SIEMENS
Ingenuity for life

Simcenter Femap version 2020.1

Benefits

- First implementation of synchronous technology for geometry modification
- Streamlined workflows and re-use of previous data definitions
- Upgraded support for various analysis applications

Features

- Updates to UI and preprocessing operations
- Improvements to existing meshing functionality
- Added support for simulation entities used in more advanced solutions

Summary

Simcenter™ Femap™ software is a standalone 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.

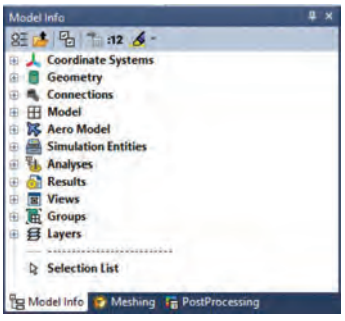
The latest release provides a variety of improvements that will improve your productivity across the simulation workflow. Model creation enhancements consist of more robust editing of geometric features, the ability to align geometric patterns useful for mapped meshing, and improved defeaturing capabilities for fillets and blends. For meshing, existing workflows have been enhanced to allow greater flexibility and more options when creating line elements from the edges of other elements or sweeping elements along an existing mesh. There are also several updates to solver integration for Nastran®, including support for monitor points and direct matrix inputs (DMIGs), as well as Ansys®, Abaqus® and LS-DYNA®.

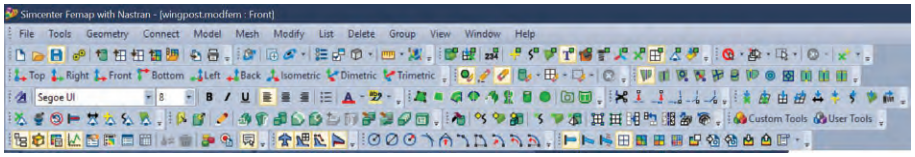
Simcenter Femap is now being released on a biannual schedule in the spring and the fall, which began with version 2019.1 and continues with version 2020.1. The software is now referred to as Simcenter Femap to reflect that it is a part of the Simcenter portfolio of Siemens CAE products. For the same reason, NX™ Nastran® software is now Simcenter™ Nastran®.

Visualization and user interface

Modernized icons

All icons found throughout the user interface have been modernized to incorporate design elements utilized in commercial software applications across all industries. This includes icons on toolbars, in the menu structure, within dialog boxes and in the model tree, toolboxes and other panes.





siemens.com/femap

Simcenter Femap version 2020.1

In addition, on every dialog box that offers the ability to select a color for an entity, the palette button has been replaced by a color block that displays the current color, and if specified, the line style of the entity. This enables the user to determine the current color without having to know the identifier of the color.

Unified label architecture and enhanced digit control

All labels displayed in the graphics window are now being drawn using a unified architecture, meaning labels will not change when using performance graphics, legacy OpenGL graphics, or a mix of both.

In conjunction with moving all labels displayed in the graphics window to a unified architecture, control of how digits are displayed was enhanced significantly to offer users greater flexibility. There are now four methods available, Femap classic, exponent, significant figures, and decimal places, each of which offers unique features for digit display. Global settings for digit control can be specified via the preferences and overrides can be specified via view options for load vectors, constraints, deformed model, contour/criteria style, contour/criteria legend, contour arrow options, and freebody vectors.

Geometry enhancements

Smart select for feature editing

Feature editing via the meshing toolbox has been enhanced to leverage portions of Siemens synchronous technology by addition of the smart select option, which automatically selects additional surfaces to improve the probability that feature editing will be successful. When smart select is enabled, five additional options become available – offset, identical, concentric, colinear, and tangent edges – which act as rules that govern the selection of additional surfaces.

Improved feature removal for blends

The feature removal tool in the meshing toolbox has improved the blend removal process to remove blends and fillets from solid geometry that could not be removed in previous versions of Simcenter Femap. In 2020.1, an attempt is now made to remove all the blends at once, then a second attempt is made to remove individual chains of blends one at a time. In addition, the removal of blends on stepped geometry is now more robust.

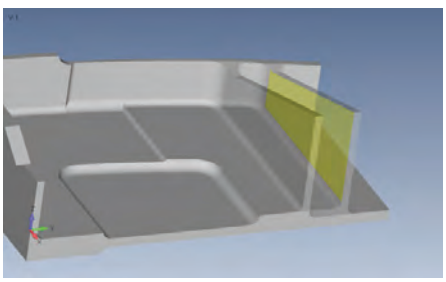
Washer pattern alignment for geometry editing

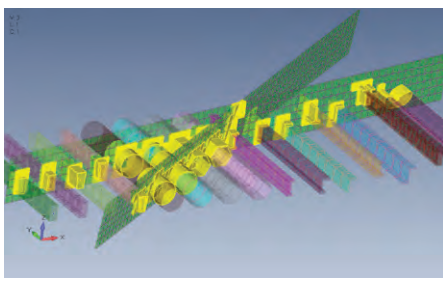
An alignment option has been added to the washer operation in the geometry editing tool of the meshing toolbox. The options can be used to force the split lines of the washer pattern to be aligned with a specified vector, tangent to a curve, or perpendicular to a curve.

Preprocessing enhancements

Mesh interference checking

Version 2020.1 adds the ability to determine if any edge(s) of any line, planar, or solid element passes through other element(s), which can reduce the possibility of this common modeling error. After selecting the tools→ check→ mesh interference command, any element that fails this check will be highlighted in the graphics window, then users are asked if they want to create a group of the highlighted elements.





Multi-set functionality added to existing commands.

The workflows for several commands that use set-based entities for input have been streamlined to allow the selection of multiple sets. Also, additional options have been added to improve the usability of each command.

The model→ load→ from output command now offers the ability to create loads of a specific type, and in some cases new load sets, from the specified output vectors. When using a single output set, the new loads will be placed in the active load set.

The tools→ check→ sum forces command now offers the ability to perform a summation of the applied forces for a single set, all sets, or any number of selected sets. In addition, the individual summation of each load set can now be sent to the data table.

More efficient creation of face-based loads

The model→ load→ elemental on face command allows the user to create face-based loads (pressure, heat flux, convection or radiation) with less interaction by moving the selection of the element faces to the beginning of the command.

Meshing

Creation of elements using edges or faces of existing elements

For 2020.1, the commands on the mesh, edge/skin elements menu have been simplified by moving selection of element edges or element faces to the beginning of the command, which is more intuitive. This allows the user to preview the edges or faces where new elements will be created before moving forward to select a property. In addition, users are now asked if they want to offset any newly created line elements, which prevents them from having to update the elements using another command. Also, it is now possible to select the edges of solid elements using the standard edge selection dialog box.

Additional options when sweeping elements along element edges

For 2020.1, options to merge the newly created elements with the existing elements and/or limit the sweep to only the elements that are currently visible in the active view are now available.

Improved automatic creation of rigid and interpolation elements

It can be useful to automatically create rigid (RBE2 in Nastran solvers) or interpolation (RBE3) elements between a source node and other target nodes based on proximity. In Femap, this is accomplished with the mesh→ connect→ rigid command, which now offers the ability to specify a minimum number of target nodes for each source node and/or allow a target node to be connected to multiple source nodes.

These same options have also been added to the model→ load→ from free-body command when using the multi-model workflow.

New node merging option when creating a solid

While it has been possible to set a node merging option when creating a solid tetrahedral mesh for quite some time, there were some limitations with how the merging of coincident nodes would occur. This has been addressed with the new "into model" option, which attempts to merge the newly created nodes with any existing nodes but does not attempt to merge all nodes in the entire model.

Solver support

Solver interface updates include support for monitor points and direct matrix inputs for Simcenter Nastran solvers, as well as enhancements for Simcenter Nastran's multi-step nonlinear solution sequences. In addition, this release also includes further enhancements to the ANSYS, Abaqus, and LS-DYNA solver translators.

Nastran enhancements

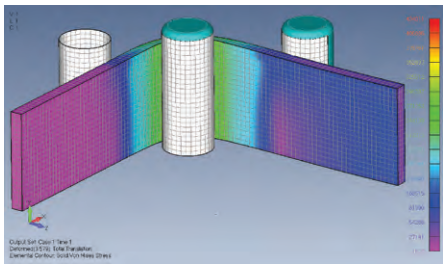
Support for reading and writing of monitor points (MONPNT1 and MONPNT3) was added for Femap 2020.1. In addition, the analysis set manager has been augmented to select which monitor point(s) should be exported for analysis. While Simcenter Nastran and MSC Nastran both offer MONPNT1 and MONPNT3, they are implemented differently, so please consult the solver documentation for guidance on solver support.

Direct matrix input entities are used by analysts to add stiffness, mass, damping and other quantities to a model. They are available for almost all solution sequences and can be selected in the master case and/or any subcase.

Simcenter Nastran enhancements

For SOL 401 and 402, a section has been added to the multi-step global control options dialog box to specify parameters used to perform restarts. In addition, items have been added to the multi-step control options dialog boxes, along with the solution and convergence options dialog boxes, for both SOL 401 and 402, making it possible to specify everything available on the NLCNTL and NLCNTL2 entries from within Simcenter Femap. Finally, support was added for output types that are only created by SOL 401 and SOL 402.

The Simcenter Femap 2019.1 release includes Simcenter Nastran 2019.2 (version 1884).



MSC Nastran enhancements

Added support for thermal contact parameters on the BCTABLE entry which specify values for HCT, HCV, HNC, BNC and EMISS.

ANSYS translator

To better align with the support offered for Simcenter Nastran solvers, a group can now be selected using the "portion of model to write" drop-down in the ANSYS command and model control dialog box to export only a portion of the model to the ANSYS input file.

For random vibration analysis, support was added to allow specification of a PSD table to allow for correlation of multiple subcases with multiple excitations. The 2020.1 release includes improved support for planar elements that have variable thickness defined.

Target KEYOPTs can now be specified on the ANSYS tab of the define connection property dialog box.

Abaqus translator

- Added read and write support for directional pressure loads (DLOAD, TRVEC)
- Improved read support for contact defined in the Abaqus input file to create more accurate connection regions and connectors.
- Spring/damper elements and corresponding properties are now created when importing connection section entries for bushing and beam types.

LS-DYNA translator

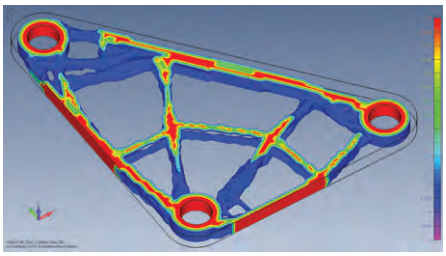
To better align with the support offered for Nastran solvers, a group can now be selected using the "portion of model to write" drop-down in the LS-DYNA model options dialog box to only export a portion of the model to the LS-DYNA input file.

Release 2020.1 adds the ability to specify Timoshenko as an element formulation for beam elements. The release also improves support for planar elements that have variable thickness defined.

Postprocessing

Capped isosurface plots

Two capping options are now available when viewing an isosurface plot. When using the negative cap option, any portion of the model that is below the value specified for single isosurface will be contoured, while using the positive cap options will contour any portion of the model above the specified value. This type of plot can be very useful when reviewing results of topology optimization or thermal analysis.



New output vector IDs

The ID ranges of output vectors have been changed for Simcenter Femap 2020.1 to offer larger ranges of IDs for certain types of output. The output vector IDs will be automatically updated when opening a model or importing a neutral file created in a version of Simcenter Femap prior to 2020.1.

Siemens Digital Industries Software
siemens.com/software

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

© 2019 Siemens. A list of relevant Siemens trademarks can be found here. Other trademarks belong to their respective owners.

81125-C6 11/19 H