resulting model, which needs to be incorporated in a Monte Carlo Simulation loop - there are numerous off-the-shelf products which support this capability - possesses unprecedented levels of realism.

In order to make full use of this new development, it is necessary to resort to a multi-run environment, which can spawn a certain number of independent MSC Nastran executions, collect the results, allow the user to perform statistical post-processing. With the self-randomization capability in MSC Nastran, all the user needs to define are the outputs he wishes to monitor, such as stresses, eigen-frequencies, temperatures, displacements, etc. There is no need to define inputs, as these are defined automatically by MSC Nastran. The process is, essentially, error-free.

Benefits

A basic assumption of MSC Nastran is that the inputs to the analysis are known exactly so that the computed responses are also known exactly. This is, of course, an invalid assumption in that there will always be some uncertainty in the inputs with a corresponding variations in the outputs. The MSC Nastran 2007 r1 release of Nastran provides a way of introducing this uncertainty into the analysis process by automatically randomizing user input real numbers based on the input values and statistical quantities that characterize the variation.

Input

The randomization capability is driven by a new STOCHASTICS case control command as shown in the Quick Release Guide. If STOCHASTICS=ALL is used, all real quantities on connectivity (those starting with C), Material and Property entries as well as any loads and SPCD quantities are modified based on a covariance factor of 0.05. A Gaussian distribution is used to randomly select the perturbed quantity with the restriction that the value can be no more that a specified number of standards deviations from the user input mean value. The default number of maximum standard deviations is 3.

Alternatively, the STOCHASTICS command can point to a STOCHAS bulk data entry that provides the ability to selectively randomize different types of input quantities using user specified covariance values and number of allowed standard deviations. In this case, only the types of input specified are randomized so that, for example, it is possible to randomize the loads input while leaving the property values unchanged.

Output

There is no new output produced by this capability at present.

Guidelines and Limitations

The randomization algorithm involves using a random number generator, a Gaussian distribution, the prescribed covariance and a mean value based on the user input to come up with a randomized value that is to be used in the analysis. In order to avoid physically meaningless properties, the random value is prescribed to be within *m* standard deviations of the input value, where *m* is a user input value with a default value of 3.0.

The product of *m* * COV should not be greater than 1.0 to eliminate the possibility of the property changing sign.

The full benefit of this beta capability requires submitting multiple runs with the same randomization parameters. Each would produce a unique randomization and it is possible to collect the results of each of these analyses and produce statistical information on the variability of the responses. At present, MSC does not have software that performs these functions and it is unlikely that users would carry out this type of multiple run analysis on their own.

If the user input property value is 0.0, no randomization occurs. If it is recommended that any property values that are 0.0 (say orientation angle on a PCOMP entry) be set to some non-zero value that is not negligible.

Random Elimination of Element Types (Pre-release)

Introduction

There has been a long time capability in Nastran that allows the user to specify the random elimination of a specified percentage of the CWELD elements that are contained in a bulk data file. This was done using Nastran PARAM CWRANDEL, with an additional CWDIAGP PARAM providing the option of printing the ID's of the deleted elements. This capability has been extended to the CELASi, CFAST, CSEAM, and 1-D mass (CMASSi, CONM1 and CONM2) elements. Further, the user interface has been changed from the NASTRAN PARAM input to the MDLPRM entry.

Benefits

The ability to randomly delete various 1-D elements provides the user with some assessment of the integrity of the design. For instance, if randomly deleting 20%, say, of the CWELD's from a model caused a negligible change in the first ten natural frequencies, this was taken as an indication of the robustness of the structure. Extending this to other element types provides that many more options in this type of analysis. Placing the input on the MDLPRM entry consolidates that input so that the user does not have to deal with the PARAM entry.

Input

The MDLPRM entry has 10 new PARAMi names that support this capability. Five of them (e.g, DELELAS) select the element type to which this random elimination applies and the ratio to be deleted while an additional five (e.g., PRTELAS) provide control as to whether the ID's of the deleted elements are to be printed. The default is that the ID's will not be printed.

Output

There is no new output produced by this capability.

Guidelines and Limitations

The deletion ratio is input as a real number between 0.0 and 1.0 with 0.0 indicating no deletion is to take place while 1.0 would eliminate all elements of the specified type.

It is possible that the elimination of a series of elements will introduce mechanisms in the structure that will cause the analysis to fail. It is the user's responsibility to determine whether this failure has occurred.

A likely scenario for the use of this capability would be to submit the same deck multiple times and determine in the variation in the results. MSC does not offer an automated way of doing this at this time.

7 Rotor Dynamics & Aeroelasticity

Changes to Rotordynamics for MSC Nastran 2007

Unbalance Entry for Frequency Response

The UNBALNC loading has been available for rotordynamic transient response for several versions. Unfortunately, unbalance loading for frequency response had to be input using RLOADi entries. This manual input was uncommonly prone to error. And backward (opposite the rotor spin direction) loading was easily input. To prevent this problem and provide a more natural input, the UNBALNC Bulk Data entry can now be used for frequency response with the rotordynamics option. The entry is selected using the DLOAD Case Control command (note that the transient response solutions must still specify the transient unbalance using the RGYRO Case Control command).

The format for the UNBALNC in frequency response is the same for transient response, except the continuations for force output via EPOINTs are ignored. For further information on the UNBALNC Bulk Data entry see the *MSC Nastran Quick Reference Guide*.

New Parameter and Hybrid Damping Specifications

The specification of damping using parameter input (PARAM,G for structural damping and PARAMs ALPHA1 and ALPHA2 for Rayleigh damping) often resulted in users not knowing what damping, if any, was actually being specified in the model. This is because parameters can be entered anywhere in the Case Control or Bulk Data Sections. For large models with many 'include' files, a rigorous search would be required. To make the damping specifications easier and also allow new damping formulations, such as hybrid damping, a new damping entry has been developed that is selected in the case control. The new entry allows selection of parameter damping, such as structural damping (PARAM, G) and Rayleigh damping (PARAMs ALPHA1 and ALPHA2), and a new hybrid damping. This entry also allows the scaling of material damping (GE on material entries, MATi). The new Bulk Data entry is named DAMPING and is selected by new Case Control commands:

For specifying damping of superelements, use

SEDAMP= n

For specifying damping of the residual structure, use

RSDAMP(STRUCTURE, FLUID, or BOTH)= n

Default= STRUCTURE

Where n specifies the DAMPING Bulk Data entry.

For more information on the DAMPING Bulk Data entry see the *MSC Nastran Quick Reference Guide*.

Updating/Summing of Monitor Points

Introduction

MSC Nastran 2005 r3 greatly expanded the concept of monitor points by modifying the MONPNT1 and introducing the MONPNT2, MONPNT3 and MONDSP1. The MSC Nastran 2007 r1 release of MSC Nastran further expands the monitor points by providing two additional capabilities

1. Allowing for the modification of existing component results for MONDSP1, MONPNT1 and MONPNT3 by a scalar multiple to allow, for example, a change in sign or units.
2. Enabling the weighted summation or two or more MONDSP1, MONPNT1 or MONPNT3's that are of the same type.

Benefits

The ability to update an existing monitor point result is done by applying factors to specified components. An example of the utility of this feature is if the model has been constructed in one set of units (e.g., centimeters) and it is desired to see the results in a different set (e.g, millimeters).

The ability to sum existing monitor point results could be used; e.g., to better present running results along a wing or fuselage.

Input

The new MONSUM Bulk Data entry is used to implement both updating and summing of monitor point results. The description of this entry in the *MSC Nastran Quick Reference Guide* includes examples and numerous remarks that help in understanding the capabilities of this entry.

Output

The output of the monitor point results from the MONSUM Bulk Data entry is identical to that of the underlying monitor point. In the updating scenario, if the NAME field on the entry, is the same as the NAMEij fields, only the final updated result is given. If a new NAME is used, both results are given.

Examples (monsum.dat and monsum3.dat)

Two small test files demonstrate the use of the MONSUM Bulk Data entry. They are both variations of the familiar forward swept wing and are:

monusm.dat- contains some MONSUM Bulk Data entries that are simple enough that the results can be checked by hand

monsum3.dat – sums MONPNT3 results. An interesting feature of this example is that it demonstrates the DMAP enhancements that were required to sum MONPNT3 results produced by using what is referred to as the "mini-ema" method with results using the grid point force recovery method.

Guidelines and Limitations

The remarks for the MONSUM Bulk Data entry in the *MSC Nastran Quick Release Guide* provide a number of guidelines that should be reviewed before applying this new entry. Because it performs both an "update-in-place" and a combining function, it represents a powerful, compact capability.

The two examples shown in the *MSC Nastran Quick Reference Guide* provide a further explanation of this entry. In the first, a new monitor point is being synthesized from three existing monitor point results to provide the user with a blended result that has special meaning to the designer. In this case, results at three stations along the root chord are being summed with different weights for the desired shear and bending moment resultants. Since the COEFi fields are left blank for the shear components, the default of 1.0 is applied. In the second example, the MONSUM is used to convert the units of the monitor point output from English to metric units. Note that in the first case, monitor point results will be presented for the summed quantity as well as for each of the original MONPNT1 quantities that contribute to the sum. In the second case, only the final result is presented since its NAME is shared with the NAMEij attributes. If output in both units systems is desired, one needs to simply provide a unique NAME in order to get both results.

Stripwise Aerodynamic Results

Introduction

An additional monitor point, MONCNCM, has been provided that streamlines the task of providing stripwise lift and pitching moment results for a doublet-lattice type of aerodynamic model.

Benefits

Stripwise aerodynamics; i.e., aerodynamic lift and pitching moments, can be quite useful in visualizing the aerodynamic results on lifting surfaces. In particular, these can be compared with wind tunnel results with the possibility of weighting the computed aerodynamics to match the test results. It is possible to generate these results using an aerodynamic MONPNT1, but the preparation of the input data is a tedious, error-prone process. The new entry automates this process so that it is possible to obtain the results for each aerodynamic strip in the model with a single MONCNCM entry.

Input

The new MONCNCM Bulk Data entry is used to provide the stripwise aerodynamic results. The remarks of this entry in the *MSC Nastran Quick Release Guide* provide guidance in its use. In particular, use of CAERID1=ALL produces results at each aerodynamic strip. The code internally determines if the flat plate panels are abutting and, if they are, considers this a single strip from the leading edge to the trailing edge of the surface made up of the multiple CAERO1 entries.

Output

A new table in the .f06 file that is included as part of the aerodynamic monitor point output. A sample output from the ***moncncm*** test case discussed below is shown here:

A E R O D Y N A M I C   M O N I T O R   P O I N T   I N T E G R A T E D   L O A D S

CONFIGURATION = AEROSG2D   XY-SYMMETRY = ASYMMETRIC   XZ-SYMMETRY = SYMMETRIC

MACH = 9.000000E-01   Q = 4.000000E+01

MONCNCM NAME = ALL   INT GROUP ID = 1   CLASS = STRIP

LABEL = ALL STRIPS FOR THE AIRPLANE

CP = 1   AERODYNAMIC COORDINATE SYSTEM

						FORCE		MOMENT	
STRP	YS	ZS	XREF	REFC	REFS	RIGID	ELASTIC	RIGID	ELASTIC
1	1.250E+00	0.000E+00	0.000E+00	1.0000E+01	2.5000E+01	1.5715E+00	1.5735E+00	1.4352E-01	1.4300E-01
2	1.250E+00	0.000E+00	1.428E+01	1.0000E+01	2.5000E+01	-3.1044E-01	-2.9946E-01	-6.2361E-02	-6.1159E-02
3	3.750E+00	0.000E+00	0.000E+00	1.0000E+01	2.5000E+01	1.2319E+00	1.2336E+00	1.1194E-01	1.1147E-01
4	3.750E+00	0.000E+00	1.283E+01	1.0000E+01	2.5000E+01	4.9708E-03	1.6573E-02	-9.2159E-03	-8.0990E-03
5	6.250E+00	0.000E+00	1.139E+01	1.0000E+01	2.5000E+01	1.1896E+00	1.2036E+00	1.7237E-01	1.7369E-01

6	8.750E+00	0.000E+00	9.948E+00	1.0000E+01	2.5000E+01	1.1173E+00	1.1319E+00	9.4866E-02	9.5348E-02
7	1.125E+01	0.000E+00	8.505E+00	1.0000E+01	2.5000E+01	1.0168E+00	1.0308E+00	3.8398E-02	3.8538E-02
8	1.375E+01	0.000E+00	7.061E+00	1.0000E+01	2.5000E+01	8.8463E-01	8.9719E-01	3.4616E-03	3.4251E-03
9	1.625E+01	0.000E+00	5.618E+00	1.0000E+01	2.5000E+01	7.1419E-01	7.2468E-01	-1.3503E-02	-1.3636E-02
10	1.875E+01	0.000E+00	4.175E+00	1.0000E+01	2.5000E+01	4.8032E-01	4.8760E-01	-1.6023E-02	-1.6161E-02

The strips are arranged based on increasing y-coordinate of the strip in the aerodynamic coordinate system. The z coordinate and the x location of the point at which the pitching moment is computed (based on the MREF field of the MONCNCM entry) is given as are the local chord length and strip area. This is followed by the lift and pitching moment at the trim state for the rigid vehicle and for the elastically deforming vehicle.

Examples (moncncm and monbodi)

Two small test files demonstrate the use of the MONCNCM Bulk Data entry. They are both variations of the familiar forward swept wing and are:

moncncm provides tests of various ways of merging data across strips that may not be practical from an engineering standpoint but that exercise different code paths.

monbodi - demonstrates the application of MONCNCM when there are multiple interference groups and aerodynamic bodies. It is seen that different interference groups are not merged across aerodynamic panels and that there are no results given for CAERO2 entries.

Guidelines and Limitations

The MONCNCM entry only supports Doublet Lattice like models and only for lifting surfaces. Aerodynamic bodies are not supported. The output is in the aerodynamic panel coordinate systems so that the sign of the forces and moments are a function of the numbering of the corners of the panel (see the description of the CAERO1 entry in the *MSC Nastran Quick Reference Guide*). The results are given at the aeroelastic trim state.

Input of an Aerodynamic Mesh

Introduction

Four new bulk data entries have been provided that allow the user to define an aerodynamic mesh that is distinct from the traditional aerodynamic model that is input using a combination of CAEROi/PAEROi entries.

Benefits

The ability to input an aerodynamic mesh into MSC Nastran opens up various capabilities that are considered embryonic at this point. Among these are:

It enables the viewing of an external aero mesh on an MSC Nastran structural model using a tool such as Patran.

It is possible to spline results from the structural mesh to the new aerodynamic mesh and vice versa. The spline export feature described in section 1.6 below provides this spline for manipulation outside of MSC Nastran

The aerodynamic grid points can be loaded using the AEFORCE entry, thereby enabling the import of rigid aerodynamic loads from an external aerodynamics program.

The AESCALE entry described below opens up some interesting possibility for morphing models that have yet to be explored.

Input

The aerodynamic mesh can be input using four new bulk data entries

- The AEGRID provides coordinates of the mesh.
- The AEQUAD4 entry connects the mesh using a quad element
- The AETRIA3 connects the mesh using a triangular element
- The AESCALE entry allows for a scaling of the AEGRID data on a grid-by-grid basis

Note that the element data (AEQUAD4 and AETRIA3) are strictly for display and do not participate in the creation of any aerodynamic matrices.

Output

There is no new output.

Example (aegridf)

This simple example is a variation on the 15 degree swept wing model named ha144c.dat in the *MSC.Nastran User's Guide for Aeroelasticity*. Relative to the example in the guide, the aegridf.dat example:

- Removes the doublet lattice model
- Adds AEGRID quantities that coincide with all the structural grids in the model plus four more that represent an aerodynamic stabilizer
- Uses a SPLINE4 entry to spline the wing forces at the AEGRID's to the structure
- Uses a SPLINRB (see ?.4) to spline the AEGRID's on the stabilizer to the support point
- Removes the FORCE/GRAV artifice used to simulate constrained wind tunnel condition. Instead a 1g cruise condition is specified with the aircraft free to pitch and plunge.
- AEPARM entries specify ALPHA and STAB states.
- AEFORCE entries are used to set up load vectors for the intercept, unit angle of attack and unit stabilizer setting.

The stability derivative print from running the example includes:

```
N O N - D I M E N S I O N A L   S T A B I L I T Y   A N D   C O N T R O L   D E R I V A T I V E   C O E F F I C I E N T S

        CONFIGURATION = AEROSG2D     XY-SYMMETRY = ASYMMETRIC     XZ-SYMMETRY = SYMMETRIC
              MACH = 4.5000E-01                Q = 2.0000E+00
        CHORD = 2.0705E+00        SPAN = 1.1050E+01         AREA = 1.1440E+01


  CONTROLLER STATE: INTERCEPT ONLY, ALL CONTROLLERS ARE ZERO

  TRIM VARIABLE   COEFFICIENT          RIGID                 ELASTIC                  INERTIAL
                UNSPLINED      SPLINED     RESTRAINED    UNRESTRAINED    RESTRAINED    UNRESTRAINED

  REF. COEFF.    CX      0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
                 CY      0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
                 CZ      3.496504E-02   3.496504E-02   3.496504E-02   3.496504E-02   0.000000E+00   3.496504E-02
                 CMX     0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
                 CMY    -1.249997E-02  -1.249997E-02  -1.249997E-02  -1.249997E-02   0.000000E+00  -1.249997E-02
                 CMZ     0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00

  ALPHA          CX      0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
            CY      0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
            CZ      2.596154E+00   2.596154E+00   2.596154E+00   2.596154E+00   0.000000E+00   2.596154E+00
            CMX     0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
            CMY    -1.296420E+00  -1.296420E+00  -1.296420E+00  -1.296420E+00   0.000000E+00  -1.296420E+00
            CMZ     0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00

  STAB           CX      0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00
```

	CY	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
	CZ	7.517483E-01	7.517483E-01	7.517483E-01	7.517483E-01	0.000000E+00	7.517483E-01
	CMX	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
	CMY	-2.760425E+00	-2.760425E+00	-2.760425E+00	-2.760425E+00	0.000000E+00	-2.760425E+00
	CMZ	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
URDD3	CX	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
	CY	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
	CZ	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	1.793749E-03	0.000000E+00
	CMX	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00
	CMY	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	-6.412643E-04	0.000000E+00
	CMZ	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00	0.000000E+00

It is seen that the splined and unsplined rigid aerodynamic terms are identical, which means the splines are functioning well, and that the rigid and elastic results are the same, which reflects the fact that there is no aeroelastic feedback in this case.

Trim results from the run include:

```
        A E R O S T A T I C   D A T A   R E C O V E R Y   O U T P U T   T A B L E S
      CONFIGURATION = AEROSG2D     XY-SYMMETRY = ASYMMETRIC     XZ-SYMMETRY = SYMMETRIC
                MACH = 4.500000E-01              Q = 2.000000E+00
      CHORD = 2.0705E+00          SPAN = 1.1050E+01          AREA = 1.1440E+01

   TRIM ALGORITHM USED: LINEAR TRIM SOLUTION WITHOUT REDUNDANT CONTROL SURFACES.
                          AEROELASTIC TRIM VARIABLES

        ID    LABEL            TYPE        TRIM STATUS    VALUE OF UX

              INTERCEPT        RIGID BODY                 FIXED     1.000000E+00
        501   ALPHA            GENERAL CONTROL            FREE     -1.334841E-02  RADIANS
        502   STAB             GENERAL CONTROL            FREE      1.973043E-03  RADIANS
        503   URDD3            RIGID BODY                 FIXED     1.000000E+00  LOAD FACTOR
        505   URDD5            RIGID BODY                 FIXED     0.000000E+00  RAD/S/S PER G
```

It is seen that the very lightweight structure is in trim based on intercept aerodynamics, a negative angle of attack and a slightly positive stabilizer.

Guidelines and Limitations

The aerodynamic mesh cannot be used in conjunction with the existing CAEROi format for creating an aero mesh.

The CMPID listed on the aerodynamic elements can be reference on an AECOMP that is used in the aerodynamic splining. However, the splining is only done based on the AEGRID data that are provided on an AELIST entry identified with the AECOMP.

Unlike the CAEROi format, there is no central grid on an aerodynamic element. The only aerodynamic grids are at the element vertices.

There is no integration of user input pressures or downwashes, so the AEFORCE entry is the only means of loading the aerodynamic mesh and, again, this can only be done at the AEGRID locations.

AIC's are not supported for this type of mesh.

Rigid Body Spline

Introduction

A new spline technique has been introduced that enables the splining of aerodynamic data to exactly six structural degrees of freedom in a rigid body fashion.

Benefits

The primary application of the rigid body spline is to introduce aerodynamic loads into the structural analysis process when there is no structure underlying the aero mesh. For example, this enables introducing loads from the tail component in a wing design when only the wing has structural detail.

Input

The new rigid spline method is enabled via the SPLINRB entry.

Output

There is no new output.

Example (splinrb)

A very simple example is provided that is based on the ha144e test case of Section 7.5 of the *MSC.Nastran User's Guide for Aeroelastic Analysis.* The example in the User's Guide uses separate 1D splines (the SPLINE2 entry) to distribute the aerodynamic load on the beam structure. This example replaces these two splines with SPLINRB entries that spline all the loads to the support point. The effect of this is that these loads no longer create an aeroelastic affect and the rigid and splined stability derivatives are identical.

Guidelines and Limitations

In addition to the loads being transferred to the 6 structural degrees of freedom, it should also be noted that the motion of the aerodynamic surface corresponds to the rigid body motion defined by these 6 dofs.

Wendland Spline Functions for the Spline4/5

Introduction

Options have been added to the SPLINE4 and 5 bulk data entries to support selection of radial interpolation functions. Reference 1 contains the mathematical development of these functions while Reference 2 further develops them in the context of an aeroelastic application.

Benefits

The references indicate that these alternative splining techniques should provide smooth results with improved performance relative to the existing methods. As indicated in the Guidelines and Limitations portion of this subsection, MSC does not have enough experience with the new methods to quantify these benefits.

Input

The METH field for the SPLINE4 now supports RIS in addition to the existing IPS, TPS and FPS methods. If RIS is selected, the order of the interpolation method is either defaulted to WF2 or can be set to WF0. RCORE is used to define the radius of support. For the SPLINE5, METH is a new field, with BEAM (infinite beam spline) the default. If METH=RIS is specified, FTYPE (default=WF2) and RCORE also require input.

Example (ha144c_ris)

The ha144c example has been modified to change the spline from SPLINE1 to SPLINE4 with METH=RIS, FTYPE=WF2 and RCORE=5.0. Running the example produces answers that are equivalent but not identical to ha144c. The differences are comparable to those that results from replacing the infinite plate spline with a finite plate spline so that it is deemed these results are reasonable.

Guidelines and Limitations

MSC has had limited experience with the radial interpolation functions. They are being provided in a “Beta” state so that clients can exercise these new options. This is best done in conjunction with the Spline Verify option of MSC.Flightloads so that a visual check of the quality of the fit can be made.

Because the radial interpolation functions act over a limited area, specified using RCORE, it is believed that a single spline can be effectively applied to a large area. For example, with a high aspect ratio wing, a single spline may be able to suffice.

If the linear SPLINE5 includes the rotational flexibility (DTHX/DTHY > -1.0) from the structural deformations, the FTYPE =WF2 will be used regardless of the user input value of FTYPE.

Spline Blending

Introduction

A capability has been added to blend spline effects across multiple splines.

Benefits

Prior to this MSC Nastran 2007 release, it was an error if the same aerodynamic grid was invoked on multiple splines. This could result in discontinuities at the panel boundaries so this rule has been relaxed so that an aero grid can be selected on multiple splines and the results blended in some user specified fashion. This results in smoother displacement patterns which will be of particularly of benefit as the aeroelastic analysis includes more CFD aerodynamics.

Input

A parameter has been provided to indicate that blending is allowed to occur. The parameter is MPTSPLIN and is input using the MDLPRM Bulk Data entry. This parameter has the following meaning:

MPTSPLIN = 0 – Do not allow an aerodynamic grid to be referenced by more than one spline (Default)

= 1 – Allow an aerodynamic grid to be reference by multiple splines.

This parameter does not apply to aerodynamic grids defined with AEGRID discussed above. These grids can always be referenced on multiple splines.

Two new Bulk Data entries (SPBLND1 and SPBLND2) have been provided to support spline blending. If neither of these is used, the splines are averaged. The SPBLND1 performs strip based blending that can be either averaged (with a user defined weighting), linear or cubic in nature. The SPBLND2 performs a curve based blending based on user input curve.

Example (ha145e_blnd)

The existing ha145e test deck has a single SPLINE1 entry. In this example, this has been replaced by two splines that overlap the two center rows of boxes. MDLPRM,MPTSPLIN,1 is used to enable this overlap when using the CAERO1 generated aerodynamic meshes. A SPBLND1 entry is then used to blend the results in a linear way. If the SPBLND1 entry had not been used, the splines would have been averaged. For this simple example, the blending has little affect on the flutter results. The intent of the example is to demonstrated the new features.

Guidelines and Limitations

This is another area where MSC has limited experience so there is limited guidance. It can be said that the blending developments have been made in concert with the alternative aeromeshing of section 1.3 above and is therefore pointed at applications that use external aerodynamics. It is felt that the blending

will be most useful in areas like wing fuselage junctions were the standard splining techniques are likely to result in gaps in the aerodynamic deformations.

Export of the Spline Matrix

Introduction

A capability has been provided to export the spline matrix to either an .op2 file or to the punch file.

Benefits

Users can now easily obtain the spline matrix in a format that can be readily applied in another procedure or reintroduced into another Nastran run.

Input

The SPLINOUT case control command is used to produce the exported spline matrix. If the .op2 format is used, the user should also assign a file that will contain the matrix in OUTPUT2 format.

Output

If the punch output is requested, the GPGK spline matrix is written to the .pch file using the DMIG format. The DMIG print is preceded by a direct table input (DTI) data that provides a map of the matrix columns to the aerodynamic degrees of freedom. With the default .op2 method, an output2 file is written.

Examples (splinopch and splinoop2)

These two examples show the use of the .pch and .op2 options for exporting a spline mat

References:

Wendland, H., Piecewise Polynomial, Positive Definite and Compactly Supported Radial Functions of Minimal Degree, *Adv. Computational Mathematics, 4(1995),* 389-396.

Beckert, A. and Wendland, H,. Multivariate Interpolation for Fluid-Structure-Interaction Problems using Radial Basis Functions, *Aerospace Sci. Technology,* 00 (2001), pp 1-11.