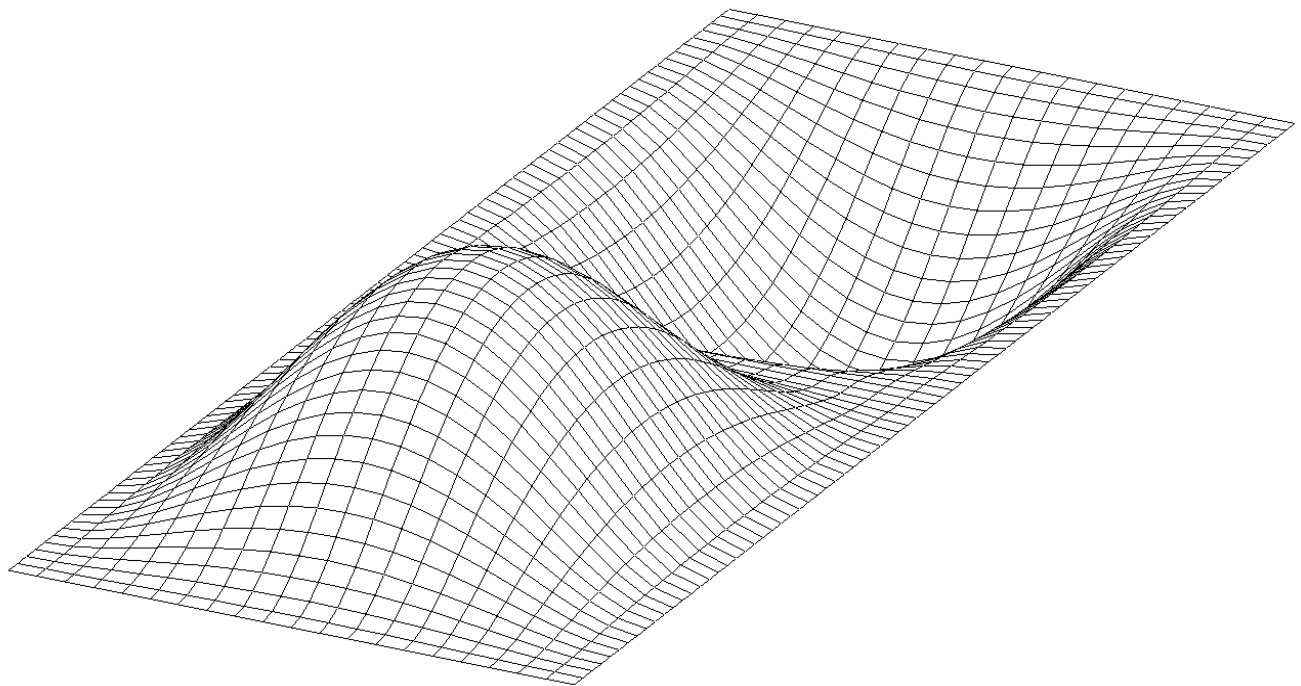


# NEiNastran Engine Version 9.1

## New Features and Enhancements



Information in this document is subject to change without notice and does not represent a commitment on the part of Noran Engineering, Inc. The software described in this document is furnished under a license agreement or nondisclosure agreement. The software may be used or copied only in accordance with the terms of the agreement. It is against the law to copy the software on any medium except as specifically allowed in the license or nondisclosure agreement. No part of this manual may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the express written permission of Noran Engineering, Inc. The material presented in this text is for illustrative and educational purposes only, and is not intended to be exhaustive or to apply to any particular engineering problem or design. Noran Engineering, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Copyright © Noran Engineering, Inc. 2007. All rights reserved.

Printed and bound in the United States of America.

**Noran Engineering, Inc.**  
5555 Garden Grove Blvd., Suite 300  
Westminster, CA 92683  
U.S.A.

Tel: (714) 899-1220

E-mail: [info@noraneng.com](mailto:info@noraneng.com)  
URL: [www.NENastran.com](http://www.NENastran.com)

**NE, NE/, NEi**, and the NEi logo are registered trademarks Noran Engineering, Inc. NASTRAN is a registered trademark of the National Aeronautics and Space Administration.

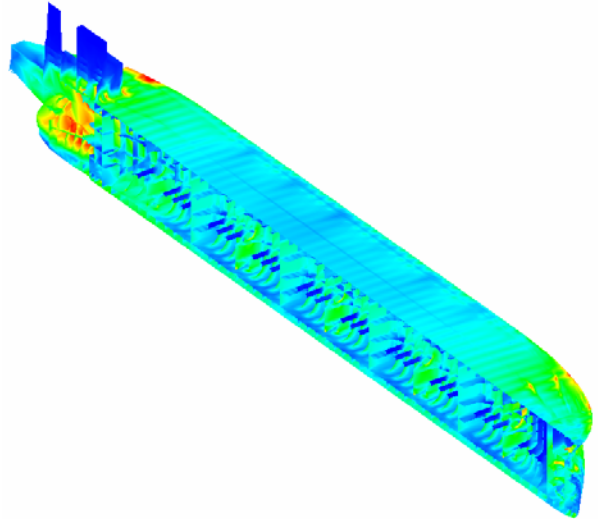
# TABLE OF CONTENTS

- Summary ..... 1
- Performance Enhancements ..... 2
- Linear Analysis Enhancements ..... 4
- Nonlinear Analysis Enhancements..... 5
- Dynamic Analysis Enhancements ..... 8
- Composite Analysis Enhancements..... 11
- Element Enhancements ..... 12
- Results Enhancements ..... 13
- Other Enhancements ..... 15
- Editor Enhancements ..... 17

## Summary

NEiNastran V9.1 is a major upgrade with over 80 customer driven enhancements.

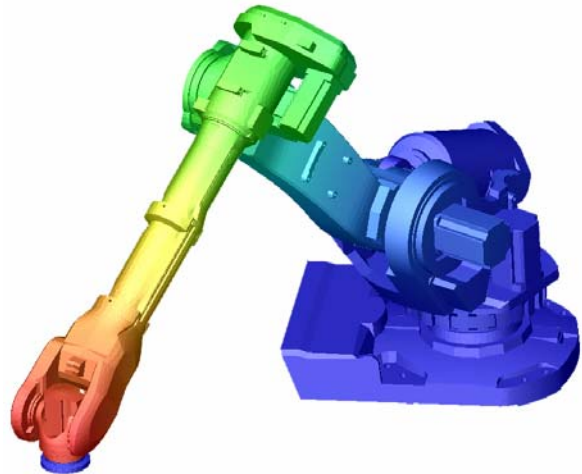
Major features include support for automated impact analysis, enhanced nonlinear and dynamic solutions, improved solver and overall performance, and new aerospace, maritime, and automotive productivity tools. The NEiEditor GUI has been modified to enhance the user interface usability. Also, several other NEiEditor enhancements have been included such as grid point stress and grid point strain display, capability to move through the same composite result measure ply by ply, and the ability to restore default settings in the Setup property page.



## Performance Enhancements

Many significant performance enhancements have been made affecting all solution sequences.

**Added support for a new Parallel Sparse Solver (PSS):** Added support for the PSS parallel linear equation solver in all solution sequences via the DECOMPMETHOD directive. This solver may be significantly faster than the existing VSS, VIS, or PCGLSS solvers. The PSS is supported on Windows x64 and Linux platforms currently with Windows IA-32 support to follow shortly. Like the PCGLSS, is a high-performance, memory-efficient equation solver capable of handling large sparse matrixes on shared-memory multiprocessors.



**Improved nonlinear transient and nonlinear static solution performance:** Improved nonlinear transient and nonlinear static solution performance as much as 20x. The most significant performance gain is with models which are mainly comprised of linear elements (shell and solid elements without material nonlinearity) and with large displacement effects turned off and models with velocity dependent damping elements.

**Improved PCGLSS solver performance:** Improved PCGLSS direct solver modes and Lanczos eigensolver performance. Additionally, increased the maximum number of Lanczos extraction modes to 7000.

**Improved transient and frequency response solution performance:** The following areas affecting performance in transient and frequency response analysis have been improved:

- Rod, pipe, bar, and beam element processing.
- Load vector assembly processing reducing total analysis time as much as 10x in cases with large numbers of TLOADi and RLOADi Bulk Data entries.
- Large models with large numbers of output steps (greater than 500) reducing total analysis time as much as 5x.

**Improved eigenvalue extraction performance:** Improved eigenvalue extraction convergence in nonlinear prestress buckling solutions when differential stiffness is requested (PARAM, LGDISP, ON).

**Improved performance for random response solutions:** Improved performance for random response solutions that:

- Referenced multiple uncorrelated inputs.
- Requested vector output at discrete points which included forces of single and/or multipoint constraint.

**Increased performance by loading parts of the model database into memory:**

Made significant performance improvements by loading additional sections of the model database into RAM. Also, modified PARAM, DATABASEACCEL options to include an AUTO setting. Now, when DATABASEACCEL is set to ON, the model database will be loaded into memory regardless of available RAM. When set to AUTO, RAM availability is checked for files that could use large memory blocks and only if sufficient RAM is available, will load into memory.

**Reduced processing time for element strain energy results:** Reduced processing time for element strain energy results when grid point force balance results were not also requested.

**Improved Nastran Binary and XDB file generation performance and compatibility:**

Better support is now provided for both the Nastran Binary (.OP2) and .XDB results neutral files. Support is now provided for most element types and solution sequences. Additionally, better translation performance for large models with a large number of output steps is also provided.

**Reduced subcase combination processing time:** Reduced subcase combination processing time (SUBCOM).

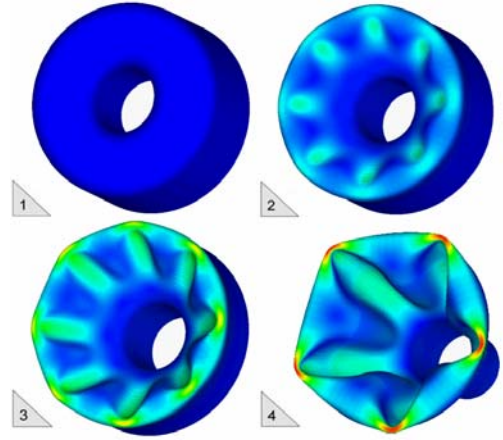
## Linear Analysis Enhancements

**Added linear surface contact analysis for linear static solutions:** Added linear surface contact analysis for linear static solutions via the LINEARCONTACT model parameter. Linear contact is supported in linear static analysis using surface contact. Convergence takes typically 2 – 3 iterations resulting in very fast analysis times on the order of a 3 subcase linear static analysis. PARAM, MAXLNCONTACTITER is provided to specify the maximum number of contact iterations and PARAM, LNCONTACTITERTOL to specify convergence tolerance.

## Nonlinear Analysis Enhancements

Several significant improvements for nonlinear analysis have been made enhancing both capability and performance.

**Added support for large strain hyperelastic materials:** Added support for large strain hyperelastic materials via the PLSOLID and MATHP Bulk Data entries. Support is provided for the Neo-Hookean, Yeoh, Mooney-Rivlin, and generalized polynomial form of up to order 5, applicable to nearly incompressible elastomers. Additionally, the Ogden generalized polynomial form of up to order 3 is supported.



**Added Automated Impact Analysis (AIA):** Added Automated Impact Analysis via the IMPACTGENERATE Case Control command.

This is a sophisticated tool which now allows the user to define a remote projectile or impactor and a target body without any preprocessor setup. Input consists simply of the direction of travel, initial velocity, and acceleration. The effective directional natural frequencies of the impactor and target in the contact state are calculated internally without any user setup. The critical time step calculations are then automatically carried out based on these responses providing a precise initial time increment and duration of the analysis. Accurate time step prediction is essential in calculating the magnitude of peak response and maintaining an energy balance during the contact event. Optional user defined time increment and duration may be specified. Additional control of output is provided via the MAXIMPACTSTEP model parameter. The internally calculated initial time increment will provide very high accuracy, however the total number of output time steps may be very large for a long duration, soft impact analysis. MAXIMPACTSTEP applies an override to limit the amount of response data so that the user can reevaluate the analysis feasibility.

**Added user definable restart capability for nonlinear static analysis:** This is a very significant productivity improvement for nonlinear users. A unique nonlinear database has been created to allow very efficient storage and retrieval of data during a nonlinear analysis. Restart capability for nonlinear static analysis is provided via PARAM, NLINDATABASE and the NLINDATFILE directive. Multiple restart database files may be generated at user defined load increments which can then be used as restart points with different loading, boundary conditions, model parameters, and material properties. One application could involve nonlinear buckling (SOL 180) where the user can predict the load region where instability is expected and confirm it with a nonlinear buckling analysis.

**Improved stability, performance, and accuracy of welded contact:** Improved stability, performance, and accuracy of welded contact in nonlinear solutions.

**Improved surface contact weld element results:** Improved surface contact weld element results for cases with large shear loads on stiff parts. Model with surface contact welds now run faster and give more accurate results.

**Added a new offset weld for surface contact:** Added a new weld form for surface contact which allows large gaps between contact surfaces while still providing accurate results and fast convergence. The new weld form permits any size gap between parts without introducing internal constraints.

**Enhanced Automated Surface Contact Generation (ASCG) for parts with large gaps:** Enhanced Automated Surface Contact Generation when contact is between parts with large gaps. Now gaps caused by midsurface meshing, joint slop, or bush tolerances can all be handled including weld elements using the new offset weld element capability.

**Added a new parameter to convert surface contact weld elements:** Added PARAM, SLINEOFFSETTOL which specifies the tolerance for converting surface weld elements to offset weld elements.

**Added a new parameter to specify a stabilization stiffness between contact surfaces:** Added PARAM, SLINESTABKSFAC which when set to a value greater than zero will add a stabilization stiffness between contact surfaces. The default zero value disables this feature. A value of 1.0 will add a stiffness approximately equal to the closed gap stiffness value. This feature is useful in stabilizing contact with a large separation between contact surfaces.

**Added a new parameter for surface contact nodal stresses calculation:** Added PARAM, SLINESTRESSLOC which specifies the location where surface contact nodal stresses are calculated: SLAVE, MASTER, or BOTH.

**Improved automated surface contact generation:** Modified automated surface contact generation to skip the generation of contact surfaces that have excessive initial slave node penetration, thus avoiding an E5072 warning.

**Added support for grid point results at points with surface contact:** Added support for grid point results at points where surface contact results were also requested. Previously, grid point results were skipped at surface contact grid points.

**Improved contact forces and stresses:** Improved accuracy of surface contact forces and stresses for coarse meshes and unsymmetric contact.

**Added new Automated Surface Contact Generation statistical information:** Added Automated Surface Contact Generation statistical information including contact area generated.

**Added a new weld form for surface contact:** Added a new weld form for surface contact which uses multipoint constraints (RBE3 elements) to connect contact surfaces.

**Added a new parameter for repositioning contact elements:** Added PARAM, NCONTACTGEOMITER which specifies the number of iterations for repositioning contact elements with initial penetration and/or protrusion.

**Added a new parameter for reducing solution time for nonlinear surface contact:** Added PARAM, SLINEMAXACTRATIO which when set to a value greater than zero, specifies the ratio of activation distance to contact surface maximum edge length. This parameter may be useful in reducing solution time for nonlinear surface contact models by deactivating contact segments far from area of active contact.

**New options for PARAM, CONTACTGEN:** Added GENERAL, WELD, SLIDE, ROUGH, and OFFSET options for PARAM, CONTACTGEN. Integer values may still be used.

**Added a new parameter to control the nonlinear differential stiffness update strategy:** Added PARAM, NITERKSUPDATE which controls the nonlinear differential stiffness update strategy.

**Added a new parameter which specifies the number of modes to be extracted during the automated impact analysis:** Added PARAM, EMODES which specifies the number of modes to be extracted during the initialization phase of automated impact analysis. A normal modes analysis is performed to determine the damping frequency of interest and the time step size.

**Added more control on line elements used in material nonlinear analysis:** Removed PARAM, NLMATSFAC control on line elements used in material nonlinear analysis. Line elements with stress-strain curves that have zero or negative slopes will now default to 1.0E-10 times the elastic value when a slope less than this value is encountered.

**Added large displacement and rotation effects for CBUSH and CBUSH1D elements:** Added large displacement/rotation capability for CBUSH and CBUSH1D elements with non-coincident grid points and with an element coordinate system defined by end grid points.

**Improved eigenvalue extraction convergence in nonlinear prestress buckling solutions:** Improved eigenvalue extraction convergence in nonlinear prestress buckling solutions when differential stiffness was requested (PARAM, LGDISP, ON).

**Added a new parameter to output nonlinear true stress:** Added PARAM, NLTRUESTRESS which when set to ON will output nonlinear true stress which accounts for change in element shape due to deformation.

## Dynamic Analysis Enhancements

Several important advanced features have been added to the powerful set of dynamic analysis solution methods. Additionally, many performance enhancements have been made.

**Added Bulk Data entries for shock and vibration mount support:** Added TABLEVF and PMOUNT Bulk Data entries for shock and vibration mount support. This allows a fully coupled description of a shock mount response from test or theoretical series equations to be defined. The velocity dependent stiffness terms can be included.

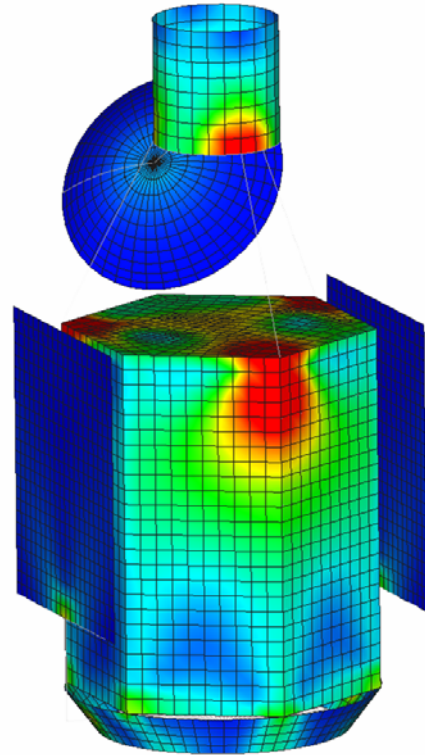
**Added mass and stiffness DMIG support for the PCGLSS solver:** The PCGLSS linear equation solver (DECOMPMETHOD=PCGLSS) and Lanczos eigenvalue extraction solver (EXTRACTMETHOD=LANCZOS) now support DMIG input.

**Increase the hard limit for nonlinear transient output:** Increased the maximum number of nonlinear time steps in nonlinear transient solutions from 100,000 to 1,000,000.

**Added single point constraints forces to the Modal Database for modal restart:** Added PARAM, MODEFSPCSTORE which when set to ON will store single point constraint forces in the modal database file for modal restarts. When set to OFF and a modal database restart is performed, single point constraint forces will be calculated, if requested, using the subcase 1 SPC set.

**Better support for the SET command in transient and frequency response solutions:** Improved the use of SET commands with real numbers for use in controlling transient and frequency response output. Previously set values were truncated to integers. This enhancement is particularly useful when using response data from external codes or test data.

**Added a parameter for identifying real values in output set lists:** Added PARAM, OUTSETTOL which specifies the tolerance for identifying real values in output set lists. A real value is considered as included in a set if the ratio of the difference between the set value and input value or the input value is less than OUTSETTOL.



**Added EXCLUDE and INCLUDE options to the MODESET Case Control command:** Primarily for use in DDAM analysis, but with applications in all response analysis, this feature allows a user to quickly isolate a few important modes from a large number of modes and test the response. Combined with the Modal Database restart feature, it becomes a fast and powerful investigation tool. Added an EXCLUDE option to the MODESET Case Control command. EXCLUDE allows the specification of a SET of modes that are to be excluded from the extracted set. Also, added an INCLUDE option which is functionally the same as SET.

**Added support for multiple MODESET Case Control commands:** MODESET now allows the use of multiple options to include and exclude modes. This productivity improvement allows great versatility in choosing target modes to investigate in modal response solutions.

**Modified a parameter which will disable frequency response neutral file vector for random response solutions:** Modified PARAM, FREQRESPRSLTOUT which now when set to OFF will disable frequency response neutral file vector and element output for random response solutions. Previously only element output was disabled. This represents a significant improvement in performance for cases where the frequency response data is not needed, only the PSD, RMS and NPX data. This is typical in practice where the user has already validated the frequency response analysis and is now carrying out the random response analysis.

**Modified a parameter which will disable random response PSD neutral file vector for random response solutions:** Modified PARAM, RANDRESPRSLTOUT which now when set to OFF will disable random response PSD neutral file vector and element output for random response solutions. Previously only element output was disabled. This represents a significant improvement in performance for cases where the PSD response for each DOF is not required for X-Y plotting, but contour mapping of RMS and NPX data is required. This is typical in practice where the user has already validated PSD response at key degrees of freedom and now wants to assess the complete structure response.

**Added a directive for enabling DMIG support for the PCGLSS solver and LANCZOS eigensolver:** Added the PCGLSSDMI directive which when set to ON enables DMIG support for the PCGLSS solver and LANCZOS eigensolver.

DDAM analysis continues to be an important tool for the maritime industry especially companies under contract with the US Navy. Significant enhancements have been put in place to improve productivity and accuracy.

**Added multi-phase DDAM operation capability:** A multi-phase DDAM operation capability has been added to improve productivity in a mixed secured and unsecured environment. The integrity of the DDAM coefficients is essential and this feature preserves this, while allowing the user more flexibility. A new parameter controls which phase of a DDAM analysis is to be run. This allows an initial (phase 1) DDAM checkout run to be carried out in an unsecured environment; the run can then be restarted in a secure environment using the Modal database (phase 2). A new DDAM database has been created which allows storage of the shock input G versus frequency spectra calculated from the DDAM coefficients. This is the only file, which needs to be exported from the secure environment, and is easily verifiable for content. The final phase 3 is then to calculate the DDAM responses in the unsecured environment.

**Modified DDAM response calculation:** Modified response calculation to include an additional requirement that all modes above 1 percent modal effective mass are included. Prior to this it was possible, but very unlikely, that the 80 percent modal effective mass cutoff would exclude a mode with greater than a 1 percent contribution.

**Modified the method used for DDAM displacement, velocity, acceleration, and single point constraint vector results calculations:** The previous method performed an NRL summation using as the maximum response mode, the entire modal vector having the largest response based on modal participation factor scaling. The new method looks at each degree of freedom to determine the maximum response mode and uses that value as the maximum for the NRL summation. This is a more conservative approach with strict compliance with NAVSEA 3010. The stress responses are calculated independently and have not changed.

**Enhanced DDAM von Mises output stress in bars, beams and shells:** As requested by the DDAM community, a new model parameter, EQVSTRESSTYPE now controls the von Mises stress calculation. When set to 2 in linear solutions this will output membrane only von Mises stress in bar, beam, and shell elements. The default setting of 0 will output membrane and bending von Mises stress. The beam and bar von Mises calculation has been completely updated to include both transverse shear terms and the torsional shear stress term. The effective section shear area is used to calculate the former. The latter uses either a user supplied effective torsional radius, or the appropriate dimension if the user inputs a section via a PBEAML or PBARL.

## Composite Analysis Enhancements

Composite analysis is a key area for many customers particularly in the aerospace industry. Several important enhancements have been requested from customers in this area.

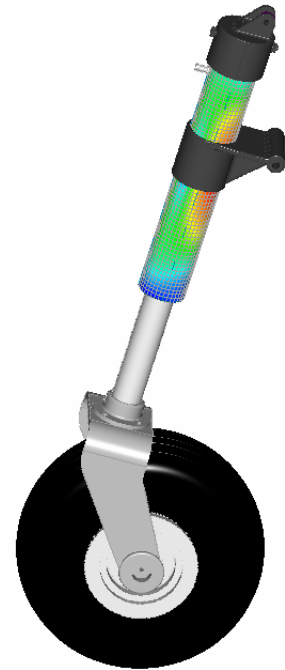
**Added 2D orthotropic material (MAT8) support for composite solid elements:** PCOMP entries associated with layered solid elements can now automatically convert the 2D properties on MAT8 entries to the 3D MAT12 format. This provides a very easy migration path for users with legacy 2D orthotropic shell models, or data, wishing to transition to the new 3D solid composite elements.

**Added failure plane angle output for LaRC02 failure criteria:** Added failure plane angle ( $\alpha$ ) output for LaRC02 failure criteria.

**Added plain strain support for orthotropic shell elements:** Added plain strain support for orthotropic shell elements via the MAT8 Bulk data entry. This is a popular method for very large, stiff joint cross sections in the maritime industry. It allows for approximation of the appropriate directional material properties in a plane strain orientation.

**Added a parameter to specify the maximum number of iterations used in determining composite LaRC02 strength ratios:** Added PARAM, MAXSRITER which is used to specify the maximum number of iterations used in determining composite LaRC02 strength ratios.

**Enhanced composite transverse shear stress:** Enhanced composite transverse shear stress for plies with orientation angles other than zero or 90 degrees and unsymmetric lay-ups. The new method gives a slightly more accurate result.



## Element Enhancements

New element enhancements provide better accuracy for models with coarser mesh densities.

**New formulation for CTRIA3 and CTRIAR element bending stiffness:** Added a new element bending stiffness formulation (DKT) which gives more accurate solutions for coarser mesh densities. The new formulation is set as the default and is controlled using the TRIELEMENTYPE model parameter. The previous element formulation can be selected by setting PARAM, TRIELEMENTYPE to SRI. The DKT option is available in all solutions.

**New formulation for CQUAD4 and CQUADR element bending stiffness:** Added a new element bending stiffness formulations (DKQ and DKT) which may give more accurate solutions for coarser mesh densities. The new formulation is controlled using the QUADELEMENTYPE model parameter. Generally, the SRI option is more accurate but in cases with large length to thickness ratios, the DKQ and DKT options may provide better results. The DKQ option is supported in all solutions. The DKT option is supported in all linear solutions.

**Added support for general elements:** Added support for general elements via the GENEL Bulk Data entry.

## Results Enhancements

**Added results neutral file compression:** Added results neutral file compression via the RSLTFILECOMP directive for grid point vector and element load files. When RSLTFILECOMP is set to ON or AUTO and ON is selected by the program, disk space requirements for intermediate results neutral files are significantly reduced and in most cases performance is increased especially for models with composite properties and SUBCOM Case Control commands.

**Added a parameter to output normalized stress error measures at each grid:** Added PARAM, STRESSERROR which when set to ON will output normalized stress error measures at each grid and a relative solution error for all shell and solid elements by surface and volume respectively. This is a very useful feature as it provides a graphical assessment of mesh quality allowing easy identification of areas requiring mesh refinement.

**Added support for extended results labels:** Added support for extended results titles and labels in the FEMAP Binary Results File (.FNO). The new labels are now up to 80 characters long and provide a better description of the associated results measure.

**Added support for real-time element X-Y plotting and .CSV file generation:** Added element data support for XYDATA and XYDATAGEN Case Control commands. .CSV file support is also added. This enhancement and the previous nodal data output enhancement mean that the user who wants limited response data quickly, or who wishes to use response data in his own or third party programs can avoid the need to parse the Model Results Output File. Previous MSC or NX users in particular will find great utility in avoiding the need for parsing the F06 file.

**Additional output for bars and beams:** Added von Mises stress output for bar and beam elements. The beam and bar von Mises calculation has been completely updated to include both transverse shear terms and the torsional shear stress term. The effective section shear area is used to calculate the former. The latter uses either a user supplied effective torsional radius, or the appropriate dimension if the user inputs a section via a PBEAML or PBARL.

**Additional support in linear solution for output stress in bars, beams and shells:** Added PARAM, EQVSTRESSTYPE which when set to 2 in linear solutions will output membrane only von Mises stress in bar, beam, and shell elements. The default setting of 0 will output equivalent stress for material nonlinear solutions and von Mises stress for all others.

**Support for translating the global applied load vector into FORCE and MOMENT Bulk data entries:** Added the TRSLLOADDATA directive which when set to ON will translate the global applied load vector into FORCE and MOMENT Bulk Data entries. The OUTLOADSETID directive is used to specify the output set id.

**Added support for translating the global displacement vector into SPC Bulk Data entries:** Added the TRSLDISPDATA directive which when set to ON will translate the global displacement vector into SPC Bulk Data entries. The OUTDISPSETID directive is used to specify the output set id.

**Added a parameter for the method used in grid point force balance calculations:** Added PARAM, GPFORCEMETHOD which specifies how grid point forces are calculated. The NORAN option uses the previous method which only calculates element force contributions for elements which have an element FORCE request. The NASTRAN option considers all elements regardless of FORCE request and is consistent with MSC and NX Nastran.

**Provided support for MSC Nastran element type codes and labels compatibility:** Added the PCHFILETYPE directive which when set to NASTRAN will provide compatibility with MSC Nastran element type codes and labels.

**Enhanced DMI strain data:** Increased precision of exported strain and direct matrix import (DMI) data.

**New output results for static and prestress static solutions:** Added multipoint constraint vector resultant output to linear static and prestress static solution sequences.

**Added new output vector to the Model Results Output File:** Added rigid body acceleration vector output to the Model Results Output File when PARAM, INREL is set to ON or -1.

**Added support for specific strain output:** Added support for specific strain output via the THERMAL, MECH, and TOTAL options on the STRAIN, ELSTRAIN, and GPSTRAIN Case Control commands. The THERMAL option will output thermal strain and the MECH option will output mechanical strain in place of the default TOTAL strain.

**Enhanced Model Result Output File:** Reduced Model Result Output File size.

**Added results support for CADAS:** Added results neutral file support for the CADAS pre- and post-processor via the RSLTFILETYPE directive and CADAS setting.

**Added a parameter for specification of units for output results:** Added PARAM, UNITS which allows the specification of the units used in the model for output labeling and report generation.

## Other Enhancements

**Added better temperature dependent material support:** Added support for temperature dependent density in linear static solutions with more than one subcase. Previously, the temperature dependent density from the first subcase was used for all subcases.

**Added support for heat flux interpolation:** Added support for heat flux interpolation via the QBDYG Bulk Data entry.

**Modified how neutral files are deleted at initialization:** Modified how the PURGE directive handles deletion of results neutral files on initialization. Previously, all neutral files were deleted regardless of if they were requested. Now only the neutral file type requested will be deleted.

**Added new options for LOADINTERPOLATE command:** Added two new options for the LOADINTERPOLATE Case Control command: SFORCE and PNORMAL. SFORCE is similar to the FORCE option except that the output total force is scaled to equal the input total force. PNORMAL is similar to the PRESSURE option except the pressure is forced to be normal to the element surface.

**Added support for translating element pressure loads into PLOADG Bulk Data entries:** Added the TRSLPRESDATA directive which when set to ON will translate element pressure loads into PLOADG Bulk Data entries. The OUTGRIDOFFSET directive is used to specify the starting grid point id associated with the generated PLOADG entry. This enhancement improves the bidirectionality of data mapping between NEiNastran and external CFD programs or other applications requiring point wise definition of pressure.

**Enhanced translation of heat transfer models:** Improved translation performance for heat transfer models with CONV Bulk Data entries.

**Enhanced superelement analysis:** Added support for user defined single point constraints on superelement interior grid points.

**Added information to the .LOG file:** Added parameter and directives settings information to the .LOG file.

**Added a directive for providing timing information to the .LOG file:** Added ELAPSEDTIME directive for providing detailed timing information to the .LOG file.

**Added heat transfer solution support for LOAD Bulk Data entry:** Added support for load combination via the LOAD Bulk Data entry in heat transfer solutions for heat flux loads (QBDY1 and QBDY2 Bulk Data entries) and volumetric heat addition loads (QVOL Bulk Data entries).

**Enhanced model initialization directives:** Added single directive support for FILESPEC, FILEBUFFERSIZE, FILEPFACTOR, and NFILEBUFFER.

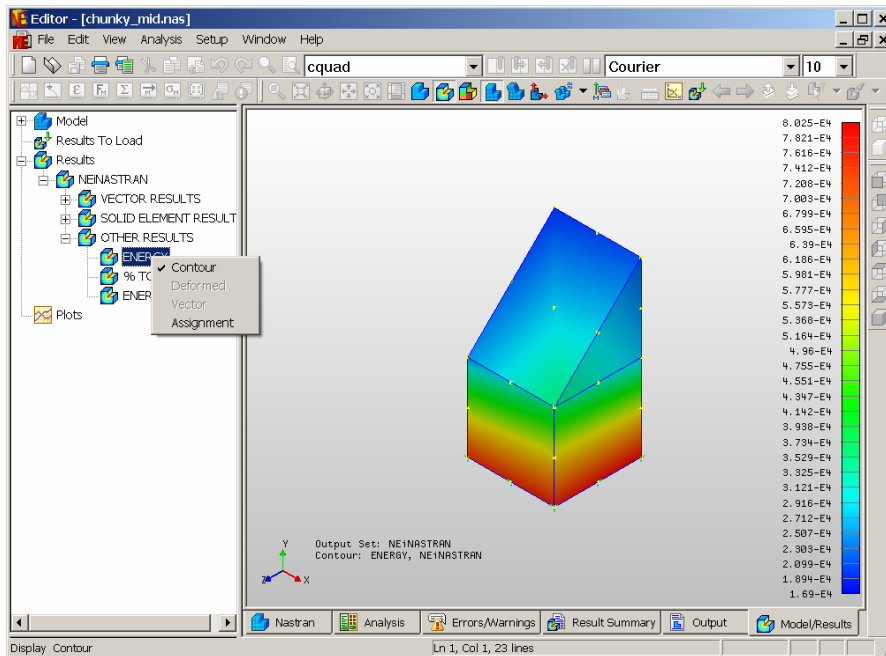
**Enhanced Bulk Data processing:** The following enhancements related to bulk Data processing were made:

- Added support for mixed lower and upper case for ENDDATA, PARAM, and INCLUDE Bulk Data entries and BEGIN BULK Case Control command.
- Added wide field support for PLOADG Bulk Data entries.
- Changed the handling of repeated SPC requests where both zero and non-zero SPC Bulk Data entries are specified at the same degree of freedom so that now a non-zero SPC will not be overwritten by a zero value at the same degree of freedom.
- Added SEID field support to the GRDSET Bulk Data entry.
- Changed default behavior for PLOAD4 Bulk Data entries when a zero magnitude orientation vector was defined so that now the pressure will be oriented normal to the surface (default).

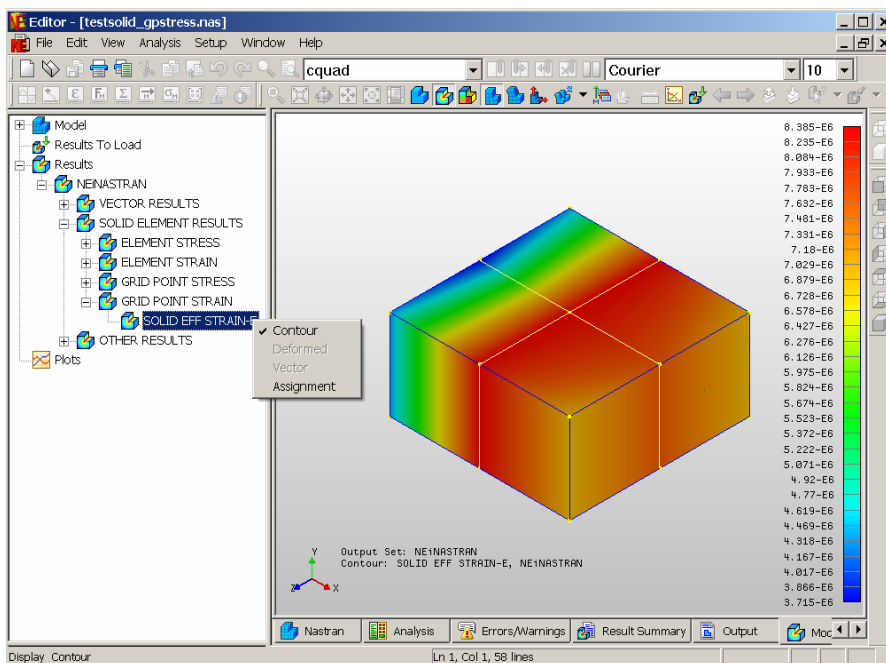
## Editor Enhancements

The NEiNastran Editor has been updated with new features, such as:

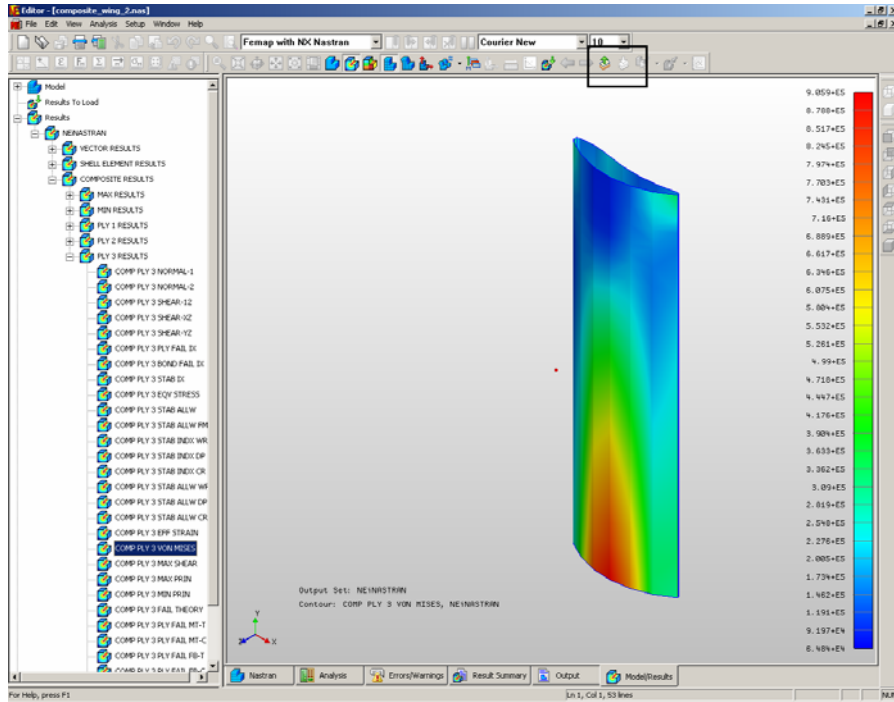
- Editor Graphical User Interface modified to enhance user interface usability.
- Added display of mid-side nodes for 2D and 3D elements.



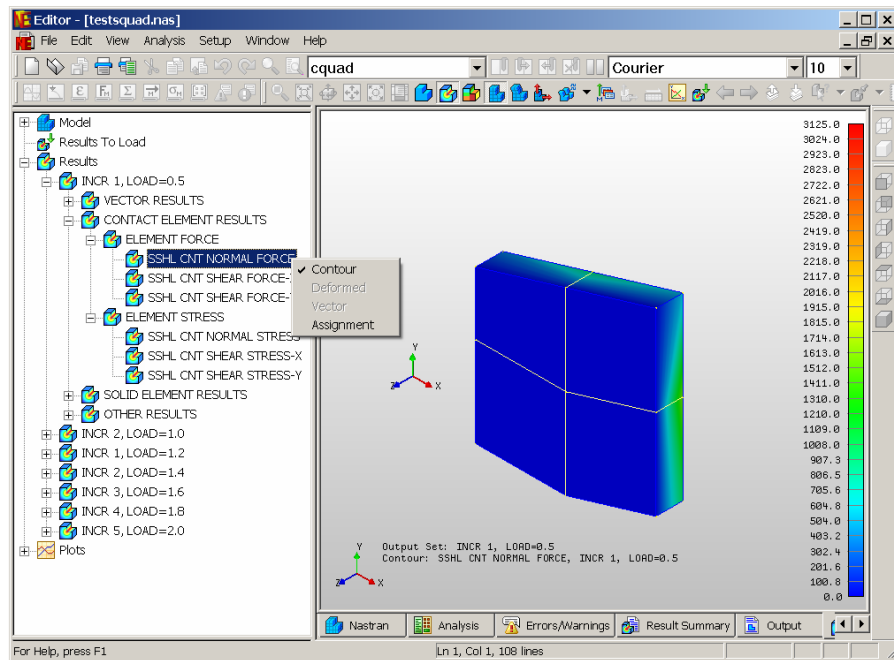
- Added handling and display of Grid Point Stress and Grid Point Strain.



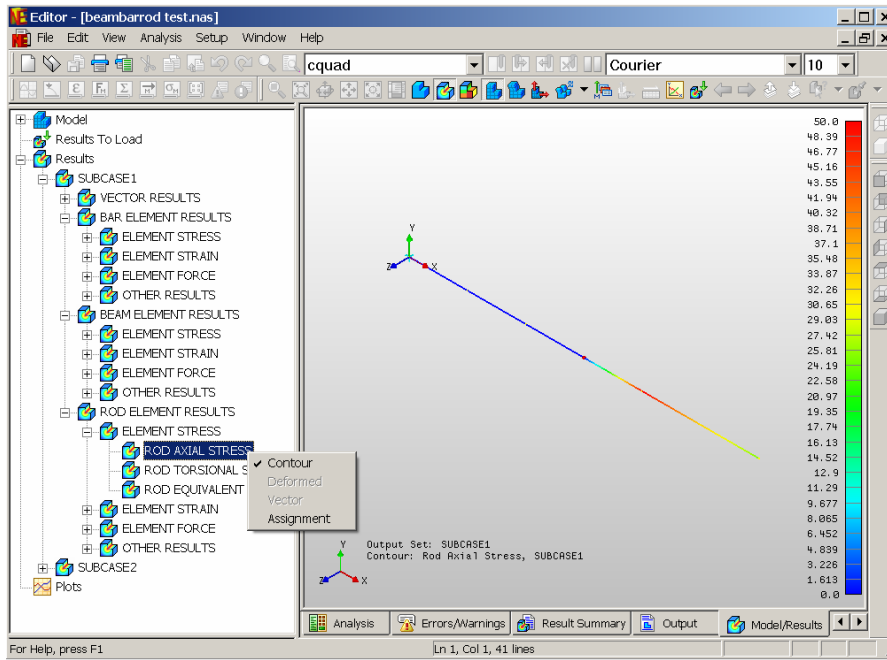
- Added capability to move from ply to ply for models with composites.



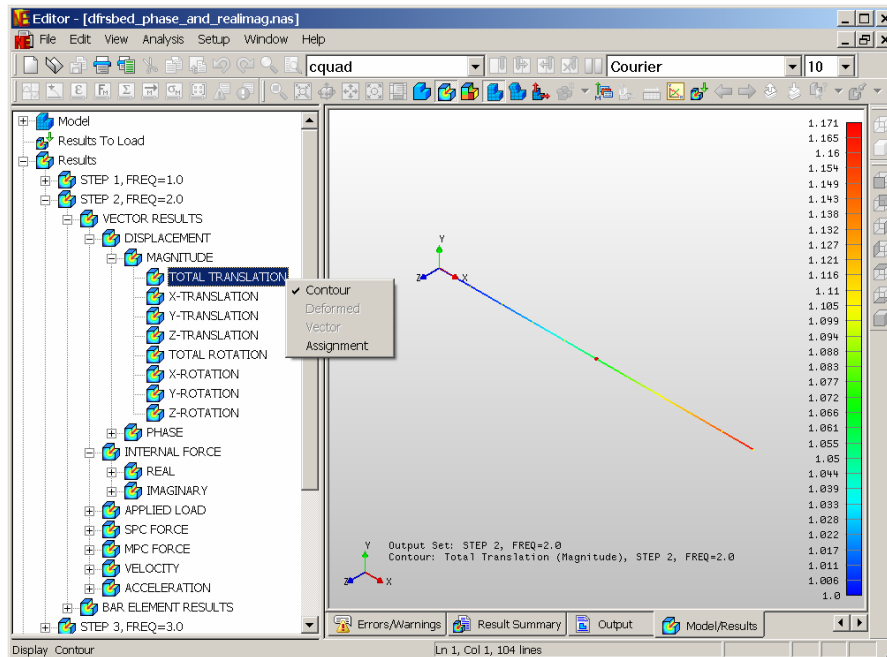
- Added handling and display of surface contact results.



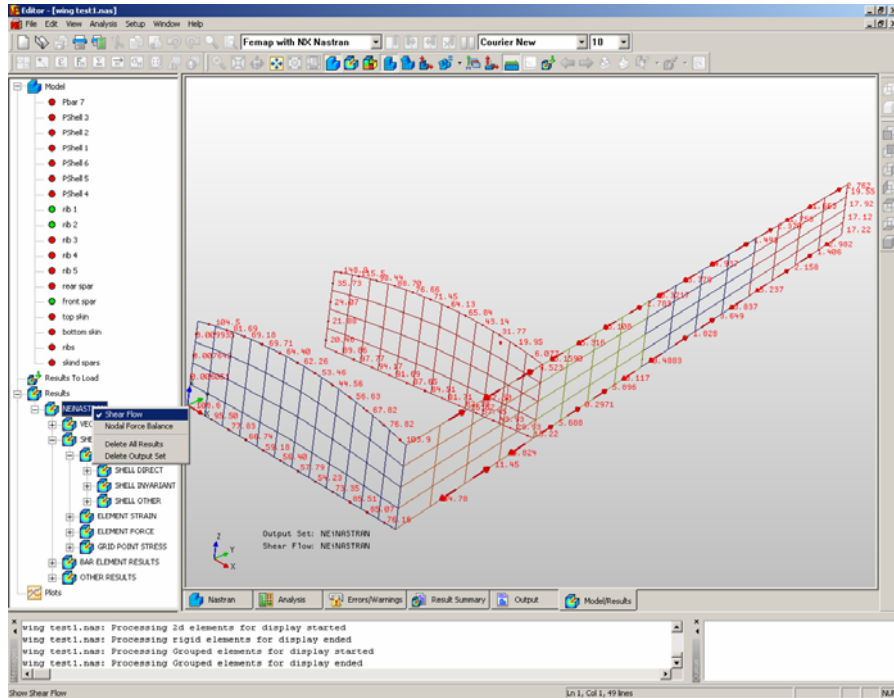
- Added handling and display of Bar, Rod and Beam results.



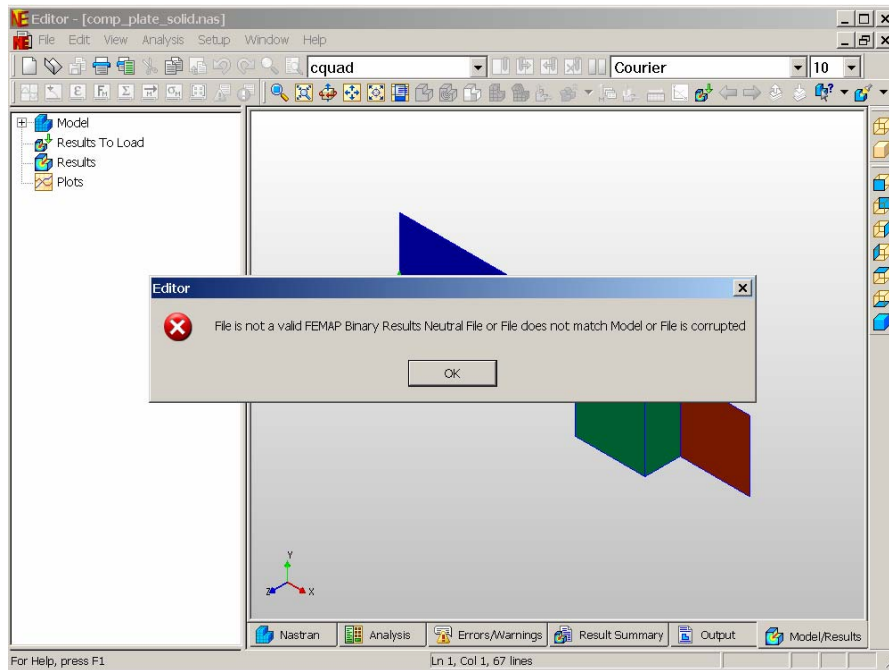
- Added handling and display of complex results for Polar and Rectangular complex data.



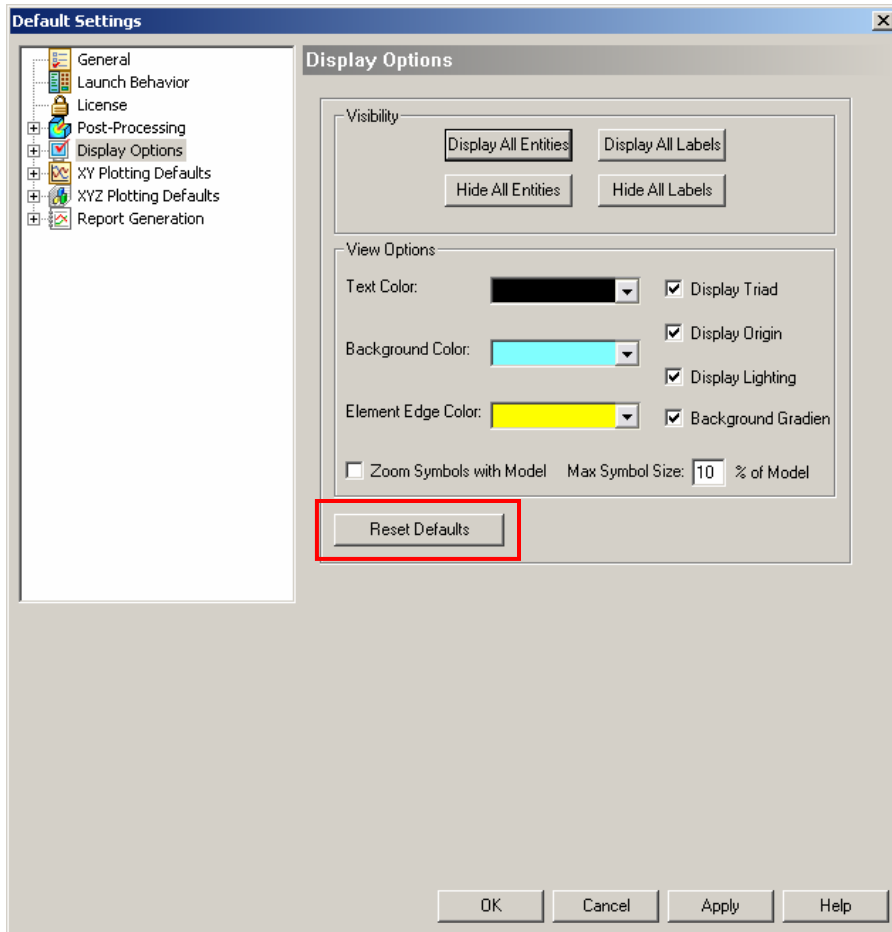
- Enhanced display for Shear Flow and Nodal Force Balance for a selected set.



- Added a cursory check for results file validity for a loaded model.

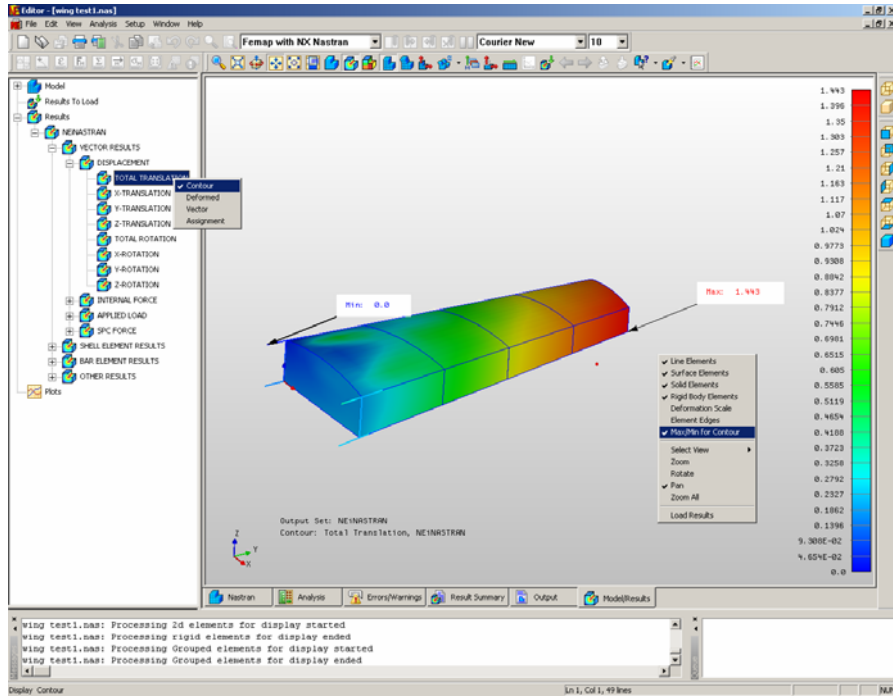


- Added the capability to restore default settings in the Setup property pages.



- Added the capability to delete all plots and to delete individual plots.
- Added visualization and display of results for rigid body elements.

- Added markers for maximum and minimum values on contour plots, which can be toggled on or off using graphics view popup menu.
- Added the "TOTAL" calculated results, such as "TOTAL ROTATION" and "TOTAL SPC FORCE", for all Tensor6 x, y and z components in Model tree and display of results.



**Noran Engineering, Inc** is aggressively focused on commitment to the customer. Detailed documentation, customized on-site training, and comprehensive technical support ensures that you will see immediate return on your investment.

**For more information about our company or our products, please contact:**

**Headquarters:**

**Noran Engineering, Inc**  
5555 Garden Grove Blvd., Suite 300  
Westminster, CA 92683-1886  
USA  
Phone: 1.714.899.1220  
Fax: 1.714.899.1369  
Email: [info@noraneng.com](mailto:info@noraneng.com)  
Website: [www.NENastran.com](http://www.NENastran.com)

**Europe:**

**SmartCAE**  
Piazza della Gualchierina, 9  
59100 Prato  
ITALY  
Phone: +39.0.574.404.642  
Fax: +39.0.574.401.265  
E-mail: [info@smartcae.com](mailto:info@smartcae.com)  
Website: [www.smartcae.com](http://www.smartcae.com)

**Asia/Pacific:**

**Digital Solutions**  
Kyoei Nakasuji Bldg., 3-7-18  
Nakasuji, Asaminami-ku  
Hiroshima 731-0122  
JAPAN  
Phone: +81.82.831.1190  
Fax: +81.82.831.1193  
E-mail: [post@digital-sol.co.jp](mailto:post@digital-sol.co.jp)  
Website: [www.digital-sol.co.jp](http://www.digital-sol.co.jp)



NEiNastran *for Windows*

From Noran Engineering, Inc.