

Simcenter Femap version 2020.2

Maximizing simulation productivity

Benefits

- Performance enhancements throughout the application to improve “quality of life” for users
- More robust editing and defeaturing of geometry to aid in idealization of complex parts
- Ability to transfer data directly to Microsoft Excel for further investigation and examination
- Expanded support for Aeroelasticity and Multi-Step Nonlinear Analysis types in Simcenter Nastran, as many of the world’s engineering tasks require more than linear static analysis

Summary

Simcenter™ Femap™ software is a stand-alone finite element modeling pre- and postprocessor for engineering simulation and analysis. The software is CAD-independent and can import geometry from all major CAD platforms and supports most CAD data formats. Simcenter Femap also works in combination with a wide variety of finite element analysis solvers, including the industry-leading Simcenter Nastran software.

The latest release provides a variety of enhancements that will improve your productivity across the simulation workflow. First and foremost, performance improvements throughout Femap should lead to greater productivity and quality of user experience, especially for those working with larger models. The modeling process, whether starting with CAD geometry that requires defeaturing before being idealized or updating the existing mesh of a legacy model imported from a solver input file, has become more capable, robust, and consistent. Direct interaction between Femap and Microsoft Excel provides seamless transfer of data used for both pre and post-processing activities, including the ability to quickly send large amounts of output directly to a formatted worksheet. There are also several

updates to solver integration for the multi-step nonlinear and aeroelasticity solution sequences of Nastran®, as well as updates for Ansys®, Abaqus® and LS-DYNA®.

Simcenter Femap is now being released on a biannual schedule in the spring and the fall, which began with version 2019.1 and continues with version 2020.2. The software is now referred to as Simcenter Femap to reflect that it is a part of the Simcenter portfolio of Siemens CAE products. For the same reason, NX™ Nastran® software is now Simcenter™ Nastran®.

Visualization and user interface

High DPI Support

Previously, all user interface scaling for Femap was either handled by Windows, which could result in blurry text and icons, or no scaling was done which could make text and icons small and hard to read. As a first step, all icons were modernized in Femap v2020.1 to improve color contrast and consistency throughout the user interface. For 2020.2, multiple sizes of all icon images are now available to allow appropriate user interface scaling to fully support high resolution (4K) displays.

Curve Connectivity

Sometimes it can be difficult to determine if, or how surfaces of idealized geometry (i.e., mid-surfaces) are connected to one another. This option allows the user to display each curve in the model using a unique color representing the connection between different surfaces, if there is a connection. For instance, Free edges displayed in Red,

- Full High DPI support to facilitate use of Femap on high resolution displays
- Curve Connectivity option to display connections between surfaces in idealized geometry to promote a fully connected mesh
- Send to Excel commands to directly transfer Data to Excel from many of Femap's tools
- Augmented capacity to display results from advanced analysis types

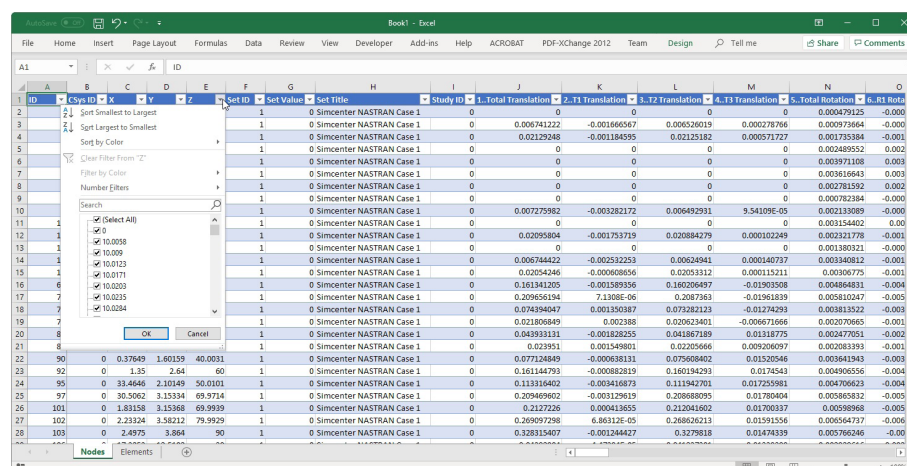
When using any menu command to renumber an individual entity type, a Renumbering Options section has been added to simplify renumbering with either Offset method.

Send to Excel commands have been added to various tools in Femap to automatically transfer data directly to Microsoft Excel. By default, the data is sent using formatted HTML, which offers some advantages over Text and Rich Text formats. These icons and buttons can be found in the Standard Entity Selection and Layup Editor dialog boxes, along with the Charting, Data Surface Editor, Function/Table Editor, Mesh Point Editor, Connection Manager, and Data Table panes.

The Tools, Mass Properties, Solid Properties command was enhanced to allow calculation of mass properties for any number of selected solids, either by treating them as an assembly and computing a single set of values for all selected solids or computing mass properties for each individual solid. In addition, options to create a representative mass element with appropriate properties and connect it to an existing mesh are now available to specify via a consolidated interface.

Send to Clipboard options were added for both commands on the Tools, Mass Properties menu, which send a copy of the listing created by either command to the clipboard in Text, Rich Text, and HTML formats.

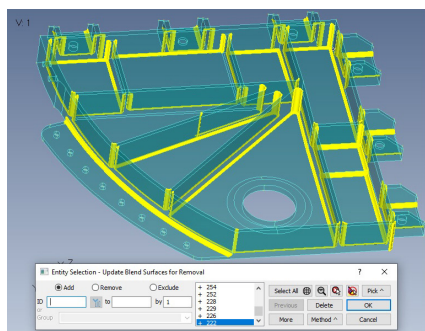
In Nonlinear analysis, it is quite common for the model to experience large deformations. It can be beneficial for force and moment loads to continue to act in the original direction, relative to the model, as the model deforms, thus the load essentially “follows” the model. Follower forces and follower moments can now be defined using either two nodes or four nodes to maintain the direction of the loads as the model deforms. These types of loads can also be applied to geometric entities, with the direction being maintained by selection of points instead of nodes.



Meshing

Feature Removal Tool

Removing blends and fillets is somewhat typical when defeaturing solid geometry before the geometry is idealized using midsurfacing techniques. In earlier versions of Femap, setting Feature Type to Blends in the Feature Removal tool of the Meshing Toolbox could be used to select only blends and fillets that were automatically detected. With the advent of the Preview option, the user can now see which blend and fillet surfaces are targeted for removal, then add or remove surfaces from the selection before completing the operation.



Geometry Editing Tool

While it has been possible select faces to extend or shorten solid geometry using a few different tools in the Meshing Toolbox, the process has now been streamlined when using the Extend Operation in the Geometry Editing tool. Simply choose the Surface/Replace Face option, select a single face as a destination, then select any number of faces to extend to or be replaced by the destination face.

One particularly helpful aspect of this functionality is it can be used to remove selected blend and fillet surfaces by replacing those faces with the single destination face.

Mesh Editing

The use of legacy solver input files can provide a challenging set of issues, as they have no associativity to any type of geometry, thus the existing mesh cannot simply be deleted, the model resized, and meshed again. For this reason, commands have been added to the Mesh, Editing menu over several different releases, with some of the newer tools better equipped to handle entities associated with the elements being split or refined. Because of this, updates have been made to all tools on the Mesh, Editing menu to improve consistency with regard to how loads, constraints, regions, element options (tapered beams, offsets, plate thickness, and material direction), groups, and when available, geometric associativity are handled.

Solver support

Simcenter Nastran enhancements

The Simcenter Femap 2020.2 release includes Simcenter Nastran 2020.1 (version 1915).

Nastran enhancements (MSC and Simcenter)

The NASTRAN GENEL is now supported. GENELs can be entered in FEMAP via a tabular interface and are displayed graphically via their connected degrees-of-freedom (DOFs).

Full read and write access allows for "round trip" support of existing NASTRAN input decks.

Follower Forces and Follower Moments have been added to provide new critical loadings for nonlinear analysis. Two Node (FORCE1 and MOMENT1) and four node (FORCE2 and MOMENT2) are both supported.

Simcenter Nastran SOL 401/402

Additional Functions and Control options have been added to support changes in Simcenter NASTRAN SOL401 and SOL402, including new Stress vs. Strain functions, shell offset control, and contact stabilization options.

The MATDMG material type has been added for Simcenter NASTRAN's Multi-Step Nonlinear Solver. This material type provides damage model specification that is used to compute damage values, energy, status, and crack density to predict progressive ply failure in models containing solid laminates.

ANSYS Interface

Added options to skip writing subsets of the Femap model to ANSYS. Users can now choose to skip Coordinate Systems, Nodes, Materials, Properties, Elements, Groups, Shapes, and/or Misc. (View and Solution Settings). Furthermore, to provide full flexibility to ANSYS users, manual control (override) options have been added for ANSYS Command and Model Control, Load Steps, Time Steps, Modal Analysis, Harmonic Analysis, Random Vibration, Buckling, Nonlinear, and Output Requests.

Import of results from ANSYS results files which have been compressed by the solver is now supported, as is the ability to request ANSYS create the results file using the compressed format.

LS-DYNA

Similar to the ANSYS options above, control is now provided to skip entities that are written to the LS-DYNA input file. Users can now skip Coordinate Systems, Nodes, Materials, Properties, Elements, Groups, Functions, and/or Misc. (Comments, Titles, and Solution Settings)

Added an Analysis Monitor for LS-DYNA to monitor the progress of an analysis launched on the same machine as Femap and optionally automatically import the analysis results.

ABAQUS

Added options to automatically bundle loads or constraints that reference a set entry in the ABAQUS input file into a Femap Load Definition. Also, added option to export them in this format as well.

Post-Processing

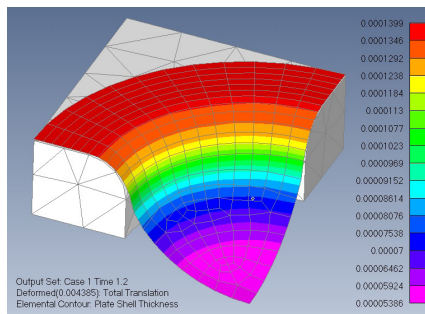
Postprocessing Send to Excel

List, Output, Results to Excel will automatically open Excel, then transfer user selected nodal or elemental output data. Options are included for row/column control, transformations, and formatting.

Display of Results from Advanced Analysis Types

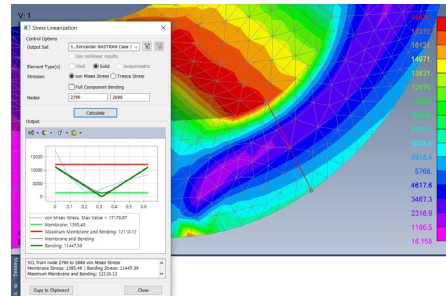
Additional post-processing options for Aeroelasticity have been added. Femap can now recover Aero Pressures, Aero Coefficients, and Aero Forces results from NASTRAN Static Aeroelasticity solution SOL 144 (recovered from the .F06 file), then display results on the aero mesh of the aero panels and aero bodies.

Similarly, Femap can now recover results from Multi-Step Nonlinear solutions including Plastic Strain (SOL 401/402) and Shell Thickness (SOL 402), as well as Damage Status, Damage Values, Damage Energy, and Crack Density for progressive ply failure (SOL 401/402).



Stress Linearization

View, Advanced Post-Processing, Stress Linearization has been added to Femap to replace a Visual Basic Add-On tool that has been provided via the Femap Community site. This tool implements the ASME Pressure Vessel Stress Linearization that calculates a linearized membrane and linearized bending stress along a user-defined Stress Classification Line (SCL).



Customization

API – Multiple Femap Sessions

Two new API functions, `feAppRunningApplicationInfo()` and `feAppGetRunningApplication()` allow software developers to connect with multiple FEMAP sessions running concurrently on the same machine. This functionality can be accessed from any software development platform, including from inside of Microsoft Word or Excel.

API – File, Merge

A new object, `feMergeTool`, has been added to expose the capabilities of the File, Merge command. It allows selection of the "From Model" by name and properties to specify options, along with methods to toggle entity types, either individually or by group, add related entities, and specify orientation/transformation options.

API – Stress Linearization

The same object used to calculate the ASME linearized membrane and bending stresses along a user-specified SCL is now available via the `feStressLinear` Object.

Siemens Digital Industries Software
[siemens.com/software](https://www.siemens.com/software)

Americas +1 314 264 8499
 Europe +44 (0) 1276 413200
 Asia-Pacific +852 2230 3333