

NEiNastran Version 9.0

Noran Engineering announces the release of NEiNastran Version 9.0, its latest upgrade with extensive enhancements in performance and capability.

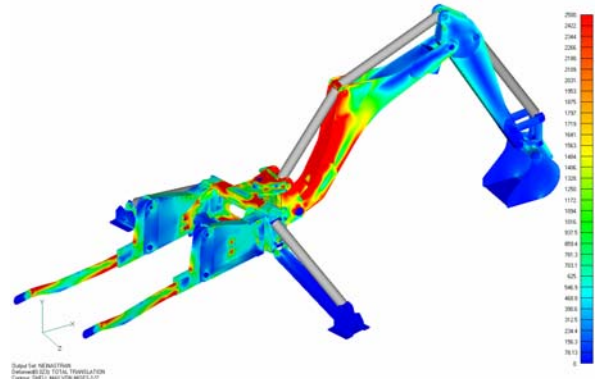
NEiNastran V9.0 is a major upgrade with over 50 customer driven enhancements. Major features include support for automated internal superelement generation, enhanced nonlinear and dynamic solutions, improved solver and overall performance, a faster Lanczos eigensolver, support for XDB results neutral files, and new aerospace, maritime, and automotive productivity tools. Several NEiEditor enhancements have been included such as a fully automated HTML report generator, shear flow plots, assignment plots, an enhanced X-Y plotting interface, and a secure DDAM data input form.

Performance Enhancements

- Nonlinear solutions are now up to 2x faster.
- Enhanced Lanczos eigensolver with over 4x faster performance on models where 50 or more modes are requested.
- Improved performance for models with composite laminates and large numbers of plies where individual ply results are requested.
- Performance is improved for models with HEXINODE or QUADINODE set to ON or AUTO when ON is automatically selected and for models with tapered beam elements.
- Added support for model input files over 10 million lines in size.
- Enhanced VSS solver to automatically revert to out-of-core mode if unable to allocate memory for in-core solution.
- Added MINRAM directive to specify the minimum amount of memory for the VSS solver to use if an out-of-core solution is performed due to a failed attempt at an in-core one.
- Reduced memory requirements for direct transient response analysis.

Nonlinear Analysis Enhancements

- Enhanced nonlinear surface contact analysis improving both accuracy and performance as well as solution stability with complex contact scenarios.
- Added PARAM, NITERMUPDATE which controls the material nonlinear update strategy.



- Added two new forms of surface contact: bi-directional slide and rough contact. Bi-directional slide prevents contacting regions from separating or closing but permits sliding (zero coefficient of friction). Rough contact allows separation and closure but does not permit sliding (infinite friction). Both types support symmetric and unsymmetric contact and are selected in field 8 on the BSCONP entry.
- Added displacement based stiffness updating for surface contact friction. When the friction condition is in the stick state the transverse stiffness for stick is adjusted to achieve the specified displacement.
- Added PARAM, MAXBISECTRESTART which when set to ON will permit restarting a nonlinear static solution which has terminated on a FATAL ERROR E5076: MAXIMUM NUMBER OF BISECTION PERMITTED REACHED error.
- Added a THRU literal option for specifying grid point IDs on BSSEG Bulk Data entries. This option is useful when defining contact between a surface and a point or line.
- Added an AUTO option for PARAM, MAXADJEDGE which sets a value of 50 if PARAM, SLINEMAXACTDIST is set to AUTO and zero if it is set otherwise.
- Changed labeling of effective plastic strain to EFF STRAIN-E for linear solutions and nonlinear solutions without material nonlinearity where von Mises strain is output. For nonlinear solutions with material nonlinearity the effective plastic strain is labeled EFF STRAIN-P.

- Added PARAM, NITERCUPDATE which controls the surface contact update strategy. The default AUTO setting uses an optimum value based on other nonlinear parameters.
- Added PARAM, NLTOL which specifies default NLPARM Bulk Data entry EPSP and EPSW tolerances. Values range from 0 to 3 with 0 being the highest tolerance (accuracy) and 3 being the lowest (performance).
- Changed the default value for PARAM, BISECT from OFF to ON to improve accuracy in material nonlinear solutions.
- Added an AUTO option for the MAXITER field on the NLPARM and TSTEPNL Bulk Data entries. When MAXITER is set to AUTO or blank, an initial MAXITER setting of 40 is used and is automatically increased as the number of iterations approaches MAXITER if the solution is converging.
- Enhanced shell and solid element initial strain for better solution stability. Initial strain input is now scaled based on the load scale factor for nonlinear static solutions.
- Improved convergence in nonlinear static solutions with parabolic tet elements with material nonlinearity and with PARAM, SPARSEITERMODE set to 1, 2, or 3.
- Added PARAM, NLKDIAGMINAFACT which sets the minimum nonlinear static augmentation factor when PARAM, NLKDIAGAFACT is greater than zero.
- Added logarithmic interpolation for the TABLES1 Bulk Data entry.

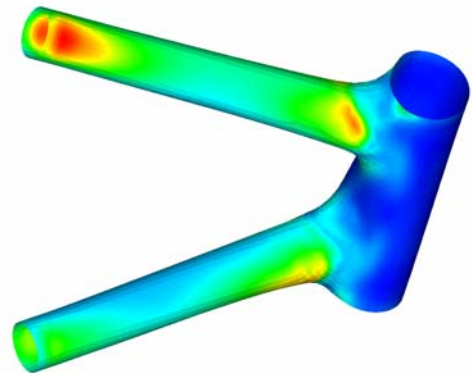
Dynamic Analysis Enhancements

- Added support for vibration fatigue analysis in random response solutions via the VIBFATIGUE Case Control command and the SCDATA Bulk Data entry. Output includes both life and damage results measures as well as the biaxiality ratio.
- Added von Mises and optional maximum and minimum principal stress/strain results for random response via PARAM, RANDRESPINVLEVEL which when set to 1 will output von Mises stress or strain and when set to 2 will also include principal and max shear stress or strain and biaxiality ratio in frequency and random response solutions.
- Added results output for bush elements in random response analysis.
- Added multipoint constraint force output for random response solutions.

- Added PARAM, MAXEIGENRESTART which defines the permitted number of eigensolver restarts when an invalid shift scale is either externally defined or internally estimated. Also, added PARAM, EIGENSHIFTFACT which specifies the shift scale multiplier used to increase the shift scale for an eigensolver restart.
- Added viscous damping force output for bush elements in transient and frequency response solutions.

Heat Transfer Analysis Enhancements

- Modified surface contact in heat transfer to require an active contact element to be in contact for a conduction element to be generated.
- In transient heat transfer solutions grid points with different initial condition temperature and enforced transient boundary temperature will now have the initial condition temperature corrected as well as issue an E5036 warning message. Previously, only a warning message was generated.
- Removed the requirement for a TEMPERATURE(INITIAL) Case Control command to be specified in steady state heat transfer analysis when there are no temperature dependent materials specified.



Composite Analysis Enhancements

- Added von Mises strain output for linear solutions using EFF STRAIN-E measure when either STRESS or STRAIN is requested. Also added von Mises stress and strain using the EQUIVALENT STRESS and EFF STRAIN-E measures respectively for composite laminate individual ply results. These measures are available when either STRESS or STRAIN output is requested.

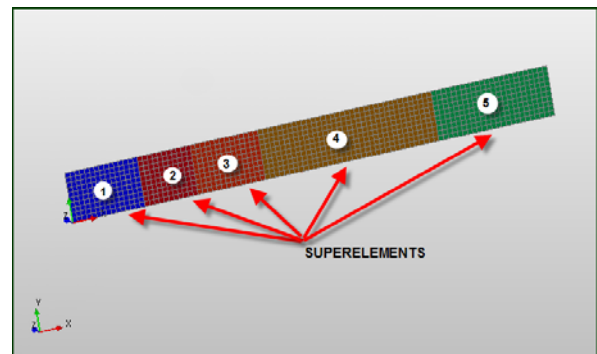
- Added a new composite laminate failure theory based on the Puck PCP criteria.
- Added ACS option on PCOMP Bulk Data entry for automatic selection of core material type (foam = FCS or honeycomb = HCS) for stability index calculations. FCS is selected automatically for isotropic materials and HCS for orthotropic materials.
- Added failure theory used for composite laminate individual ply results data written to the FEMAP Binary Results Neutral File.
- Added support for strength ratio calculations for the LaRC02 failure theory.
- Added feature to revert failure theory used in composite laminate individual ply results from LaRC02 to Tsai-Wu if a non-unidirectional material is detected. PARAM, LARC02TSAITOL has been added to control the tolerance that triggers reversion based on the ratio of E1/E2, XT/YT, and XC/YC.
- Improved performance for models with composite laminates and large numbers of plies where individual ply results are requested
- Modified how the Hill failure theory handles cases where one or both of the extensional allowables is different in tension versus compression. Previously, the sign of the interaction term was used to separately select whether to use the tensile or compressive allowable.
- Added dedicated support for 3D orthotropic materials via the MAT12 Bulk Data entry (similar to the MSC MATORT).
- Added support for laminated solid elements. Both the CHEXA and CPENTA now support layered material definitions using the PCOMP Bulk Data entry. Either isotropic or 3D orthotropic materials can be specified. Output is similar to 2D laminated composite shells and includes in-plane and interlaminar stress, strain, and failure index data. Support is provided for all linear structural solution sequences.
- Added strain energy output for composite shell elements.

Element Enhancements

- Added support for parabolic shell elements via the CQUAD8 and CTRIA6 Bulk data entries.
- Added the WELDGENERATE Case Control command for the automated generation of CWELD elements from existing bar elements.
- Added the SHELLEGRID directive which when set to ON converts all QUAD4/QUADR and TRIA3/TRIAR elements to QUAD8 and TRIA6 elements respectively.
- Added bush element support to the grid point force balance output table in the Model Results Output File. Previously, only the individual elas element components were displayed.
- Enhanced tension-only quad element export of reverted element data via a new TRSLTOQEDATA directive. For each subcase CQUAD4/CQUADR and CSHEAR entries are written out to the Bulk Data Output File when the TRSLTOQEDATA directive is set to ON.

Superelements Enhancements

- Added superelement generation capability via the SELEMGGENERATE Case Control command which generates superelements using standard output sets.
- SESET and SEELT Bulk Data entries have been added for superelement support.



- Add Added automated internal superelement generation and full data recovery for linear static, normal modes, and modal response solutions using Craig-Bampton CMS reduction techniques and a very easy to use yet robust user interface. Users can define regions of their model for individual superelement generation using standard Nastran SET commands and NEiNastran automatically generates internal superelements with automatic partitioning and full data recovery. Also supported are the standard SESET and SEELT Bulk Data entries as well as the SEID field on the GRID Bulk Data entry for defining superelements. The number of component modes is defined via PARAM, NCBMODE. Each internal superelement may be exported after generation to DMIG format via the EXTSEOUT Case Control command. User specified external superelements can be supplied via standard DMIG Case Control commands and Bulk Data entries and are automatically assigned to a superelement.
- Changed the model database default file extension from .DAT to .NDB to prevent conflicts with other applications which use the .DAT extension to define Nastran input files.
- The Model Input File can now have a .DAT extension when specified on the Nastran command line.
- Added PARAM, AUTOFIXRIGIDSPC which when set to ON will automatically correct improperly constrained RBEi elements by adding a CELAS element with stiffness defined by PARAM, KRIGIDELEM between the dependent grid point and the applied single point constraint.
- Added the TRSLRBSEDATA directive which when set to ON will translate out CELAS and GRID Bulk Data entries added when PARAM, AUTOFIXRIGIDSPC was set to ON and modifications were made.
- Added complete support for tabbed model input files.
- Enhanced surface contact weld elements with significant protrusion.
- Reduced surface contact weld elements with significant protrusion.

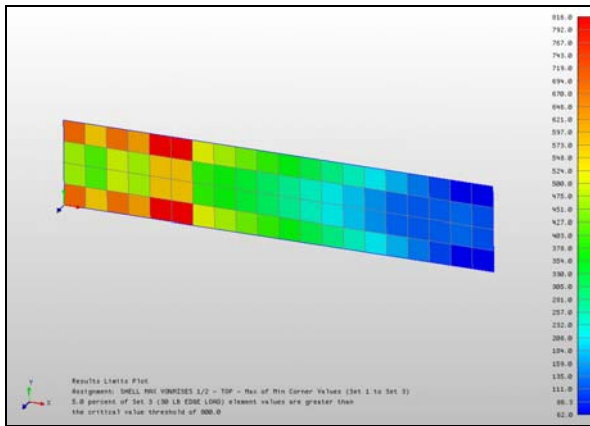
Other Enhancements

- Modified how local coordinate systems are handled on rigid elements and MPC equations when connected to other rigid elements and/or MPC equations. Now the dominating local coordinate system will be used excluding the basic system unless only the basic system is specified.
- Added EIGR Bulk Data entry for controlling the subspace eigensolver. Support is also provided for eigenvector normalization based on a specified grid point.
- Added large field support for CORD1X Bulk Data entries.
- Added an AUTO setting for PARAM, NDAMP which uses 0.01 for all solutions except transient heat transfer where 0.3 is used
- Added automated surface contact and surface weld generation via the CONTACTGENERATE Case Control command and the CONTACTGEN and CONTACTTOL model parameters. Support is also provided for export of generated BSCONP and BSSEG Bulk Data entries when the TRSLMODLDATA directive is set to ON.
- Added DMIGADD Case Control command which combines multiple DMIG matrixes into a single set which can then be referenced via K2GG, M2GG, etc. commands.
- Coupled mass matrix formulation is no longer forced when a DMIG mass matrix is imported unless the matrix has off-diagonal terms. Previously coupled mass was used regardless.

Results Enhancements

- Added PARAM, PARTGEOMOUT which when set to ON will output to the Model Results File mass and mesh properties by property type and identification number. Support will be provided later for output by group. This parameter is mainly provided to support analysis report output.
- AddedPARAM, PARTMASSOUT which when set to ON will output to the Model Results File mass properties by property type and identification number. Support will be provided later for output by group.
- Added .PCH PUNCH file support for real element results data.

- Added von Mises strain output for linear solutions using the EFF STRAIN measure when either STRESS or STRAIN is requested. These measures are available when either STRESS or STRAIN output is requested.
- Added support for strain curvature output via the STRCUR option on the STRAIN Case Control command.
- Modified RESULTLIMITS Case Control command to include a step set id field which allows individual time steps, frequency steps, etc. to be specified when determining results measure limits.



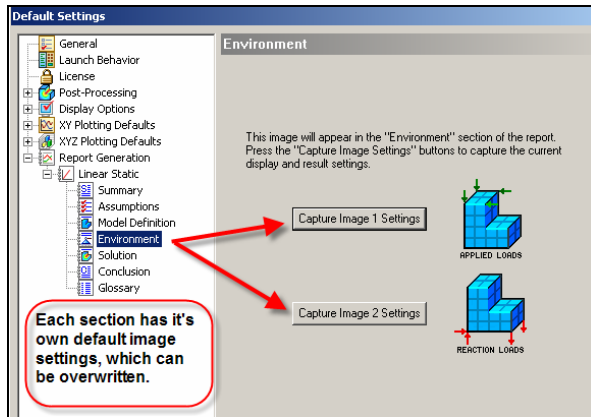
- Added PARAM, PCHFILEDBLEPRCS which when set to ON will output real data in PUNCH output using 15 decimal places versus 6 when set to OFF.
- Added bush element linear strain output.
- Added RESULTLIMITS command support for SUBCOM generated subcases.
- The PURGE directive now deletes incremental FEMAP Binary Neutral Files generated when the INCRSLTOUT directive is set to ON.
- Added the RSLTFILEMETHOD directive which defines the method used to generate the Nastran Binary Results Neutral File. The SINGLEPASS option is faster but may not be compatible with all post processors due to the way data blocks are sequenced. The MULTIPASS method will be slower but is typically more compatible.
- Reduced results output not specifically requested by the user.

- Added support for the Nastran XDB results file format through the RSLTFILETYPE directive when set to NASTRANXDB. The Nastran XDB format allows selective import of results into FEMAP and other post processors.
- Increased xy-plot capability. The previous XYDATAGENERATE limit of 50 grid points has been increased to 1000.

Editor Enhancements

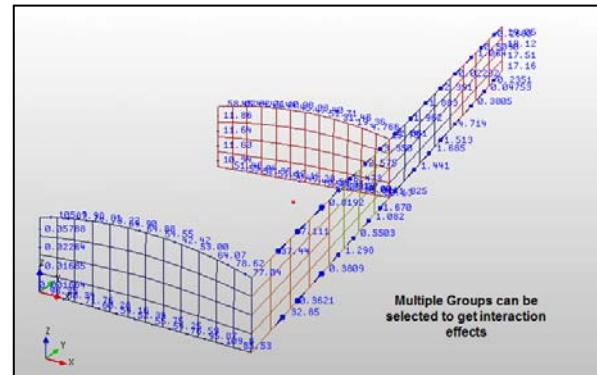
- Custom X-Y plotting of results:
 - Users can click on the Create Plot icon after loading the results in the Model View and a dialog is displayed. Users can select output sets for the X-axis and results to plot for the Y-axis.
 - Users can enter a node or element ID to plot, or can select the node or element with the mouse.
 - Users can then click a button for a Plot Preview. This is helpful because there may not be any data the user wants to save.
 - When users close the Plot Preview, they are prompted whether to save the plot or not. If yes, the plot is saved and displayed in the Plots tree in the Model/Results pane.
 - Alternately the user can click button for View/Save. The plot is displayed and automatically saved and displayed in the Plots tree.
 - The plots are permanently saved in the .xyp file.
- Custom settings defined for each individual plot:
 - When the user double clicks on a displayed plot, a Custom Settings dialog bar is displayed. The user can select the desired settings for this plot.
 - The user can also reset the Defaults if they want. The changed settings can be immediately previewed by clicking the Apply Settings button.
 - When OK is clicked, the custom settings for this individual plot are saved (to the "xyp" file).
 - When this plot is loaded again it reflects its custom settings.
 - The user can click the Data button to save the XY data to the clipboard.

- Improved highlighting feature:
 - When the user clicks to highlight nodes or elements, a dialog is displayed on the side of the view.
 - Elements or nodes can be specified by adding them to the list, or by selection with the mouse. When the user clicks OK, all the entities in the list are highlighted.
 - In work: selecting entities by box selection with mouse.
- Improved options menus in tree-like format:
 - The setup dialog box has been re-implemented using a tree-control on the left side and property pages on the right side.
 - When the user clicks a tree item the property page for the options for this item are displayed.



- Added report writer:
 - HTML format report can be generated after the users load the .fno file while some PARAMs are set to ON.
 - The HTML report contains summary of the analysis. Other data includes group definition, contact definition, element initial distortion summary, applied load vector resultant, reaction load vector resultant, displacement data, peak displacement component, and stress result summary of the model.
 - Users can add their own conclusion and modify the default summary and glossary list.
 - Default images are generated for the HTML report, or users can go to “Default Setting” to setup their own image preference.

- Users just have to click the “Generate Report” button and a wizard will come up to guide them to generate their reports.



- Added shear flow vector plots:
 - The Editor now plots vectors and values representing the shear running load around the external edges of any group of shell elements.
 - The action to plot is controlled via a button icon.
 - The user must include Grid Point Force Balance in the output request.
 - The groups defined in the NEiNastran Modeler can now be retrieved by the Editor from the Neutral file. This allows the external shear flow of any component to be visualized.

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
Website: 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
Website: 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
Website: www.digital-sol.co.jp



NEiNastran for Windows
From Noran Engineering, Inc.