Femap automatic meshing simplifies virtual testing of even the toughest assignments

fact sheet

Siemens PLM Software

www.siemens.com/plm/femap

Summary

Femap® version 10 software is the latest release of the robust finite element modeling pre and postprocessor application known for its tight integration with the Nastran solver. Femap with NX™ Nastran® software, an extensive and reputable industry standard CAE modeler and solver, is part of the Siemens PLM Software Velocity Series™ portfolio. Femap v10 extends a more than 20-year history of productivity and functionality improvements in FEA modeling and postprocessing for engineers. The v10 release focuses on core simulation tasks, including preprocessing and meshing, and provides new tools to mesh complex structures more accurately.

Benefits

Improves accuracy of FEA results through higher quality meshes

Reduces modeling times with more intuitive workflows

Easier to use with more automated meshing and simulation processes

Optimizes modeling process with increased modeling fidelity only where it is needed to ensure efficient analyses and faster turnaround times

Features

True 3D meshers produce top quality meshes first time and are not constrained by the limitations of parametric meshers

Interactive mesh generation gives you greater control over mesh generation, and streamlines the whole meshing process

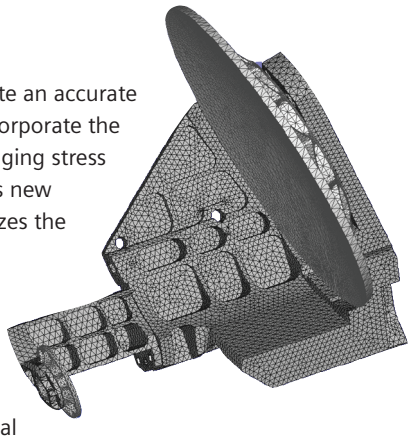
Live element quality feedback lets you see how good the mesh is as you create it, assuring you get answers as accurate as they can be

Geometry cleanup tools simplify the process of creating a viable FEA model from CAD data by cleaning up the geometry and preparing it for meshing

64-bit support allows large models to fit into available memory so you can preprocess and solve larger models in shorter times

Stress and displacement transformations can now be performed on-the-fly, adding versatility to postprocessing

Meshing improvements

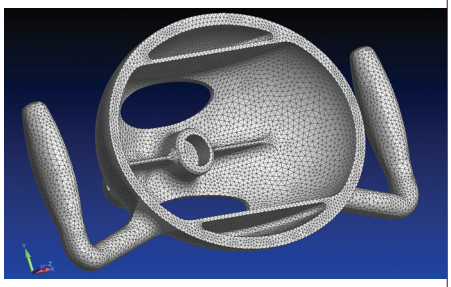


Central to the whole modeling process is the ability to create an accurate finite element mesh easily and quickly. Meshes need to incorporate the right level of detail in areas such as those with rapidly changing stress in order to calculate an accurate response. Femap v10 adds new solid and surface meshers and new meshing tools, centralizes the existing and new re-meshing functionality within the user interface improving workflows, and adds new interactive mesh checking tools.

For the occasional FEA user, engineers new to FEA or expert analysts looking for high quality first pass meshes, automated meshing that's easy to use and accurate is critical for successful product performance simulation. Femap v10's simplified and intuitive workflows and mesh modification tools enable you to create consistent and dependable simulation models every time.

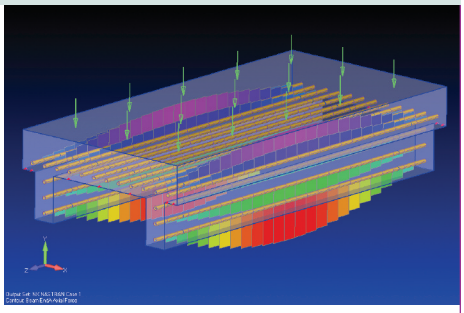
For analysts, control that assures accuracy is critical to success. Femap v10 provides the ability to interactively manipulate element meshes as well as maintain full control over the creation of complex meshes.

3D solid and surface meshers



New 3D solid and surface meshers have been introduced in Femap v10 that produce top quality meshes first time. The new meshers guarantee the creation of highest quality elements, even on curved surfaces, and drastically reduce the time taken to produce accurate finite element models.

The meshing function is performed in 3D space rather than parametric space eliminating element warping and creating a higher quality more consistent mesh with more evenly shaped, nondistorted elements. By working in 3D space, the meshers can deal with inconsistencies such as poles and other surface irregularities or mathematical imperfections – characteristics that are typically present in

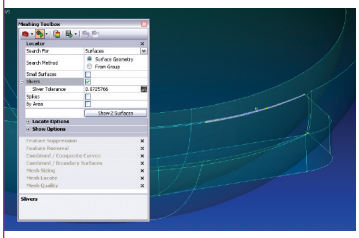


poorly parameterized geometry and problematic to a conventional parametric mesher. The new meshers can also deal with more complex boundary and nonplanar surfaces providing a consistent mesh that extends beyond planar topology limitations.

The solid mesher is also able to handle internal faces and branching faces and curves, creating a congruous, fully connected mesh internally.

Meshing Toolbox

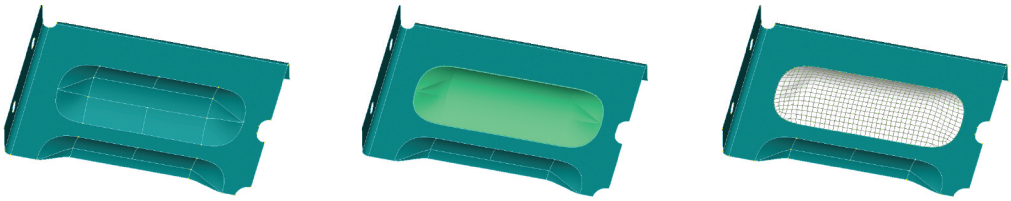
All of the existing and new **re-meshing** and geometry cleanup functionality now resides in a new Meshing Toolbox pane within the user interface. From here it's possible to access the consolidated new and existing **re-meshing** and mesh preparation functionality in one central area.

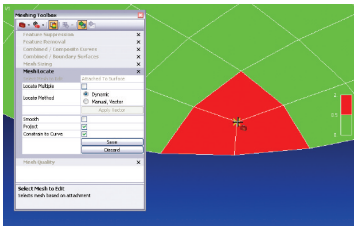


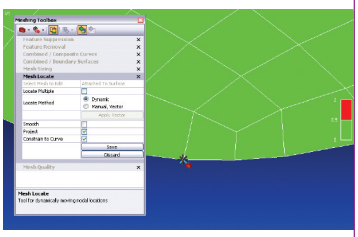
Locator. The entity locator can be used to find and identify small geometric features such as sliver surfaces, spikes and short edges – features that are likely to be problematic during the meshing process.

Feature suppression and removal. Here you can control the level of detail desired in the model by either suppressing or completely removing features. Feature suppression removes features temporarily and only affects the resulting mesh, not the geometry model. Feature removal actually deletes the features from the geometry model permanently. Features can also be removed from an already meshed model with immediate mesh update, so it's possible to interactively remove features and view the effect on the mesh live.

Combined curves and boundary surfaces. Here you can combine geometric curves or surface entities together to form larger entities within the model. By using these entity combination tools, you can overcome potential meshing problems that may arise due to the presence of awkward geometric topology, very small curves and sliver or spike surfaces. If the model has already been meshed, combining geometry entities can, if desired, cause the mesh to update automatically allowing you to see the results of the changes on the mesh immediately.





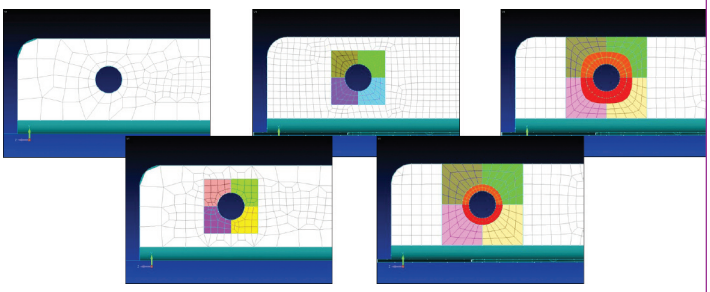


Mesh sizing and locate. The mesh sizing toolbox allows interactive changes to be made to the mesh sizing parameters on the geometry. With mesh locate you can manually move the nodes in the mesh with optional adherence to the underlying geometry. The ability to interactively update meshes removes all the manual steps required to improve the mesh – evaluation, deletion, modification of sizing update commands, re-meshing etc.

Mesh quality. Mesh quality is reported as you modify the mesh using the tools mentioned above with live element quality criteria plots, so you can immediately see the effect of your modifications to the mesh and verify element quality as you proceed. In the mesh quality toolbox you can select the quality checking method and setup defining parameters for the plots. Also, entities failing the mesh checking parameters can also be posted to the data table where you can easily highlight and investigate them.

Geometry subdivision tools

An important aspect of creating a good finite element model is the placement of well-shaped elements where they are needed at stress raisers where stress levels vary greatly over short distances. Femap v10 introduces the washer and pad geometry tools that essentially offset curves or split the geometric surfaces surrounding stress raisers like holes to allow for an improved mesh to be created that ensures solution accuracy. The pad transitions the mesh from the curved mesh around a hole to an imprinted orthogonal shape, and can allow for mapped meshing within the imprinted surfaces. The pad and washer can also be used in combination.



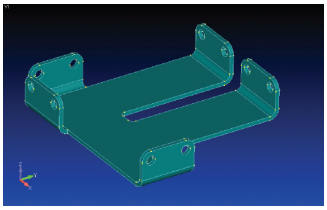
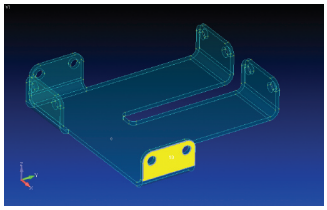
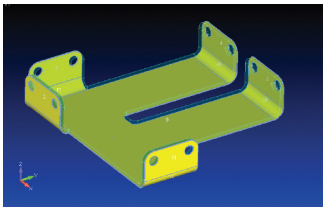
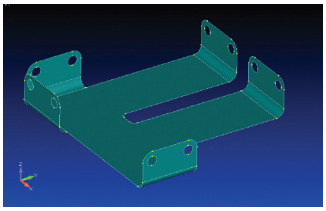
Often it's also necessary to break up the geometry model to create better defined surfaces and improve the geometric topology, or to ensure that surface entities line up to help generate a better mesh. Femap v10 introduces new face splitting tools that can be used to split up surfaces as required.

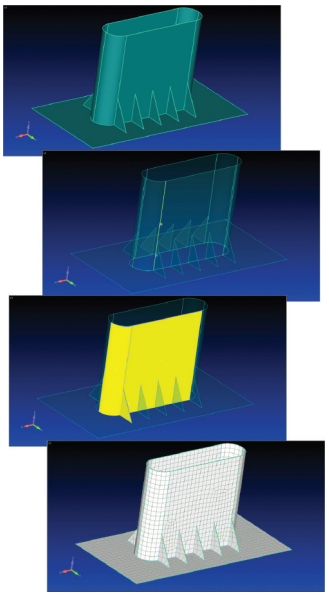
Non-manifold geometry

Geometry handling for shell modeling has been enhanced to support non-manifold geometry. This allows a contiguous mesh to be created over complex surface geometry intersections where multiple surfaces intersect, such as T-junctions. Also, contiguous meshes are created at intersections of surfaces with solids ensuring continuity throughout the mesh and correct connection between solid and surface elements inside the mesh.

New mid-surfacing method

A new mid-surfacing method has been added to Femap v10 designed for true sheet metal parts. By selecting one external face of the part, Femap will locate all tangent surfaces and offset them to the mid-plane position to create the mid-surface.



NX Nastran support enhancements

Femap has always had excellent integration with NX Nastran, and Femap v10 adds even more support for several new and existing capabilities, including:

- Axisymmetric elements (quads and trias)
- Solution monitor display in Femap so you can check solver progress during analysis
- CWELD extensions to the existing fastener and weld element
- CFAST: introduction of a new connection element that connects two shell element patches
- Linear contact extensions including output of contact separation distance
- Glued stiffness extensions that add more control over connection modeling
- Expanded model read functionality for legacy thermal and dynamics analyses decks

64-bit support

This latest release includes 32- and 64-bit versions of Femap. The availability of 64-bit Femap removes limitations on Femap's database memory cache, and enables extremely large FEA models to fit entirely into memory. Combined with the 64-bit capabilities of the solver products and with sufficient RAM, you can preprocess and solve larger models in shorter times.

"We knew that [the 64-bit version] would be better, but we are still amazed about it."

Mark Harrison, Engineering Manager, RPC Technologies Australia

Before: 32-bit, 1x dual core CPU, 4GB RAM
250,000 nodes and 1,450,746 degrees of freedom
Elapsed time: over 2.5 hours

After: 64-bit, 2x quad core CPU, 16GB RAM
250,000 nodes and 1,450,746 degrees of freedom
Elapsed time: 4.5 minutes

Stress transformations

An important aspect to finite element analysis is the ability to quickly evaluate the calculated stress and displacement distributions. To be effective, the displacements and stresses sometimes need to be evaluated in multiple coordinate systems, or in a single component of a particular coordinate system to make meaningful engineering decisions. Femap v10 adds new, on-the-fly transformation options for displacements and stresses. Users can quickly display just the radial component of a displacement, for example, in any cylindrical coordinate system. Displacement transformations can be a critical requirement in many machined parts or complex assemblies. Stresses can be transformed in a similar manner, making it possible to view hoop or longitudinal stresses, or a stress in any direction.

Graphics updates

- Geometric entities now use less memory and display faster
- Updated faceting gives higher quality images
- Faceting match between curves and surfaces gives higher quality images

Mesh/geometry association

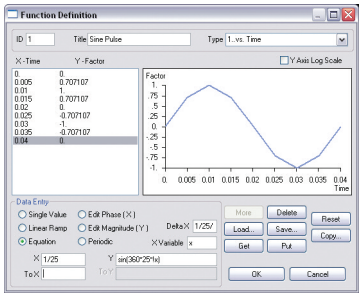
You can now associate existing meshes with geometry. Supply Femap with geometry and an orphaned mesh, and Femap automatically re-associates the mesh with the geometry:

- Nodes to points, curves and surfaces
- Elements to curves, surfaces and solids

New API methods

- AddOutput on data table object
- feElementFreeEdge and feElementFreeFace on the application object
- SelectID on all entity objects
- GetCentroid, GetEdgeNodes, GetFaceNodes, and IsParabolic on the element object

Other features



- Function definition graphs allow you to view the function as you are defining it
- Cross section dimension comments: beam/bar cross section dimensions are written as comments in the Nastran read/write
- File open detection: Femap can detect whether a file is open and locked by another application during a read operation, and prompts you to either retry or cancel the read
- Nastran single-field format precision improvement: Femap has a new preference allowing you to select either the improved precision or the previous standard numerical format approach
- Direct access to SolidWorks parts and assemblies
- Quad pattern: Quad_4Tria pattern mesh is available in mesh editing
- Highlighting: Selecting highlighting in the Model Tree automatically highlights entities you have already selected
- API extend: Enhancements continue to the powerful API extend methods and functionality
- Full Windows Vista support

For more information, contact your local Velocity Series™ portfolio representative:

Contact

Siemens PLM Software – www.siemens.com/plm/femap

Americas 800 807 2200
Europe 44 (0) 1202 243455
Asia-Pacific 852 2230 3308

SIEMENS