

# Simcenter Femap version 2019.1

## Improving simulation productivity

### Benefits

- Clearer model entity visualization and selection
- Streamlined workflows and re-use of previous data definitions
- Improved robustness for analysis application support

### Features

- Updates to UI and preprocessing operations
- Meshing enhancements including sweep and pyramid mesh transitions
- Extended support for nonlinear analysis solutions
- Enhanced results review options for partial results vectors

### Summary

Simcenter™ Femap™ software is now being released on a biannual schedule in April and October, beginning with version 2019.1. 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®.

Simcenter Femap 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 range from entity selection and geometry splitting operations to meshing, including new sweep and mesh transition

options. There are also several updates to solver integration for Nastran®, Ansys® and Abaqus®, and new results viewing selections in postprocessing.

### Visualization and user interface

#### Entity selection

Entity selection methods have been enhanced to allow you to select only from those entities that are visible on the screen at the time of selection, using the new “select visible” icon button on the entity selection dialog.

Also, elements can be selected based on a reference to an orientation node. These additions enhance the current array of available tools, making entity selection more versatile and flexible. You'll also notice that the ID fields in the entity selection and other dialog boxes throughout Simcenter Femap have been widened.

#### Previous vector

Simcenter Femap 2019.1 includes the ability to recall a previously defined vector in any operation that requires a vector direction definition to be created. Accessing previously defined data accelerates the model creation process, enhancing productivity.

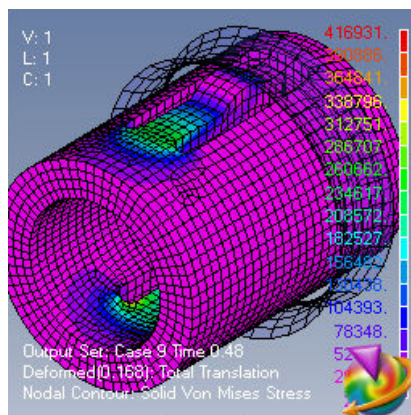
#### Locate in model info

A new “locate in model info” toggle is added as a new selector mode to the selector toolbar that highlights the selected entity (solid, CSYS, property, material or layout) in the model info tree. This makes it easier to locate the selected item in the model info tree when there are many items of that type in the model.

# Simcenter Femap version 2019.1

## Modfem preview and tooltips

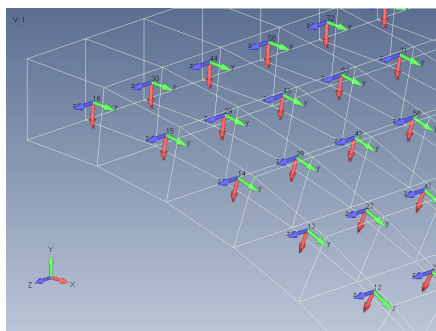
Even before you launch Femap, in the file explorer you can preview the modfem file model image, and via tooltips, review basic model statistics including version, size, save date and numbers of FE and geometry entities. This capability helps you locate and select the model you want earlier, without having to start up Femap first.



## Preprocessing enhancements

### Solid material coordinate systems

Simcenter Femap 2019.1 now supports the Nastran solid material coordinate definition system on elements, which allows each element to refer to a separate coordinate system ID. Support for reading and writing the solid element coordinate system ID entity (MATCID) is also added.



## Libraries

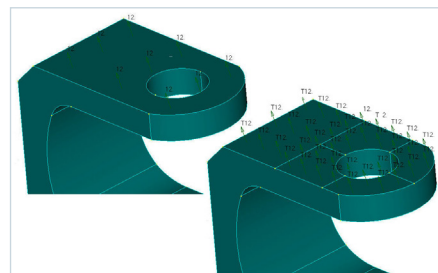
Version 2019.1 adds greater flexibility in the use of libraries to save and load modeling data including materials and properties. In addition to the Femap standard libraries, you can now create your own personal libraries in a directory of your choosing, instead of in the installation directory. Also, you can define a library that can be shared with other users – the shared library path is set within the file preferences dialog.

### LBC and solid data propagation with geometry splitting or slicing

Any existing geometry-based definitions of loads, constraints or regions will be automatically propagated during geometry split operations. Therefore, the original intent of the load and boundary condition (LBC) definition is maintained, which eliminates any need to rebuild the definitions after splitting, facilitating model setup.

For solid entities, all solid data is maintained after geometry slicing operations as well. All definitions of loads, constraints, regions, colors, layers and solid attributes are fully propagated.

These enhancements also add flexibility to the model setup workflows making these types of operations order-independent.



## Bearing force and torque on curve

New bearing force and torque on curve commands are now available that are similar to the bearing force or surface commands available previously.

## Point and node projection

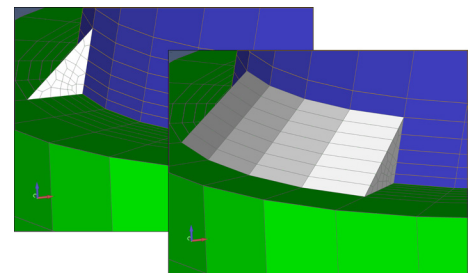
The point and node projection commands offer new options to project those entities radially about an axis, and spherically about a point.

## Meshing

### Sweep elements, element faces and edges

In the mesh sweep command there is a new option to sweep along element edges. The element edges form a path for the sweep operation which acts on a number of selected elements or element faces. Only a single element edge pick is required, and the sweep operation continues until it connects back on itself (around a circular structure for example) or an end is reached. There is also an option to specify a stop node. As the elements are swept, the mesh morphs to maintain a match with existing mesh edges.

In addition to sweeping plates and solid faces to create solids, you can now sweep element edges and beams to create 2D plates.

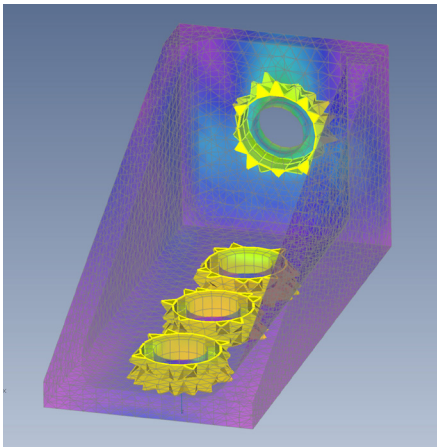


### Hexa to tetra pyramid element transitions

Simcenter Femap 2019.1 can now automatically generate pyramid elements to transition between hexa and tetra elements in a solid mesh. To facilitate this operation there are also new node matching selections and an option to select additional areas to create pyramid elements.

In addition to creating a mesh transition automatically, Simcenter Femap 2019.1 will also manage node connection between linear hexa elements and parabolic tetra elements by removing midside nodes where appropriate to ensure correct element connectivity.

Essentially, you can use pyramids to transition from any surface quad mesh to transition to a solid tetra mesh. Using this approach, it is also possible to define contained hex-meshable regions interior to a solid that would serve to reduce the number of degrees of freedom in the model.



### Mixed linear to parabolic element connection tool

Also available in Simcenter Femap 2019.1 is a tool to connect mixed linear and parabolic elements. The midside node on the parabolic element can be removed to connect the elements together. Alternatively, a midside node can be added to the linear element to make the connection. The tool works with shell elements and mixed meshes comprising beams, shells and solids.

### New element extrusion options

New options to widen the extrusion capabilities for shell elements have been added.

- Element extrusions along element normals to surfaces to create solid elements from the original plates to selected geometry surfaces
- Extrude shell elements in both directions at once to create solid elements on both sides of the original plates

### Solver support

Solver interface updates include extensions to Simcenter Nastran support for multi-step nonlinear solution sequences and enhancements to the design optimization capability. In addition, this release also includes further enhancements to the ANSYS and Abaqus solver translators.

### Simcenter Nastran enhancements

New subcase types have been introduced into SOL 401 and SOL 402 multi-step nonlinear solution sequences including nonlinear dynamics, linearized buckling and arc-length methods in statics. Bolt preloads can now be modeled using 1D beam elements, and there is now support for layered composite element properties PCOMP and PCOMPG. Also, various enhancements improve the robustness of contact modeling.

Several performance enhancements have also been completed including the following:

- Various numerical improvements to recursive domain Lanczos method (RDMODES) solutions
- Shared memory parallel (SMP) now extends to include element matrix assembly (EMA) module operations
- Automatic selection of frequency response numerical improvements

The Simcenter Femap 2019.1 release includes Simcenter Nastran 2019.1

### Nastran enhancements

SOL 200 design optimization now supports frequency response analysis with analysis setup similar to statics and modal.

The STATSUB case control entry is now supported that selects the static solution to be used in determining the differential stiffness for buckling analyses. Within the linear buckling solution sequence SOL 105, multiple subcases can be defined that include STATSUB entries, providing greater flexibility with analysis case setup.

### ANSYS translator

- Further support has been added to version 2019.1 to read and write non-normal pressure definitions, including SURF153, SURF154 and SURF156
- Beam preload (PRETS179) and solid preload (PSMESH) are now supported and written as ANSYS SLOAD
- For random vibration analysis, read/write support is added for various commands including SPOPT, PSDUNIT, PSDFRQ, PSDVAL, PFACT and PSDCOM
- The output control KEYOPT(8) for laminates and solid laminates has been added

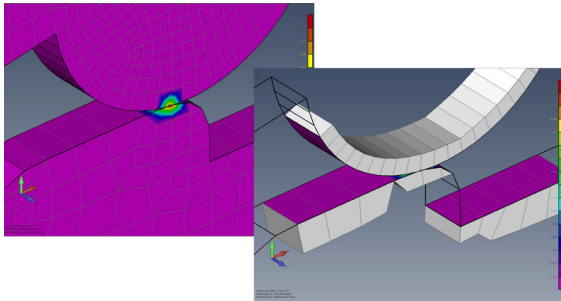
### Abaqus translator

You can now specify thermal options for gap elements and contact via the Abaqus thermal button, which allows you to edit a gap or connection property.

### Postprocessing

#### Contour plots

New contour plot options are available to view models with partial results, such as contact forces and stresses. Previously, all elements were contoured with a value of zero applied to all nodes without results.



### Listing formatted output

The previous standard and formatted output list commands have been consolidated into a single command in Femap 2019.1. When listing formatted output, you can select an output format from a personal shared or Femap standard library or select a custom format.

### Aero mesh displacements

Simcenter Femap 2019.1 now allows you to display aero mesh displacements from a flutter analysis.

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

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

Restricted © Siemens 2019. Siemens and the Siemens logo are registered trademarks of Siemens AG. Femap, HEEDS, Simcenter, Simcenter 3D, Simcenter Amesim, Simcenter FLOEFD, Simcenter Flomaster, Simcenter Flotherm, Simcenter MAGNET, Simcenter Motorsolve, Simcenter Samcef, Simcenter SCADAS, Simcenter STAR-CCM+, Simcenter Soundbrush, Simcenter Sound Camera, Simcenter Testlab, Simcenter Testxpress and STAR-CD are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries or affiliates in the United States and in other countries. All other trademarks, registered trademarks or service marks belong to their respective holders.  
 77951-C3 6/19 A