NEi Software Presents

2013 Nastran User Conference

June 25 - 27 2013

COVINGTON, KY

NEi Nastran Linear Tips and Tricks

David Weinberg
NEi Software

NEi Software®
Design, Analysis, and Simulation

NEi Nastran Basic Tips and Tricks

Topics

- Selecting the Right System to Run On
- Element Type Options
- Improving Accuracy
- Assessing Accuracy
- Troubleshooting

Selecting the Right System to Run On

Computer Recommendations

Order of Importance when purchasing a computer:

1) RAM
2) CPU
3) Storage

Computer Recommendations

NEi Software®

Design, Analysis, and Simulation

RAM

- You must have sufficient memory to run the analysis, otherwise it will slow down drastically
- Use the task manager and check the memory usage of the Nastran.exe process to estimate memory usage
- Make sure you are running a 64-bit OS to take advantage of higher memory limits

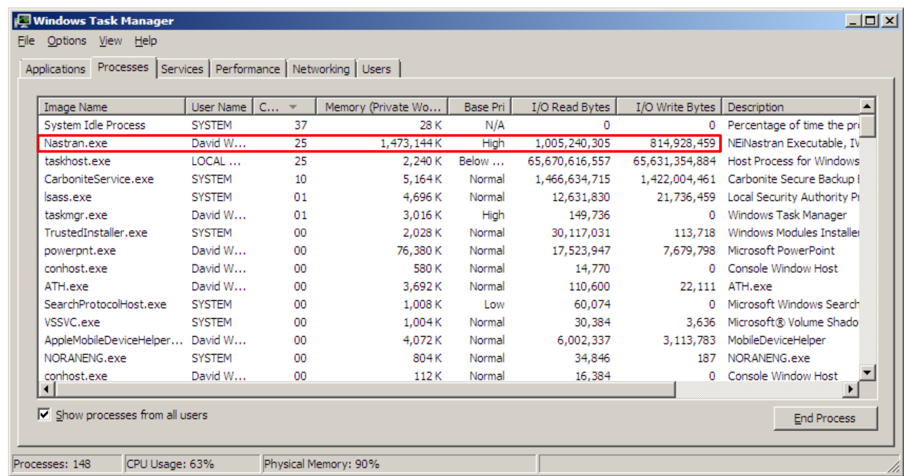


Image Name	User Name	C...	Memory (Private Wo...	Base Pri	I/O Read Bytes	I/O Write Bytes	Description
System Idle Process	SYSTEM	37	28 K	N/A	0	0	Percentage of time the pr
Nastran.exe	David W...	25	1,473,144 K	High	1,005,240,305	814,928,459	NEiNastran Executable, I\
taskhost.exe	LOCAL ...	25	2,240 K	Below ...	65,670,616,557	65,631,354,884	Host Process for Windows
CarboniteService.exe	SYSTEM	10	5,164 K	Normal	1,466,634,715	1,422,004,461	Carbonite Secure Backup I
lsass.exe	SYSTEM	01	4,696 K	Normal	12,631,830	21,736,459	Local Security Authority P
taskmgr.exe	David W...	01	3,016 K	High	149,736	0	Windows Task Manager
TrustedInstaller.exe	SYSTEM	00	2,028 K	Normal	30,117,031	113,718	Windows Modules Installe
powerpnt.exe	David W...	00	76,380 K	Normal	17,523,947	7,679,798	Microsoft PowerPoint
conhost.exe	David W...	00	580 K	Normal	14,770	0	Console Window Host
ATH.exe	David W...	00	3,692 K	Normal	110,600	22,111	ATH.exe
SearchProtocolHost.exe	SYSTEM	00	1,008 K	Low	60,074	0	Microsoft Windows Search
VSSVC.exe	SYSTEM	00	1,004 K	Normal	30,384	3,636	Microsoft® Volume Shado
AppleMobileDeviceHelper...	David W...	00	4,072 K	Normal	6,002,337	3,113,783	MobileDeviceHelper
NORANENG.exe	SYSTEM	00	804 K	Normal	34,846	187	NORANENG.exe
conhost.exe	David W...	00	112 K	Normal	16,384	0	Console Window Host

Computer Recommendations

NEi Software®
Design, Analysis, and Simulation

CPU

- The clock speed has a direct relationship with analysis time
- Get the Intel Xeon processors as they perform better at the same clock speed vs. the consumer i5/i7 processors

Computer Recommendations

NEi Software®

Design, Analysis, and Simulation

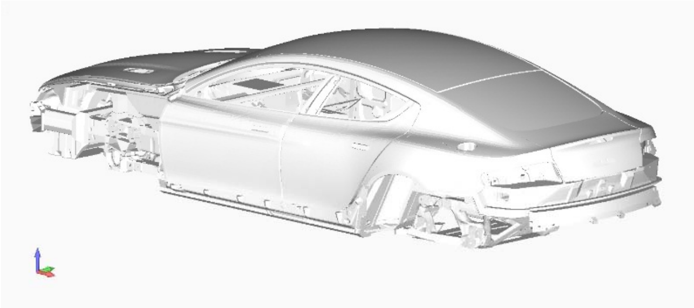
Storage

- Using a solid state disk (SSD) can significantly increase the performance of large models that use a lot of temporary files

CPU – Clock Speed vs. Cores

NEi Software®
Design, Analysis, and Simulation

- A highly scalable 1.3 Million DOF model was compared on 2 systems:
  - System 1 – Intel Xeon quad-core @ 3.4GHz
  - System 2 – Dual Intel Xeon 8-core @ 2.6GHz
- The goal is to find out if higher clock speeds are faster or if more cores are faster



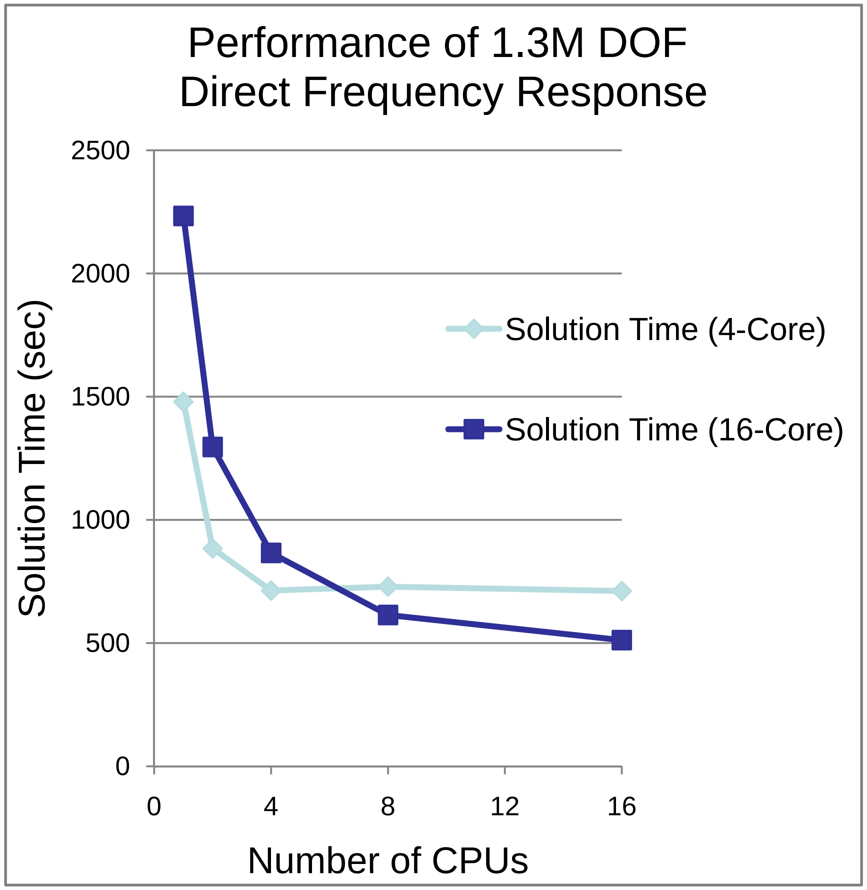
1.3 Million DOF Direct Frequency Response Model

Computer Recommendations CPU Clock Speed vs. Cores

NEi Software®
Design, Analysis, and Simulation

- Most models (such as linear static) won't see as much scaling so in general the higher clock speed CPU will be faster for typical models
- If you do significant direct frequency response analysis, the larger core count CPUs may be faster

# of CPUs	Solution Time (4-Core)	Solution Time (16-Core)	16-Core % Faster
1	1479	2234	-34%
2	884	1296	-32%
4	713	866	-18%
8	729	614	19%
16	711	512	39%

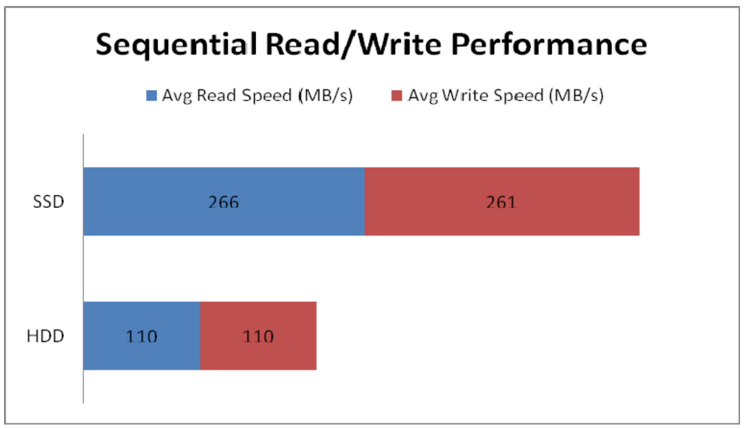


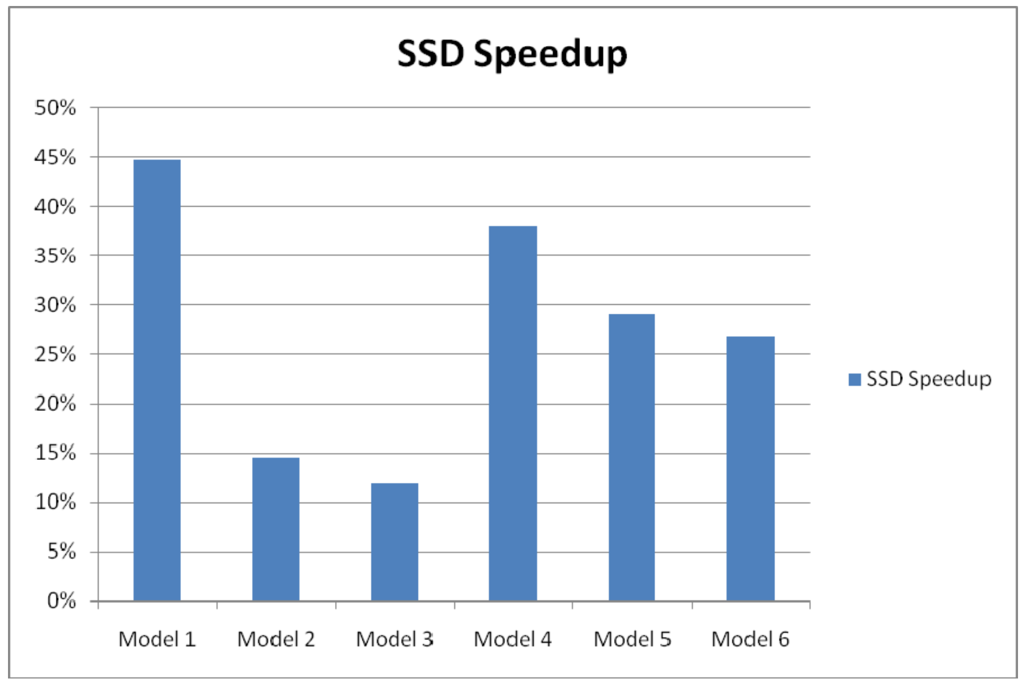
www.NEiSoftware.com

Computer Recommendations
SSD vs. HDD

NEi Software®
Design, Analysis, and Simulation

CPU: Intel Core i7 @ 2.8GHz
RAM: 8 GB
HDD: Western Digital Black 7,200RPM 1TB
SSD: Samsung 470 128GB





With the SSD, an average speedup of 27% was found for large models that use a lot of temporary files

Computer Recommendations Workstation

NEi Software

Design, Analysis, and Simulation

	Budget System	High-End System
CPU	Intel Core i5/i7 – Dual or Quad Core at high clock speed	Dual Intel Xeon Quad-core CPUs at high clock speed (3.5+GHz). or single Xeon 8-core CPU at high clock speed (2.8+GHz).
Memory, RAM	8 GB	32GB+
Graphics Card	1 GB dedicated video card	2+GB NVIDIA Quadro K4000 or similar
Hard Disk	1TB SATA 7200RPM and 256GB SSD for temp files	2TB 7200 SATA for file storage and 512GB SSD for temp files Note: High write speed is critical (at least 400MB/sec is good)
Operating System	Windows 7 Professional 64-bit or Windows 8 Professional 64-bit	

Computer Recommendations
Laptop

NEi Software®
Design, Analysis, and Simulation

	Budget Laptop	High-End Laptop
CPU	Intel Core i5 – Dual or Quad Core at high clock speed	Intel Core i7 – Dual or Quad Core at high clock speed
Memory, RAM	4 GB	16GB
Graphics Card	1 GB dedicated video card	2+GB NVIDIA or equivalent ATI
Hard Disk	1TB SATA 7200RPM	512GB or 1TB SSD. Note: High write speed is critical (at least 400MB/sec is good)
Operating System	Windows 7 Professional 64-bit or Windows 8 Professional 64-bit	

Element Type Options

Element Type Options

NEi Software
Design, Analysis, and Simulation

Parameter	Description	Type	Default
QUADELEMTYPE	Quad element bending formulation option. SRI – Selective Reduced-Order Integration. DKQ – Discrete Kirchhoff-Mindlin Quadrilateral. DKT – Discrete Kirchhoff-Mindlin Triangle (either two overlapping or four dissecting DKT elements depending on the setting for QUADINODE). The DKT and DKQ elements may be slightly more accurate than the SRI in very coarse meshes; however, the SRI element performs better in nonlinear and buckling solutions. All three element types handle finite transverse shear stiffness. The SRI and DKQ element types are supported in all solutions. The DKT element type is supported in linear solutions only. If QUADINODE is set to ON and the DKT element type is selected, the bending element will be comprised of four DKT subelements and a center node. If QUADINODE is set to OFF and the DKT element type is selected, the bending element will be comprised of two overlapping DKT sub elements.	SRI/DKQ/ DKT	SRI
TRIELEMTYPE	Tri element bending formulation option. DKT – Discrete Kirchhoff-Mindlin Triangle. SRI – Selective Reduced-Order Integration. The DKT element is typically more accurate than the SRI in coarse meshes and like the SRI element, works well for both thick and thin plates. Both element types handle finite transverse shear stiffness and are supported in all solutions.	DKT/SRI	DKT

Element Type Options

NEi Software
Design, Analysis, and Simulation

Parameter	Description	Type	Default
HEXINODE	Hex element internal node option. When set to ON, hex elements will produce more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting. The AUTO setting is recommended and provides optimal performance with accuracy.	ON/OFF AUTO	AUTO
QUADINODE	Quad element internal node option. When set to ON, quad elements will produce more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE, models with composite shell elements, or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting. The AUTO setting provides optimal performance with accuracy.	ON/OFF AUTO	AUTO
TETINODE	Tet element internal node option. When set to ON, parabolic tet elements will produce slightly more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting.	ON/OFF AUTO	OFF

Element Type Options

Parameter	Description	Type	Default
SHELLRNODE	Shell element drill degree of freedom option. When set to ON, CQUAD4 and CTRIA3 entries will be converted to CQUADR and CTRIAR entries, respectively.	ON/OFF	OFF

Improving Accuracy

Improving Accuracy

NEi Software

Design, Analysis, and Simulation

- SKINGEN Model Parameter
- ENHCBARRSLT Model Parameter
- ENHCQUADRSLT Model Parameter

The SKINGEN Model Parameter

NEi Software®
Design, Analysis, and Simulation

- What does it do?
  Generates nonstructural surface skin elements over the exterior of the solid element part.
- What are the options?
  DISABLE, SURFACE, HYBRIDX, HYBRIDM, and HYBRIDA, see Reference Manual for more information.
- Which options do I care about?
  Default which is DISABLE and the SURFACE option.
- When should I use this parameter?
  Anytime you have a solid model and are concerned about the coarseness of the mesh as it pertains to stress and strain accuracy.

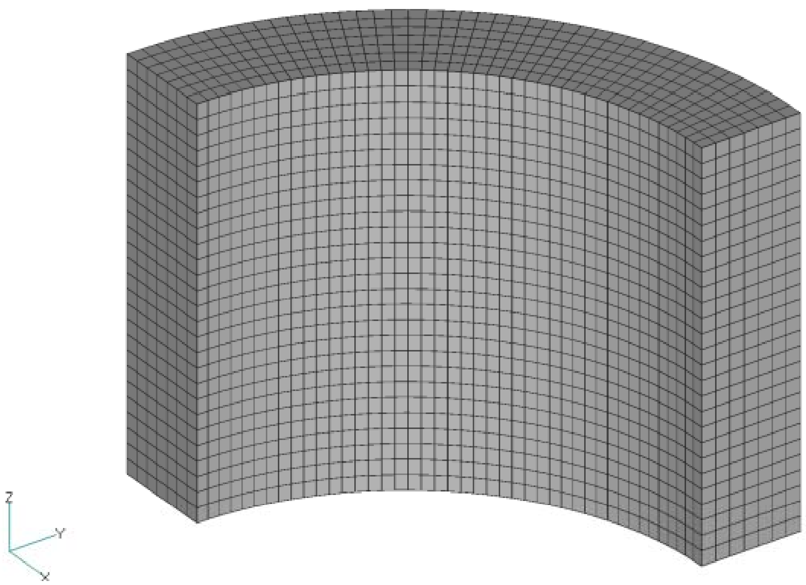
The SKINGEN Model Parameter

NEi Software®
Design, Analysis, and Simulation

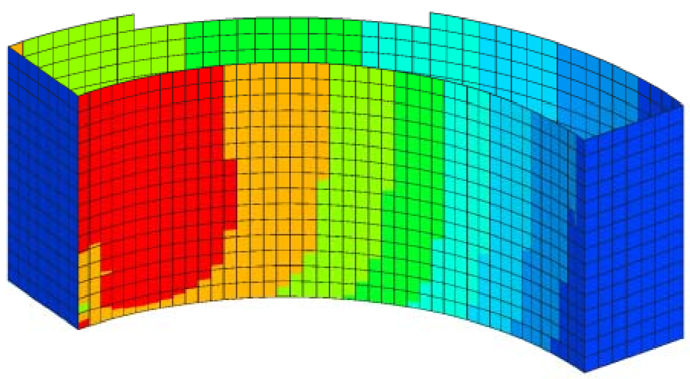
Automatic Surface Skin Generation (ASSG)

- Creates nonstructural membrane surface elements on all solid element types to enhance the accuracy of peak stress recovery

Solid Mesh



Membrane Mesh



www.NEiSoftware.com

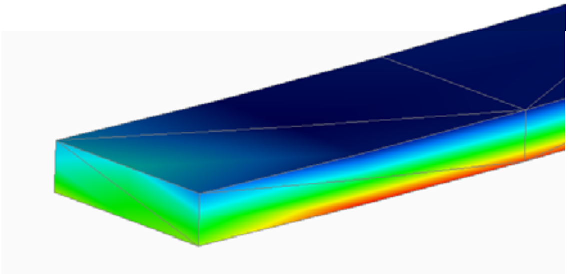
The SKINGEN Model Parameter

NEi Software®
Design, Analysis, and Simulation

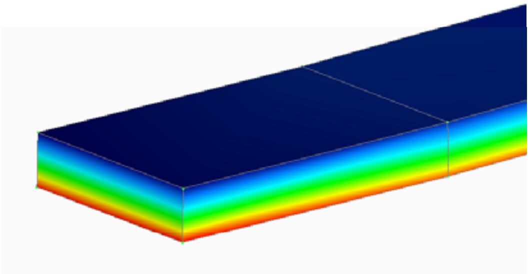
We look at 2 different meshes:

- A hex8 mesh and a tet10 mesh
- 1 element through the thickness
- Cantilever beam with a bending load

Tet10 mesh



Hex8 mesh



The SKINGEN Model Parameter

Theoretical solution

- Z-displacement

$$\delta = \frac{P\ell^3}{3EI} = 6.5 \text{ inches}$$

- Max bending stress

$$\sigma = \frac{Mc}{I} = 28.8 \text{ KSI}$$

The SKINGEN Model Parameter

Design, Analysis, and Simulation

Actual solution

- Z-displacement

Tet10 = 6.3 (2% error)

Hex8 = 6.4 (1% error)

The SKINGEN Model Parameter

NEi Software®
Design, Analysis, and Simulation

Actual solution – Element

- Max bending stress

Tet10 = 28.3/-28.3 (1% error)

Hex8 = 30.7/-30.7 (**10% error**)

Actual solution – Skin

- Max bending stress

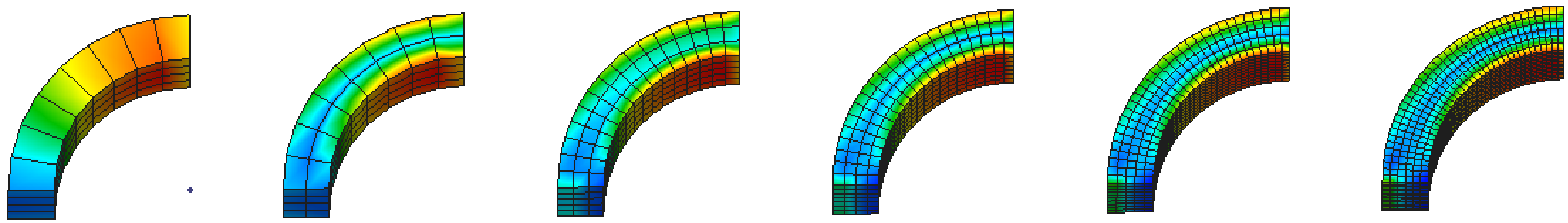
Tet10 = 27.9/-28.3 (1% error)

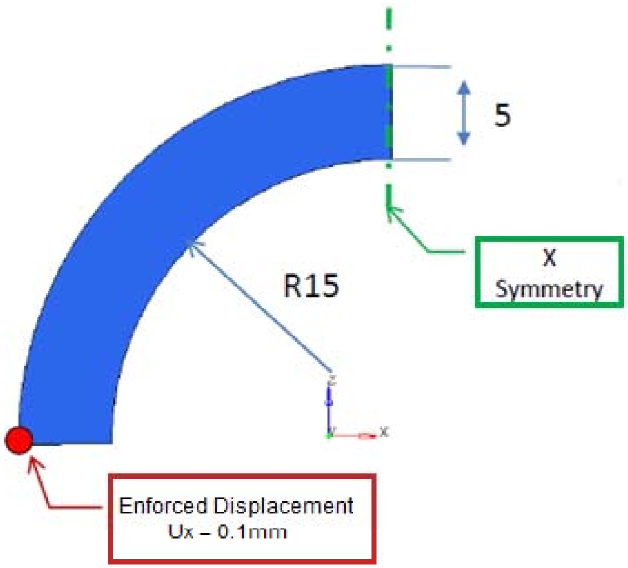
Hex8 = 27.5/-27.9 (2% error)

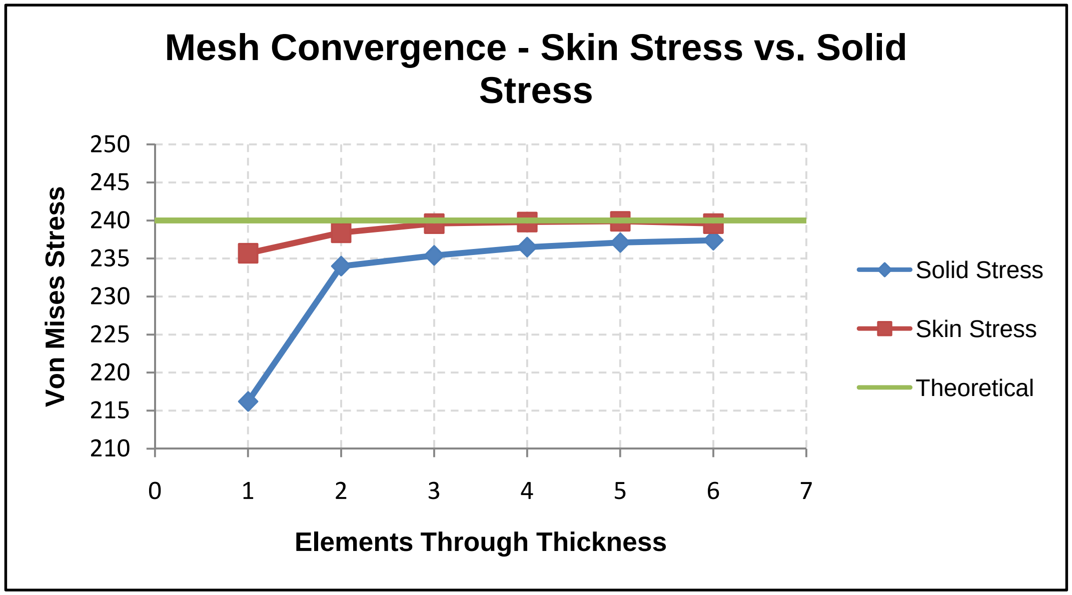
The SKINGEN Model Parameter

NEi Software
Design, Analysis, and Simulation

More accurate outer fiber stress





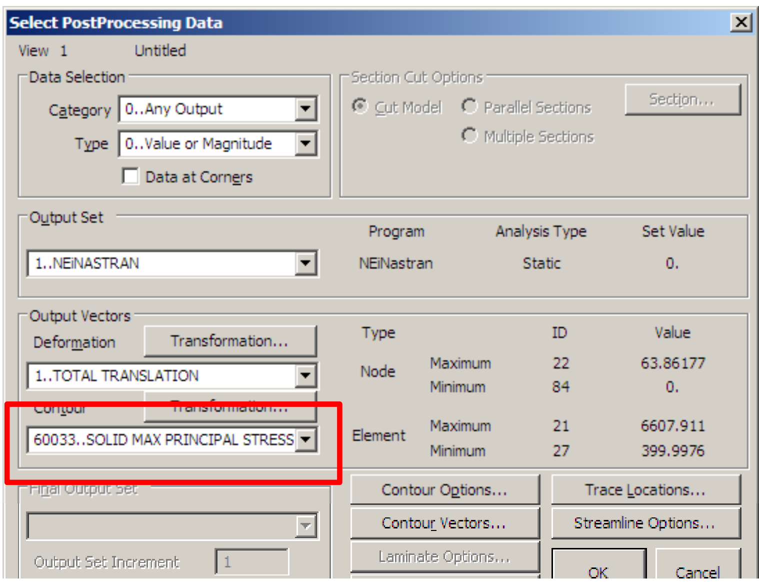


www.NEiSoftware.com

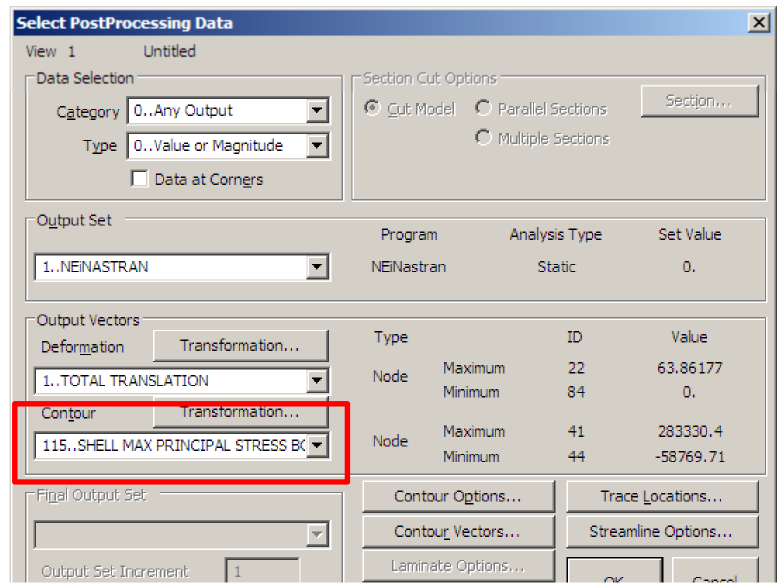
The SKINGEN Model Parameter

NEi Software®
Design, Analysis, and Simulation

Solid element stress



Shell skin element stress



The SKINGEN Model Parameter

NEi Software®

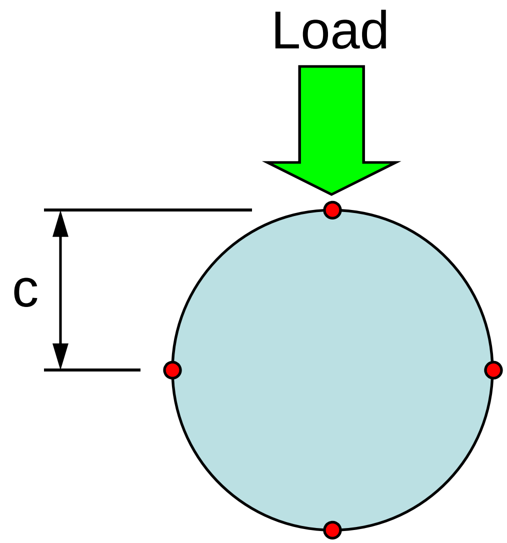
Design, Analysis, and Simulation

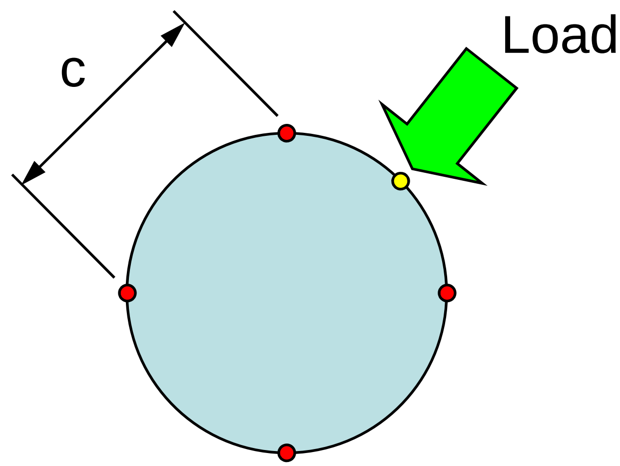
Parameter	Description	Type	Default
SKINGEN	Automated Surface Skin Generation (ASSG). Generates non-structural surface skin elements used in stress and fatigue analysis. A value between 0 and 4 defines the method used to calculate element corner results on a solid element mesh surface. There are five options: 0 – Automated surface skin generation is disabled. 1 – Surface skin elements and results are generated on the solid element mesh surface. No changes are made to the connected solid element corner results. 2 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are replaced with corresponding skin element values regardless of magnitude. 3 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are replaced with corresponding skin element values if the magnitude of the skin element component is larger. 4 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are averaged with corresponding skin element values. The character variables: DISABLE, SURFACE, HYBRIDX, HYBRIDM, and HYBRIDA may be used in place of the numerical options 0 through 4.	$0 \leq$ Integer ≤ 4 DISABLE/ SURFACE/ HYBRIDX/ HYBRIDM/ HYBRIDA	0

The ENHCBARRSLT Model Parameter

NEi Software
Design, Analysis, and Simulation

Previous Limitations of Bar Element Results





Circular bar with vertical loading

Standard stress recovery points

Stress = Mc/I

c = radius

When the load is rotated the c value changes:

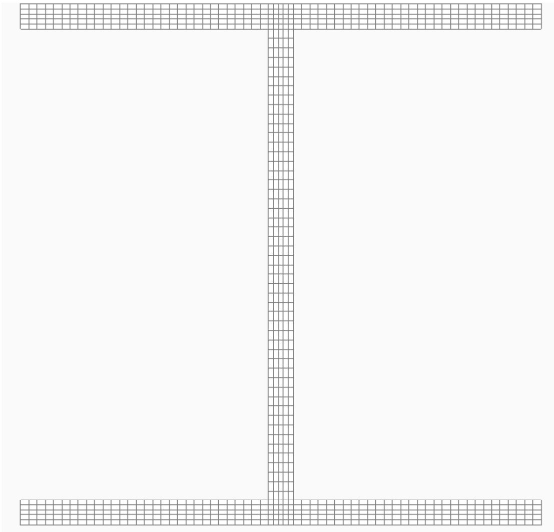
c = .7071*radius now

Stress is predicted lower

The ENHCBARRSLT Model Parameter

NEi Software®
Design, Analysis, and Simulation

- ENHCBARRSLT parameter addresses this Nastran limitation
- Small FEA model of bar is built automatically and run with loading from forces/moments
- Peak stress values extracted
- Shear stresses and torques are included
- Requires PBARL or PBEAML



www.NEiSoftware.com

The ENHCBARRSLT Model Parameter

NEi Software®
Design, Analysis, and Simulation

Advantages:

- No more stress recovery points needed
- Correct c value used
- Stresses now independent of recovery points
- Principal stresses reported which include full stress tensor

The ENHCBARRSLT Model Parameter

NEi Software®
Design, Analysis, and Simulation

New Output Vector Definition:

- SX-C = Maximum Stress
- SX-D = Minimum Stress
- SX-E = Maximum Stress Magnitude
- SX-F = Minimum Stress Magnitude (usually zero unless SX-D is positive)
- SX-MAX and SX-MIN are simply the max and min of the others

S T R E S S E S   I N   B A R   E L E M E N T S

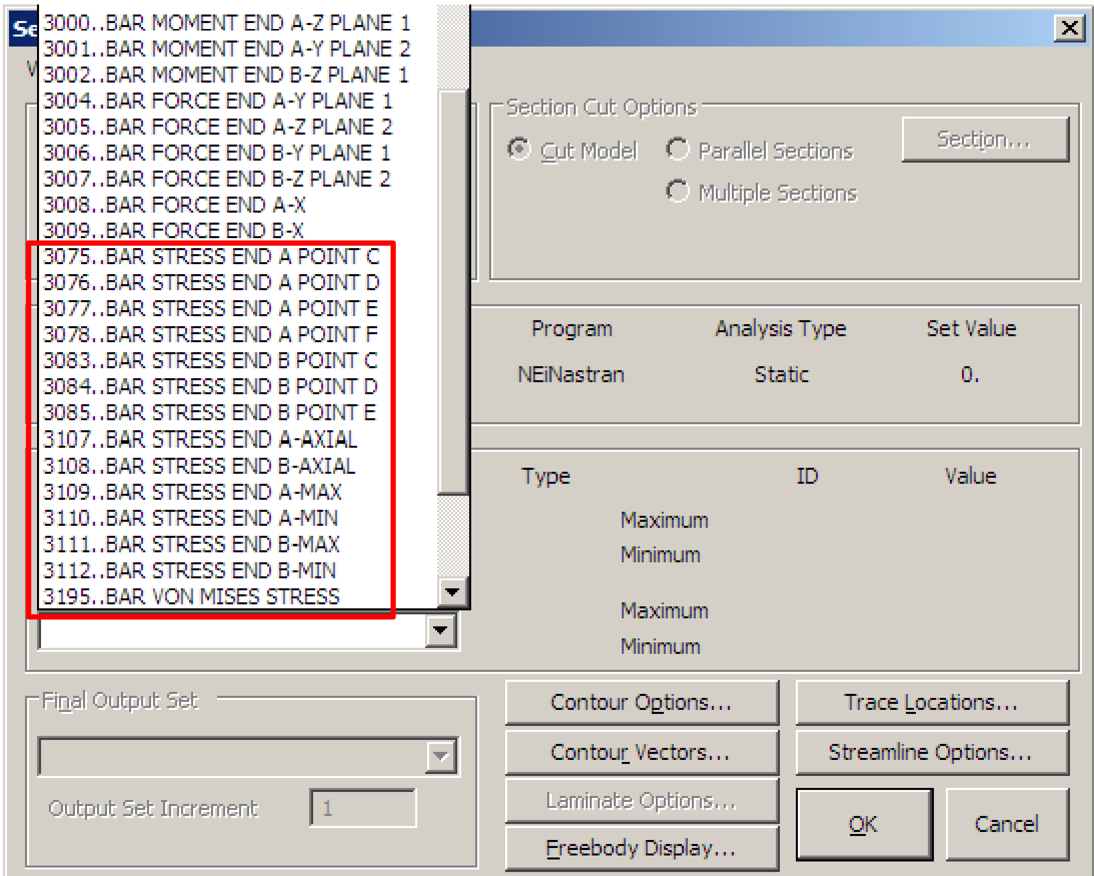
ELEMENT ID	DISTANCE	SX-C	SX-D	SX-E	SX-F	AXIAL	SX-MAX	SX-MIN
1	0.0000	1.273344E+04	-1.273344E+04	1.273344E+04	2.617484E-01	0.000000E+00	1.273344E+04	-1.273344E+04
	1.0000	4.632399E-12	-4.632399E-12	4.632399E-12	0.000000E+00	0.000000E+00	4.632399E-12	-4.632399E-12

```
MAXIMUM BAR ELEMENT TOTAL STRESS     =  1.273344E+04  AT ELEMENT 1
MINIMUM BAR ELEMENT TOTAL STRESS     = -1.273344E+04  AT ELEMENT 1
MAXIMUM BAR ELEMENT VON MISES STRESS =  1.273344E+04  AT ELEMENT 1
```

The ENHCBARRSLT Model Parameter

NEi Software®

Design, Analysis, and Simulation



www.NEiSoftware.com

The ENHCBARRSLT Model Parameter

NEi Software®

Design, Analysis, and Simulation

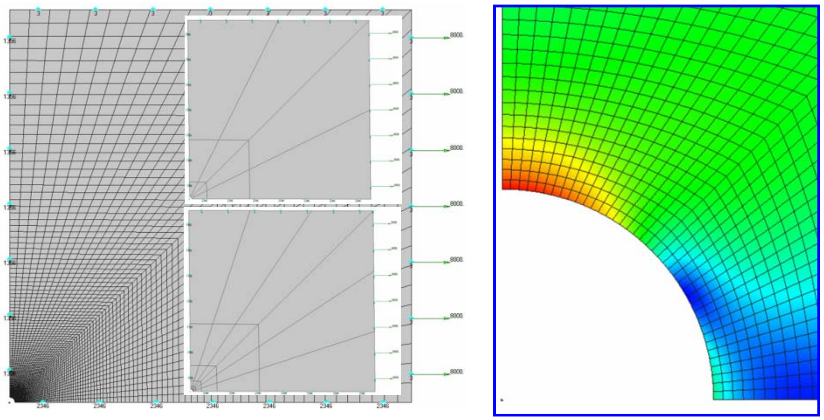
Parameter	Description	Type	Default
ENHCBARRSLT	Option for enhanced CBAR and CBEAM element results. When set to ON, an improved method for calculating CBAR and CBEAM element stress results is used when a corresponding PBARL and PBEAML property type is specified. Maximum direct and invariant stresses are determined using an automatically generated internal cross-sectional mesh at each element end. A separate finite element solution is performed on each mesh with direct and invariant results calculated at each mesh point and the maximum and minimum values reported.	ON/OFF	OFF

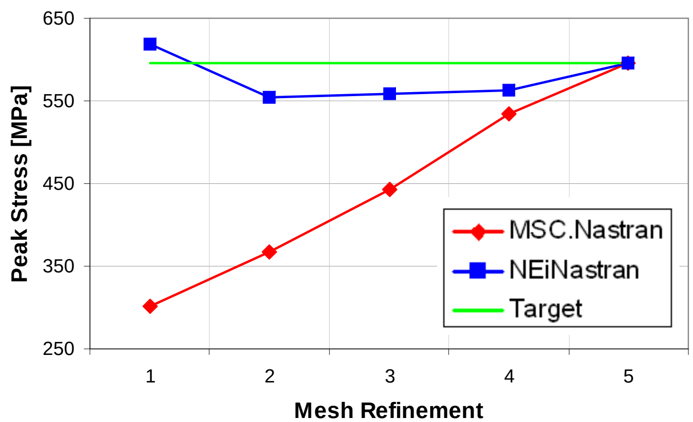
The ENHCQUADRSLT Model Parameter

NEi Software

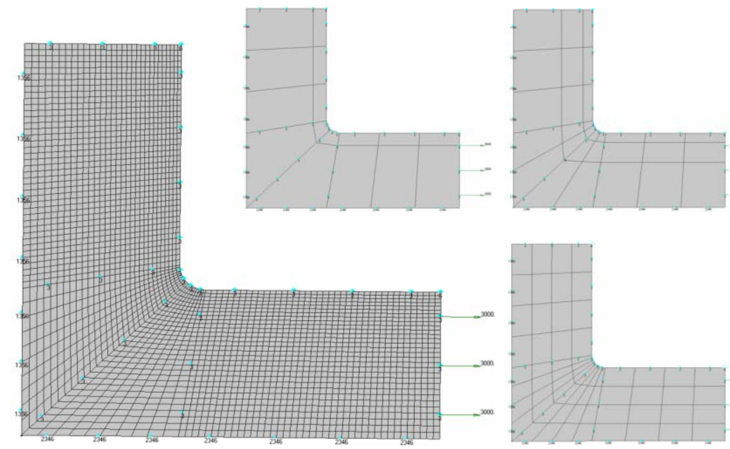
Design, Analysis, and Simulation

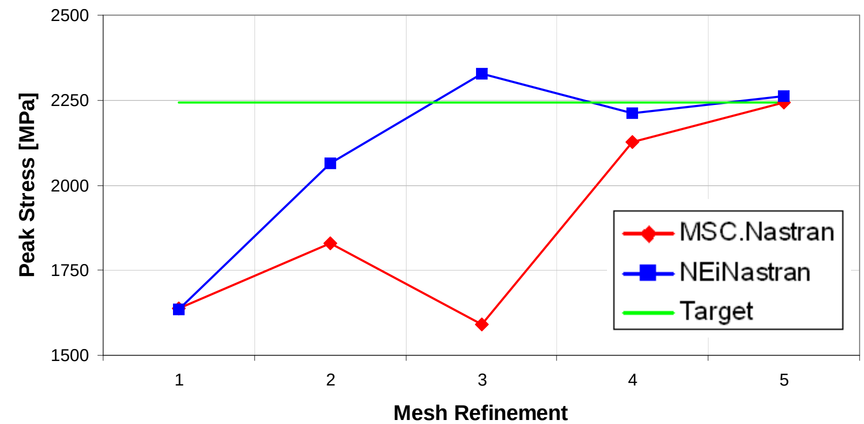
Plate with Hole





Filleted Plate





www.NEiSoftware.com

The ENHCQUADRSLT Model Parameter

NEi Software®

Design, Analysis, and Simulation

Parameter	Description	Type	Default
ENHCQUADRSLT	Option for enhanced CQUADR element results. When set to ON, an improved method for calculating CQUADR element stress results is used which gives better accuracy in regions with stress concentrations.	ON/OFF	OFF

NEi Software®
Design, Analysis, and Simulation

Assessing Accuracy

Results Quality Assessment

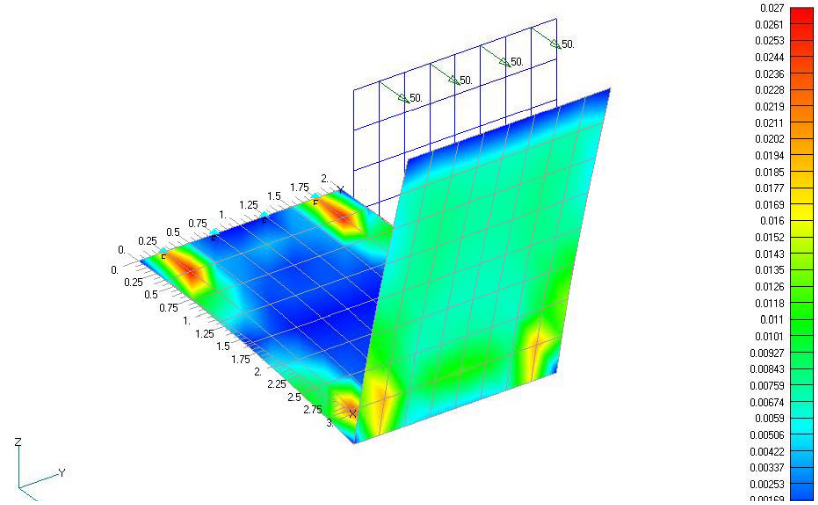
NEi Software®
Design, Analysis, and Simulation

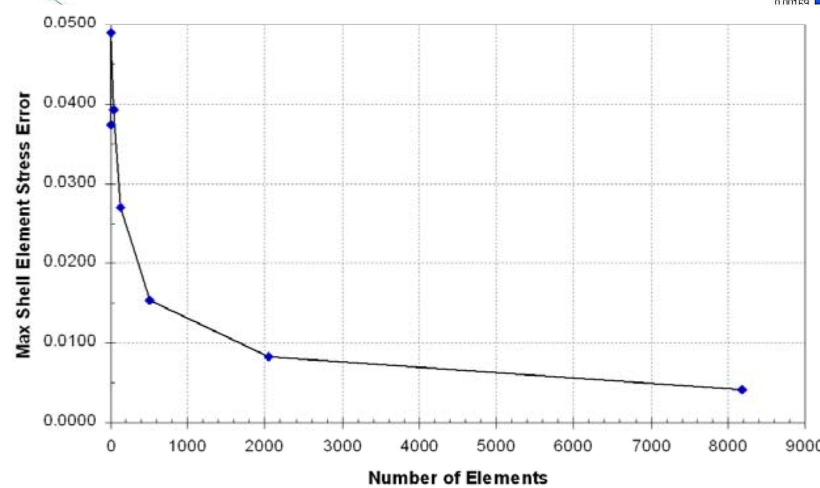
- Mesh Convergence Error
- Solution Error Measure
- Vector Resultant Balance

Mesh Convergence Error

NEi Software®
Design, Analysis, and Simulation

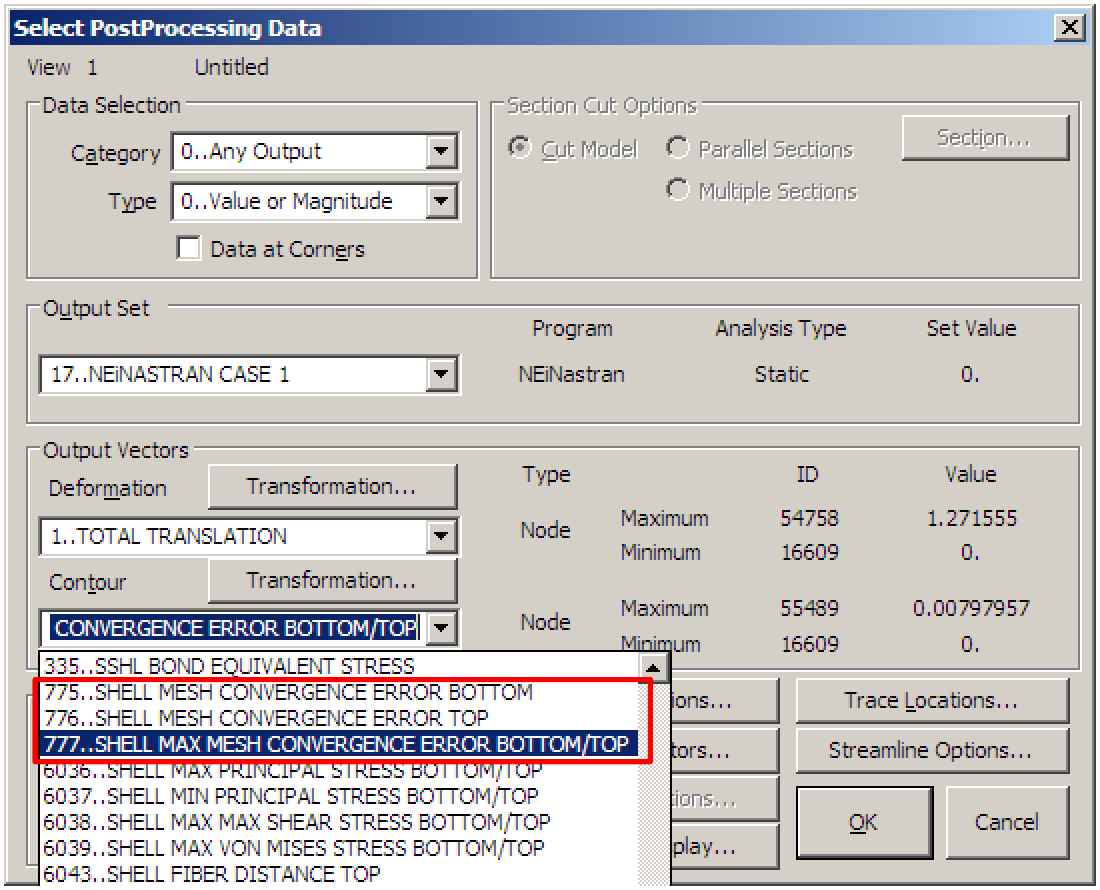
- Establishes an upper bound of the stress error associated with shell and solid elements mesh density
- Provides a normalized error based on von Mises stress at each solid or shell element grid point
- Useful in determining areas where mesh density should be increased or can be decreased
- Max/Min grid point and overall model shell and solid element errors output in all structural solutions





Mesh Convergence Error

NEi Software®
Design, Analysis, and Simulation



Mesh Convergence Error

NEi Software
Design, Analysis, and Simulation

Output from .LOG and .RSF Files

```
MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  8.116895E+03  AT ELEMENT 1
MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -8.116895E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  4.283409E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT VON MISES STRESS =  8.350951E+03  AT ELEMENT 6

MAXIMUM SHELL ELEMENT NORMALIZED STRESS ERROR =  7.650357E-02  AT GRID 6
MINIMUM SHELL ELEMENT NORMALIZED STRESS ERROR =  0.000000E+00  AT GRID 18

SHELL ELEMENT RELATIVE STRESS ERROR =  7.650357E-02
```

Mesh Convergence Error

NEi Software®
Design, Analysis, and Simulation

Parameter	Description	Type	Default
STRESSERROR	Controls the output of normalized grid point stress error (mesh convergence error). When set to ON, stress error at each grid point is calculated using $$e_i = \left[\frac{1}{N}\sum_{n=1}^{N}\left(\sigma_i^n - \bar{\sigma}_i\right)^2\right]^{\frac{1}{2}}$$ where N is the number of shell or solid elements attached to the node. σ_i is the von Mises stress predicted by element n at grid point i. $\bar{\sigma}_i$ is the mean von Mises stress at grid point i. The normalized error output is generated using e_i and a relative stress error based on element volume.	ON/OFF	ON

Solution Error Measure - Epsilon

- Measure of accuracy of the global displacement vector
- Values less than 10^{-6} are generally considered acceptable
- The DELTASTRAINEGOUT model parameter can be used to output residual strain energy at each grid point
- Causes for unacceptably large epsilon values discussed in next section

Solution Error Measure - Epsilon

NEi Software®
Design, Analysis, and Simulation

The residual strain energy vector is calculated using:

$$\delta E = (Ku - P)\,u$$

where u is the global displacement vector

P is the global load vector

K is the global stiffness matrix

The solution error measure, epsilon, is calculated using:

$$\varepsilon = \frac{\sum_{i=1}^{NDOF} \delta E}{u^T P}$$

Solution Error Measure - Epsilon

NEi Software

Design, Analysis, and Simulation

Output from .LOG and .RSF Files

```
MAXIMUM DISPLACEMENT MAGNITUDE =  4.340151E-02  AT GRID 2
MAXIMUM ROTATION MAGNITUDE     =  6.367040E-03  AT GRID 2

EPSILON        =  5.336379E-16
STRAIN ENERGY  =  2.170343E+01

SOLUTION TIME FOR 12 DEGREES OF FREEDOM = 0.0 SECONDS

MAXIMUM SINGLE POINT CONSTRAINT FORCE MAGNITUDE  =  1.005037E+03  AT GRID 1
MAXIMUM SINGLE POINT CONSTRAINT MOMENT MAGNITUDE =  1.000050E+04  AT GRID 1
```

Vector Resultant Balance

NEi Software
Design, Analysis, and Simulation

Output from .OUT File

L O A D   V E C T O R   R E S U L T A N T

SUBCASE ID	T1	T2	T3	R1	R2	R3
1	1.000000E+02	1.000000E+03	1.000000E+01	0.000000E+00	-1.000000E+02	1.000000E+04

S I N G L E   P O I N T   C O N S T R A I N T   V E C T O R   R E S U L T A N T

SUBCASE ID	T1	T2	T3	R1	R2	R3
1	-1.000000E+02	-1.000000E+03	-1.000000E+01	0.000000E+00	1.000000E+02	-1.000000E+04

Troubleshooting

Typical Issues

NEi Software®
Design, Analysis, and Simulation

- Bad epsilon (greater that 10^{-6})
- Bad eigenvalue error measures (greater than 10^{-2})
- Bad eigenvalue orthogonality losses (greater than 10^{-2})
- Singularity or non-positive definite errors
- Element geometry warning messages
- More or less than expected number of rigid body modes
- Excessive mesh convergence error
- No convergence in the maximum number of iterations permitted
- Applied load and SPC vector resultants are not equal and opposite
- Unrealistic results

Typical Issues

NEi Software®
Design, Analysis, and Simulation

Bad epsilon (greater that 10^{-6})

- Typically caused by a lack of constraint where the displacements are unrealistically high, often absurd
- An iterative solution was performed for the solution of displacements using the VIS or PCGLSS solver and convergence was not achieved or convergence tolerance too low (SPARSEITERTOL model parameter)
- There are unrealistically high or low values of stiffness (sometimes both) in the model
- Bad element geometry…use the ELEMGEOMOUT model parameter and check the .OUT file for a low or negative Jacobian

Typical Issues

NEi Software®
Design, Analysis, and Simulation

Bad epsilon (greater that 10^{-6}) - Continued

- Bad material properties…check the .OUT file material output
- Check other warning messages and refer to User’s Manual Error Messages section

Typical Issues

NEi Software
Design, Analysis, and Simulation

Bad eigenvalue error measures (greater than 10^{-2})

- Causes and actions are similar to actions taken with a bad epsilon value
- There are unrealistically high or low values of stiffness (sometimes both) in the model
- Unrealistically high values of mass
- Check other warning messages and refer to User's Manual Error Messages section

Typical Issues

Bad eigenvalue orthogonality losses (greater than 10^{-2})

- Causes and actions are similar to bad eigenvalue error measures

Typical Issues

NEi Software
Design, Analysis, and Simulation

Singularity or non-positive definite errors

- Typically due to a lack of constraint or an internal mechanism in a static solution
- Use the SOLUTIONERROR or NLKDIAGAFACT model parameters to help identify the source of the issue
- Also can use RBCHECKLEVEL and RBCHECKMODES model parameters to identify the source
- Negative stiffness can cause this so check the Grid Point Singularity Table and the .OUT file material output
- Shell element drill DOF issues can cause this so try the SHELLRNODE model parameter

Typical Issues

NEi Software®
Design, Analysis, and Simulation

More or less than expected number of rigid body modes

- More means there is a lack of constraint or internal mechanisms
- Less means there are internal constraints either due to AUTOSPC by the solver or bad MPC equations, invalid RBE3 elements, or non-collocated CELASi elements

Typical Issues

NEi Software

Design, Analysis, and Simulation

Element geometry warning messages

- If the distortion is extreme such as a quad element that resembles a tri element or a tet element that is almost completely flat it can cause other issues and generate inaccurate results
- The ELEMGEOMOUT model parameter will sort elements by worst to best making it easy to verify
- Can use ALIGNEDGENODE model parameter if the issue is related to EPLR or EPAD
- The AUTOFIXELEMGEOM model parameter will fix obvious issues like wrong node numbering

Typical Issues

NEi Software®
Design, Analysis, and Simulation

Excessive mesh convergence error

- Often corrected by increasing the mesh density in that area if it is critical
- Switching the element type (parabolic vs. linear) may help
- Can also force QUADINODE and HEXINODE model parameters to ON

Typical Issues

NEi Software®
Design, Analysis, and Simulation

No convergence in the maximum number of iterations permitted

- Typically this only happens if the user has limited the number of iterations as a performance consideration
- Increasing the maximum number of iterations (for the VIS and PCGLSS solvers use MAXSPARSEITER model parameter) will fix this

Typical Issues

NEi Software
Design, Analysis, and Simulation

Applied load and SPC vector resultants are not equal and opposite

- Often due to the PCGLSS convergence tolerance being too high
- Decreasing the convergence tolerance (use SPARSEITERTOL model parameter) will fix this
- Could also be due to bad RBE3 element definition, invalid MPC equations, or internal constraints due to bad materials, excessively high stiffnesses, non-collocated CELASi elements, etc.

Typical Issues

NEi Software®
Design, Analysis, and Simulation

Unrealistic results

- Check for other issues by first reviewing warning messages
- Verify that previously discussed issues are not the cause
- Use RBCHECKLEVEL model parameter and/or RBCHECKMODES
- Check the .OUT file material output
- Check Grid Point Weight Generator output
- Use PARMASSOUT model parameter to verify individual part mass and dimensions
- Review vector resultants in .OUT file

Useful Parameters for Troubleshooting

NEi Software®
Design, Analysis, and Simulation

Parameter	Description	Type	Default
AUTOFIXELEMGEOM	Option for automatically correcting elements that are singular due to an incorrect ordering of the element grid points.	ON/OFF	ON
AUTOFIXRIGIDELEM	When set to ON, will automatically correct improperly defined RBE3 elements by adding rotational degrees of freedom to averaging grid points as needed to prevent rigid body motion.	ON/OFF	ON
AUTOFIXRIGIDSPC	When set to ON, will automatically correct the following rigid element, interpolation element, and MPC equation issues by adding a near rigid spring at the dependent degrees of freedom: • A rigid element, interpolation element, or MPC equation dependent degree of freedom is constrained. • One or more rigid elements, interpolation elements, or MPC equations reference the same dependent degree of freedom. • A series of rigid elements, interpolation elements, and/or MPC equations are connected in a continuous link. • An RBE2 element is defined with the independent grid point located at the origin of a cylindrical coordinate system and rigidity is desired only in the R or T component direction. When AUTOFIXRIGIDSPC is set to OFF, behavior will be that of a rigid element defined in the Cartesian rectangular system which defined the specified cylindrical system. When AUTOFIXRIGIDSPC is set to ON and a translational or rotational component is missing, the local grid coordinate system at each independent grid point defines that dependent/independent segment. The spring element stiffness is defined by the KRIGIDELEM model parameter. See KRIGIDELEM below.	ON/OFF	OFF

Useful Parameters for Troubleshooting

NEi Software
Design, Analysis, and Simulation

Parameter	Description	Type	Default
ELEMGEOMOUT	Option to output individual element geometry statistics. When ELEMGEOMOUT is set to ON, the following statistics are output to the Model Results Output File for each element: • Aspect ratio • Taper ratio • Skew angle • Warping angle • Normalized Jacobian The data is sorted based on normalized Jacobian determinant, skew angle, and aspect ratio in ascending order for each element type. If ELEMGEOMOUT is set to ASPECTRATIO, then the sort will be in descending order and only based on element aspect ratio. If ELEMGEOMOUT is set to SKEWANGLE, then the sort will be in descending order and only based on element skew angle. If ELEMGEOMOUT is set to JACOBIAN1, then the sort will be in ascending order and only based on the total Jacobian determinant normalized using element volume. If ELEMGEOMOUT is set to JACOBIAN2, then the sort will be in ascending order and only based on the minimum Jacobian determinant at each corner node normalized using adjacent element edge lengths.	ON/OFF ASPECTRATIO/ SKEWANGLE/ JACOBIAN1/ JACOBIAN2	OFF

Useful Parameters for Troubleshooting

NEi Software®

Design, Analysis, and Simulation

Parameter	Description	Type	Default
PARTGEOMOUT	Individual part geometry statistics output option. When set to ON, additional part statistical information will be output including: • Material • Property type • Bounding box dimensions • Mass • Volume • Number of grid points • Number of elements	ON/OFF	OFF
PARTMASSOUT	Individual part mass properties output option. When set to ON, additional part mass properties information will be output including: • Material • Property type • Mass • Location of center of gravity • Mass moment of inertia	ON/OFF	OFF

Useful Parameters for Troubleshooting

NEi Software

Design, Analysis, and Simulation

Parameter	Description	Type	Default
NLKIAGAFACT	Specifies the stiffness to be added to diagonal terms of the global stiffness matrix. Specifying a small positive value is useful in stabilizing a solution and preventing a non-positive definite or singularity error. In nonlinear static solutions the added stiffness is decreased at the completion of each increment so to reach the value defined by NLKDIAGMINAFACT at the completion of the last increment. See also NLKDIAGCOMP and NLKDIAGMINAFACT.	Real	0.0
SOLUTIONERROR	When set to ON, it directs the program to substitute the value of FACTDIAG (default = 1.0E-10) for the factored diagonal term when a singularity or non-positive definite is detected. If FACTDIAG is set to zero, non-positive definites are ignored, while a singularity will result in program termination. SOLUTIONERROR and FACTDIAG are ignored in eigenvalue solutions and when the sparse iterative solvers (PCGLSS or VIS) are used. While this option is useful for modeling checkout, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that SOLUTIONERROR be set to OFF for production runs.	ON/OFF	OFF

Useful Parameters for Troubleshooting

NEi Software

Design, Analysis, and Simulation

Parameter	Description	Type	Default
RBCHECKLEVEL	Stiffness matrix equilibrium checks option. Equilibrium checks verify whether an unrestrained model can undergo simple rigid body motion without generating internal forces. There are six options: 0 – Do not perform any checks. 1 – Perform checks after stiffness matrix assembly before multipoint constraints are applied. 2 – Perform checks after multipoint constraints are applied before single point constraints are applied. 3 – Perform checks after single point constraints are applied before static condensation. 4 – Perform checks after static condensation before decomposition. 5 – Perform checks 1 – 4 above.	$0 \le$ Integer ≤ 5	0
RBCHECKMODES	Specifies the number of modes to solve for in an automated modal rigid body check. When set to a value greater than zero will perform an eigenvalue extraction analysis requesting that number of specified modes on the unconstrained model. Displacements and strain energy are output. Multipoint constraints requested in the first subcase of the model will be included.	Integer ≥ 0	0

NEi Software®
Design, Analysis, and Simulation

Questions