



SIEMENS

Ingenuity for life

Simcenter Femap version 2021.1

Maximizing finite element analysis efficiency with enhanced meshing tools

Benefits

- Maximize finite element analysis (FEA) efficiency with enhanced meshing tools
- Improve quality of a hex mesh using various tools in the meshing toolbox
- Simplify user interface to display only available options for selected solvers via analysis filtering
- Use the familiar interface and depth of functionality of Excel to update data in the function/table editor in real time or in bulk

Features

- Mesh control explorer pane to assist in creating complex shell and solid meshes
- Customizable options for mesh propagation in various sizing commands
- Improved performance during entity selection, especially when area or front picking
- Comprehensive support for MSC Nastran SOL 400 to perform implicit nonlinear analysis

Summary

Simcenter™ Femap™ software is a standalone finite element modeling (FEM) pre- and postprocessor for engineering simulation and analysis. The software is computer-aided design (CAD) independent and can import geometry from all major CAD platforms. It supports most CAD data formats. Simcenter Femap also works in combination with a wide variety of FEA solvers, including Simcenter Nastran® software.

Simcenter Femap, which is part of the Xcelerator™ portfolio, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software, is now being released on a biannual schedule in the spring and the fall, which began with version 2019.1 and continues with version 2021.1. The software is now referred to as Simcenter Femap to reflect that it is a part of the Simcenter portfolio of Siemens computer-aided engineering (CAE) products. For the same reason, NX™ Nastran software is now Simcenter Nastran software.

The latest release provides a variety of enhancements that will improve your productivity across the simulation

workflow. A large amount of functionality added to 2021.1 focuses on facilitating mesh creation and modification. First, matching mesh sizes and determining mesh propagation between disconnected geometric entities can be specified based on the proximity of edge curves to one another. To determine where these mesh propagation options will be enforced, tools were created in the new mesh control explorer pane to provide visual feedback for edge curves, which are fully paired, partially paired or unpaired, along with other tools to visualize which surfaces are paired for solid meshing or have a surface mesh approach assigned. In addition, multiple tools in the meshing toolbox can be used to update an existing hex mesh. Another area of expanded capabilities is support for MSC Nastran SOL 400. From a modeling standpoint, items have been added to specify property extensions, nonlinear materials and contact options designed for use with SOL 400. At the same time, a new analysis type may be selected that offers user interface (UI) elements to define critical and useful solver parameters for SOL 400. Finally, updates have also been implemented for Simcenter Nastran and Abaqus.

Preprocessing enhancements

Improved entity selection performance

Entity selection (picking) now uses OpenMP technology to improve performance, especially for area picking (box, circle, polygon and freehand) and front picking. In addition, the area within the standard entity selection dialog box

Simcenter Femap version 2021.1

where ID ranges are listed is now hidden whenever more than 1,000 ranges containing unique IDs have been selected, which also improves performance. The list can be restored by clicking into the list area, then hidden again by using the hide list button.

Function/table editor – connecting to Excel

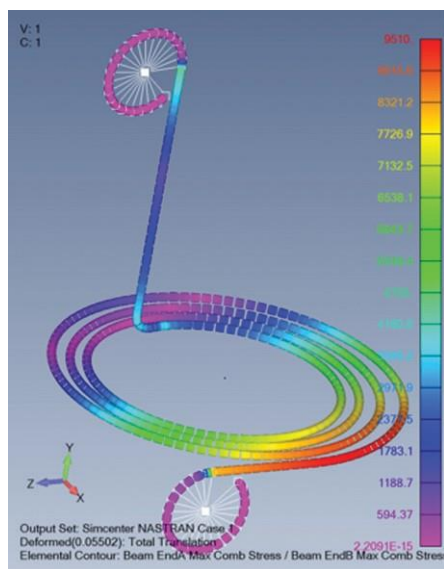
Using the send to Excel functionality introduced in v2020.2 allows data in the function/table editor to be updated in real time as it is the same data modified in Excel or in bulk after the data has been modified.

Midsurfacing overlap repair

This is an added option that can potentially limit overlap of surfaces created at intersections where the ribs approach the intersection from various directions and do not meet in a single location.

Geometry, curve-centerline

This is an enhanced algorithm to better handle more complex beam-like solids, including helical and swept shapes created along complex splines.



Tools, check, mesh interference

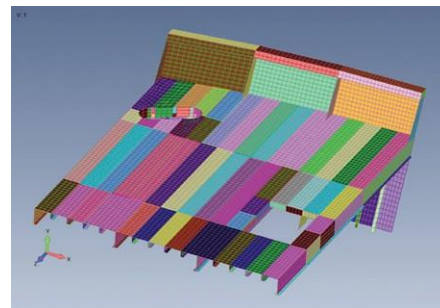
It provides improved command by adding the ability to specify a tolerance

value for the interference factor or proximity factor. When using the interference factor, interference is determined when elements overlap by more than the given tolerance. When using the proximity factor, interference is determined when elements are separated by a gap that is smaller than the given tolerance.

Meshing

Mesh control explorer

A new pane to facilitate mesh creation has been added to the 2021.1 version. Available tools can be used to determine if surfaces are paired using surface mesh approaches for solid meshing, if surfaces have other types of mesh approaches defined and/or if curves can be automatically paired based on proximity for shell meshing. It also allows specifying options to use for automatic mesh propagation.



Updating a hex mesh

The capabilities of the sizing and surface tools in the meshing toolbox have been improved to allow updating of a hex mesh without having to select multiple curves to insure correct mesh propagation on all the necessary solids.

Solver support

Analysis filtering

Analysis filtering can be used to remove solver-specific tabs for connection properties and connection regions from the interface, along with materials found in the drop-down for other types. This reduces visible UI elements to only those needed for the selected solvers.

Filtering can be done permanently via the interface tab of preferences or as needed by using this icon found in the individual controls.

Simcenter Nastran

The Simcenter Femap 2021.1 release includes a relatively recent release of Simcenter Nastran. There is enhanced support for dynamic design analysis method (DDAM) analysis with added options for mode selection, which limits modes used in the calculation based on a specified cutoff, percentage or top value. It can write the SELMODE entry to the Simcenter Nastran input file. Support was also added for results requested by PARAM, SHLMEM and YES, which include multipoint constraint (MPC) forces, element strains and membrane-only stresses for beams, shells and composite laminates.

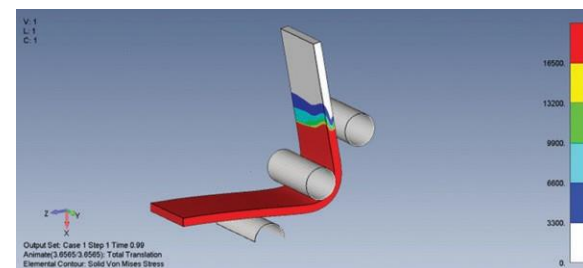
Simcenter Nastran/MSC Nastran

There is added support to limit modes by mode number which are used when running a modal frequency response or modal transient analysis. It writes the MODSEL entry for Simcenter Nastran while writing the MODESELECT entry for MSC Nastran.

MSC Nastran

SOL 400 overview

SOL 400 is used to perform implicit nonlinear analysis using MSC Nastran. The type of analysis, statics, normal modes or transient dynamics can be changed at the subcase level or within a subcase at the analysis step level. When multiple analysis steps exist in a single subcase, they are always sequentially dependent, which is known as analysis chaining.



SOL 400 – connection properties

Although many of the options on the MSC Nastran tab existed in previous versions of Simcenter Femap, support now exists for advanced cohesive contact and interference options.

SOL 400 – connection regions

Rigid body regions can now be defined using 2D planar curves or 3D surfaces, along with more traditional finite-element-based regions. In addition, the MSC Nastran tab can be used to specify friction, smoothing control and quite a few options for rigid body regions.

SOL 400 – property extensions

Property extensions to control geometric and material nonlinearity can now be assigned for rod, bar, beam, plate, plane strain, axisymmetric, laminate plate, solid, solid laminate and solid cohesive properties using the define property extension icon in the user interface.

SOL 400 – materials

Elasto-plastic, elastic 3D orthotropic, cohesive zone modeling, material failure, gasket and four different types of hyper-elastic materials are now supported for use with SOL 400.

SOL 400 – analysis set manager

Nonlinear behavior, automatic contact parameters and additional general contact parameters, such as cohesive contact options, can now be specified at the top level of the analysis set for implicit nonlinear. Control parameters that are used to specify an increment type, fixed, adaptive or arc-length, to control how the solver behaves along

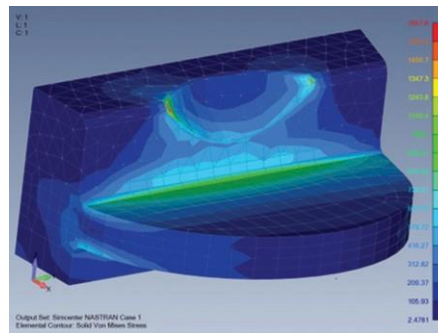
with boundary conditions and output requests can set at the master, subcase or analysis step level. Analysis steps can be defined one at a time or by creating a global analysis step that can then be referenced at the analysis step level.

Postprocessing

Solid element stress contour option

When this option is enabled, result values on linear elements and parabolic elements that share nodes are not averaged together.

One typical example of when this might occur is when a hybrid mesh has been created that transitions from linear hexahedral elements to parabolic tetrahedral elements via parabolic pyramid elements that do not have nodes on the quadrilateral face of the pyramid.



Customization via API

AnalysisStep and GlobalStep objects

These new objects, associated properties and methods have been created to allow specification of options for MSC Nastran SOL 400 Analysis Steps and Global Step programmatically.

PublishTool and PublishTable objects

These new objects and methods are used to publish customized HTML and RTF reports to either the clipboard or to a file. To automatically associate any number of PublishTable objects with a specific PublishTool Object, use the UserTable method to generate the PublishTable Objects.

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

Americas +1 314 264 8499
 Europe +44 (0) 1276 41 3200
 Asia-Pacific +852 2230 3333