

SIEMENS DIGITAL INDUSTRIES SOFTWARE

# Simcenter Femap version 2021.2

Maximizing finite element analysis efficiency

[siemens.com/software](https://www.siemens.com/software)

## Benefits

- Improve quality of triangular and quadrilateral-dominant surface meshes while at the same time reducing required geometry cleanup
- Use geometry containing facet-based surfaces and curves, which is becoming more prevalent
- Determine if geometry is properly set up to leverage automatic mesh propagation capabilities

## 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 finite element analysis (FEA) solvers, including Simcenter™ Nastran software.

Simcenter Femap — 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 2021.2. 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.

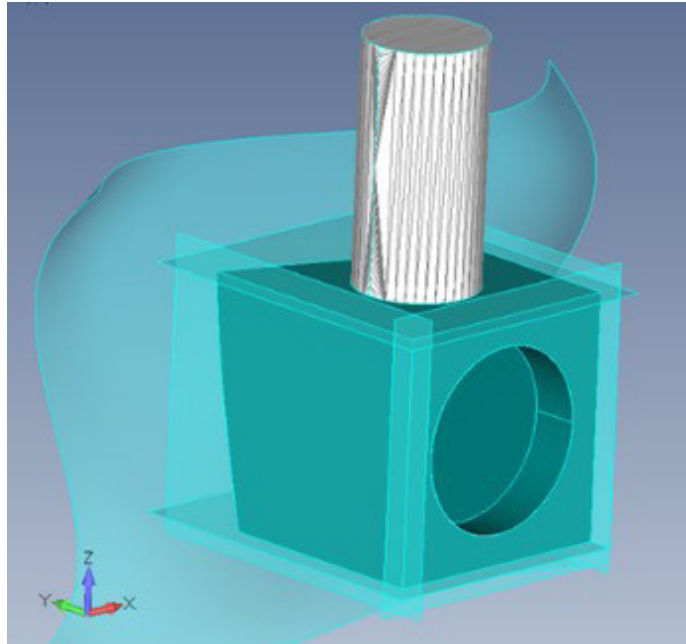
**SIEMENS**

**Benefits** *continued*

- Collaborate with other engineers and managers by creating visualization files containing important information about model setup and analysis results

**Features**

- Powerful new meshing technology that can use geometry or an existing mesh as a starting point
- Ability to import Convergent Modeling™ geometry, then modify and mesh the geometry
- Multiple subcases now available to control behavior of dynamic analysis for Nastran® solvers
- Export of enhanced JT™ data format visualization files containing additional modeling information or results from multiple output cases



The latest release provides a variety of enhancements that will improve your productivity across the simulation workflow. First and foremost, advanced meshing technology used in other products within Siemens Digital Industries Software has been implemented within Femap with the goal of creating a higher-quality mesh by using either geometry or an existing mesh as a starting point. Geometry that uses a faceted representation for surfaces and curves can now be imported into Femap, then modified using many of the available geometry tools and meshed using the new body meshing technology. Additional tools are now available to locate and display edges and surfaces that are not properly subdivided to take advantage of automatic mesh propagation within Simcenter Femap. Support for multiple subcases in dynamic analysis for Nastran solvers allows multiple dynamic or static subcases to be defined for a transient response, frequency response or nonlinear transient response analysis. Finally, the new release provides the ability to create JT visualization files containing additional CAE information for use in any JT collaboration tool.

**Preprocessing enhancements****Ability to import geometry containing facet-based curves and surfaces**

Geometry that uses a faceted representation for curves and surfaces instead of the traditional boundary representation can now be imported into Femap. Once imported, these geometric bodies, often called convergent models, can be modified using many of the available geometry tools and meshed using the new body meshing technology described below.

### **PBUSH1D option for spring/damper and spring/damper to ground properties**

This capability can be used to create a rod type spring-and-damper property to be used in conjunction with CBUSH1D elements in Nastran solvers. The CBUSH1D is essentially a one-dimensional version of the CBUSH element that supports large displacements for nonlinear analysis along with some additional behaviors, including the ability to set up a relationship using design equations to simulate a shock absorber.

### **Simulation entities - design equations**

Nonlinear force function input can be specified to control the spring, damper, shock absorber and/or generalized spring and damper behavior of a spring/damper or spring/damper to ground property and are exported to a Nastran input file as DEQATN entries.

### **All nodes on solid option for selection of elements**

This feature allows any element that has all its nodes associated to the selected solids to be added to the selection when using the standard entity selection dialog box.

### **Group, element, all nodes on solid**

Similar to the all nodes on solid option available for selection, this command adds any element that has all its nodes associated to the selected solids to the active group.

### **Enhanced picking of entities used for aeroelastic analysis**

Building on capabilities of the new picking option added for version 2021.1, which takes advantage of OpenMP to improve performance, selection of aero panel/body, aero property, aero splines, and aero control surface entities now also uses the OpenMP-powered picking technology.

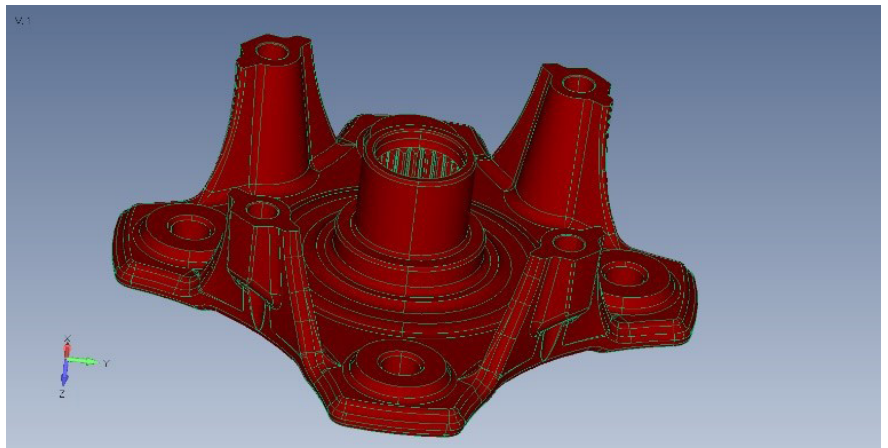
### **Tools, check, mesh interference**

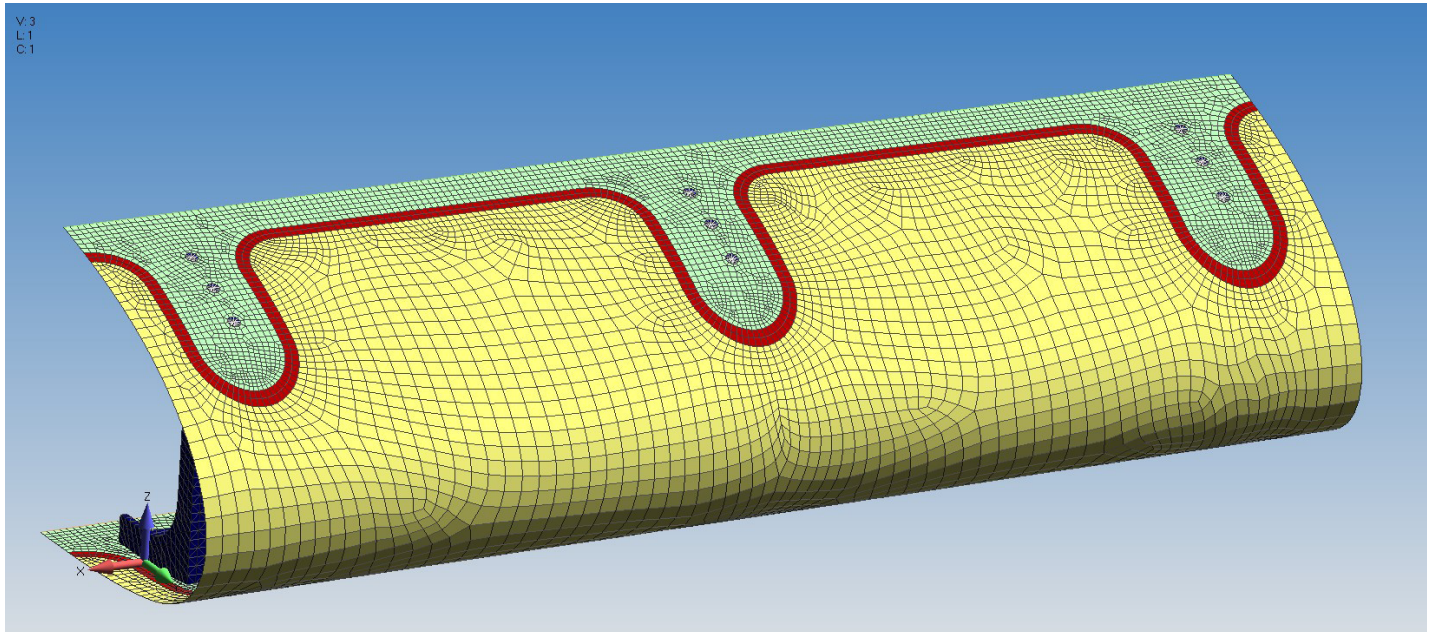
Plot only plates elements are now treated as having zero thickness, which removes numerical roundoff and causes less false positive mesh interferences in models containing plot only elements.

## **Meshing**

### **Mesh, bodies**

Advanced meshing technology used in other Siemens Digital Industries Software simulation products has been implemented within Femap with the goal of creating a higher-quality mesh. For geometric bodies, this command imposes fewer sizing restrictions than the traditional meshing process used in Femap. In addition, this meshing technology can also ignore internal edges, thus reducing the need for much of the geometry cleanup typically required to mesh less-than-ideal geometry.





**Mesh, mesh on mesh**

If geometry is not available, an existing mesh can instead be used as a starting point for the newly implemented body meshing technology to create a mesh with higher-quality elements. In addition, the user chooses which aspects of the original mesh should be maintained in the elements generated by the command.

**Mesh control explorer**

Originally introduced in version 2021.1, additional tools have been added to the mesh control explorer pane that can locate and display any edge that is adjacent to a surface where the surface has no imprinted edges that match the edge, or any surface that pierces any other surface where there are no imprinted curves at the location of the intersections between the surfaces.

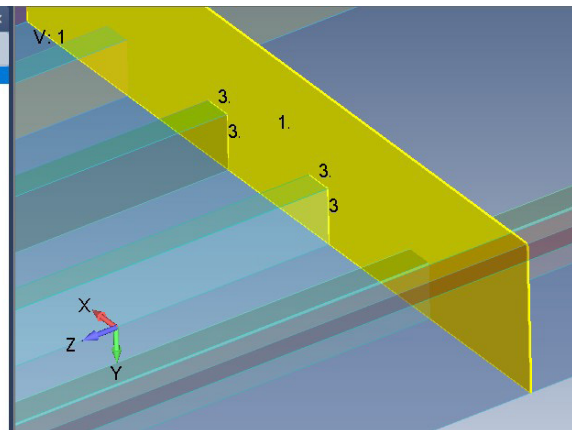
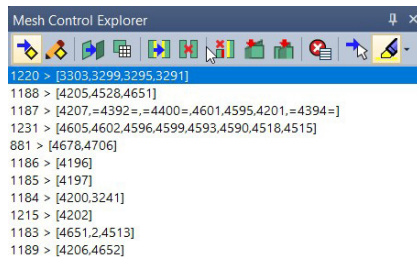
**Meshing toolbox – meshing sizing tool**

A “Set Size” option has been added to the operation section that enables the user to specify a mesh size to define the number of elements on a curve. Also, the “Set To” operation is now “Set Number.”

**Meshing toolbox – geometry editing tool**

When operation is set to “Point to Edge,” a drop-down menu to select a “Projection Mode” is now available. Choosing “Normal” performs a normal projection of the selected point to selected edge; choosing “Along Vector” allows the user to specify a vector to always use when projecting the selected point onto the selected edge, while choosing “Interactive” prompts the user for a point, then an edge, then a projection vector.

Similarly, a “Projection Mode” drop-down has also been added for the “Project Curve” operation. Choosing “Normal” performs a normal projection of the selected curves onto the selected surfaces, while choosing “Along Vector” uses the user-defined vector for all projections of selected curves onto selected surfaces.



**Solver support****Simcenter Nastran/MSC Nastran****Multiple subcases for dynamic analysis**

Support for multiple subcases in dynamic analysis has been added for Nastran solvers, which allows multiple subcases to be defined for a transient response, frequency response, or nonlinear transient response analysis. Dynamic subcases can be used to control dynamic or nonlinear behavior subcase-to-subcase, while static subcases can be defined to account for stress stiffening and optionally impose contact conditions.

**CBUSH1D/PBUSH1D**

Import and export of CBUSH1D elements and corresponding PBUSH1D properties, which are used to model rod type spring-and-damper elements, is now available and includes support of all four optional behaviors available on the PBUSH1D. The option spring, damper, generalized spring and damper, and shock absorber behaviors can be controlled by either a function or a design equation. When using a design equation, a DEQATN entry will be exported.

**Simcenter Nastran****Connection property to use for gap elements**

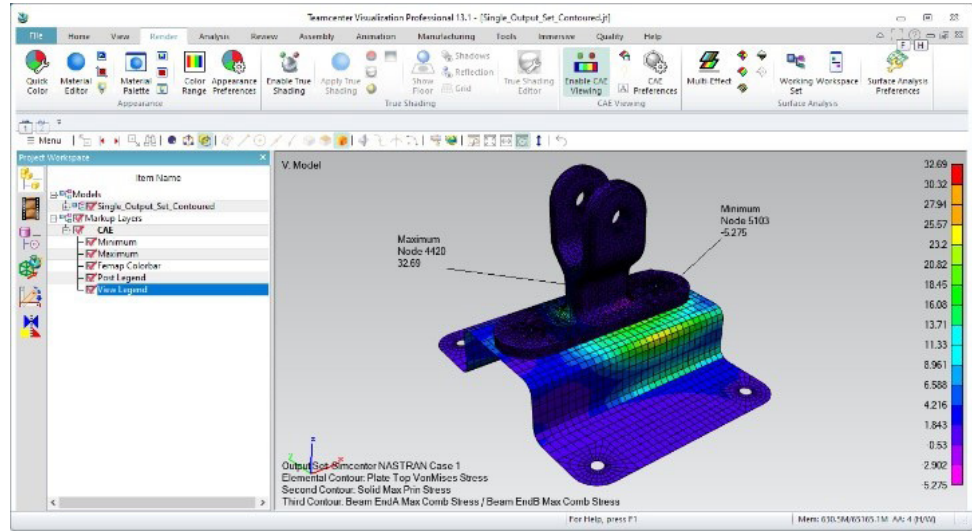
Selection of connection property to when the gaps as contact option is used when surface-to-surface contact is not specified in linear static analysis is now possible, which can control certain parameters including penalty factors in the normal and transverse direction.

**MSC Nastran****Pyramid element results**

Results from pyramid elements can now be imported from the .f06 file.

**LS-Dyna****Element formulations**

Element formulations EQ18 and EQ-18 for solid elements and element formulation EQ16 for shell elements are now supported.



## Postprocessing

### Stress linearization

This new feature copies the underlying transformed stresses to the clipboard instead of the regular output report by holding down the shift key when pressing the “Copy to Clipboard” button.

### JT visualization

JT visualization files can now be created that contain additional CAE entities or results from multiple output sets. These JT files can be used within the Teamcenter for Simulation (TC4Sim) environment or any other JT collaboration tool to visualize model data and review results without the need to open an instance of Femap.

## Customization via API

### Body mesher objects

This capability can be used to access the functionality of the mesh, bodies and mesh, and mesh on mesh commands programmatically.

### New functions

feFilePictureSaveJT and feFilePictureSaveJT-MultiResults are used to access functionality of the file, picture, save JT command.

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

Americas  
1 800 498 5351

Europe  
00 800 70002222

Asia-Pacific  
001 800 03061910

For additional numbers,  
click [here](#).