SIEMENS

Siemens PLM Software

What's new in Femap 11.1

Enhancing the productivity of finite element analysts

Benefits
- Faster model manipulation with improved graphics performance
- Greater flexibility for creating models and extending geometry modeling methods
- Easier FE model creation and control
- Faster and more flexible postprocessing
- Extended analysis discipline with enhanced NX Nastran integration

Summary
Femap™ software version 11.1 from Siemens PLM Software is the latest release of the standalone finite element modeling (FEM) pre- and postprocessor for engineering simulation and analysis. Femap can be used in combination with a wide variety of finite element (FE) analysis solvers, including the industry-leading NX™ Nastran® software.
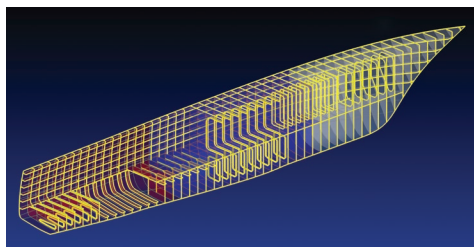
Femap 11.1 improves user productivity through an extension of the results file attach capability that became available in a prior release. Graphics performance has also been improved significantly, particularly through more efficient graphics storage and support of OpenGL 4.2 geometry shader functionality. In addition, there are enhancements that improve geometry creation and manipulation as well as meshing and FE model management.

Version 11.1 extends the scope of simulation with extensions to the design optimization capability. Also, there are several enhancements that strengthen Nastran integration, including support for pyramid elements, dynamic response output and restarts. In addition, there are many other customer-driven enhancements included in the latest version of Femap.

Geometry modeling enhancements

Non-manifold add
The non-manifold add command added in the previous release of Femap introduced a method of Boolean adding sheet solids together to form a general body. In Femap version 11.1, this capability has been extended to allow user control of a distance or separation tolerance facilitating the Boolean add process.

Non-manifold add multiple bodies
Imported geometry often contains discontinuities, which in turn will create a discontiguous mesh if not corrected. In Femap 11.1, you can use the non-manifold

www.siemens.com/plm/femap

What's new in Femap 11.1

Features

- Use of graphics memory and OpenGL 4.x functionality supports improved graphical data access and graphics performance

- Extended geometry creation methods

- Enhanced FE model merging and meshing capabilities

- Extended results data file attachment capability, charting and free body capabilities

- Greater support for design optimization and restarts in NX Nastran

add command in the geometry/surfaces menu to add multiple bodies together and create a single general body. The algorithm starts with the first and adds each next closest body one at a time controlled by a tolerance. Incremental checking validates the resulting Parasolid body after each step; if it doesn't meet specifications, it rolls back and skips that particular solid. When it finishes, the free edges are high-lighted so you can see what has been connected.

Solid sweep

New solid sweep and solid sweep between commands have been added in Femap 11.1 to create solid geometry. With the solid sweep command, it's now possible to sweep a surface along a set of curves to create a new solid. The solid sweep between command can be used to create a new solid between surfaces or faces of existing solids and can effectively create new solid geometry to fill a gap in the model.
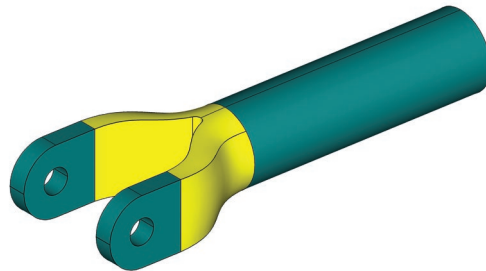
Midsurface extraction

The midsurface extraction capability has been enhanced to create a much more robust method of determining midsurfaces in thin solid models by taking advantage of recent developments in the Parasolid modeling kernel.

Surface from mesh

The surface from mesh command intro-duced in Femap 11.0 has been enhanced to create more robust surface geometry for ruled surfaces.

IGES export

Export of IGES geometry has been extended to include points and curves.

FE modeling enhancements

Model merge

The model merge capability that allows entities to be copied between models when multiple models are open in Femap has been enhanced for version 11.1. You now have complete control over the selection of entity types to be merged as well as entity renumbering. You also have control over the grouping and model transformation and orientation. You can transfer frequently used materials, properties and layups from existing models to new models.
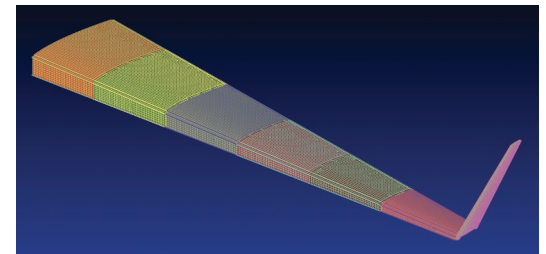
Tetrahedral meshing

The solid tetrahedral auto meshing capabil-ity has been enhanced to force multiple elements through a thickness to ensure that the model will yield more accurate results.

Femap 11.1 uses the latest version of the tetra-mesher, which creates a mesh in which most badly deformed sliver tetrahedrons are eliminated.

Meshing toolbox

New controls have been added to help orientate and align the pad (including pad and washer combination) detailed meshing feature in the meshing toolbox.
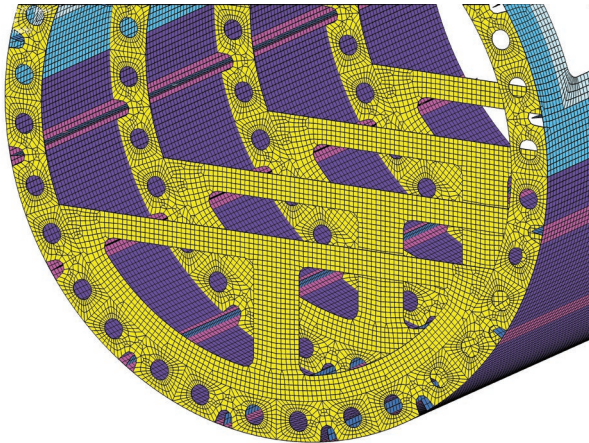
Mesh offset

You can now copy and offset a mesh along element normals to create identical meshes offset a specified distance away from the original mesh.

Graphics enhancements

Graphics architecture

Graphics performance in Femap version 11.1 is improved through more efficient storage with a single copy of graphics entity data existing within a model. This results in significant improvements in model regeneration through the <Ctrl-G> command and dynamic rotation.

Further graphics performance improvements come from the added support of OpenGL 4.2 geometry shader functionality. Essentially, this moves the workload from the central processing unit (CPU) to the graphics processing unit (GPU), taking advantage of local graphics memory, and dramatically reducing the volume of data required.

Femap version 11.1 can be used to take advantage of OpenGL 4.x graphics, which are much faster and use significantly less graphics memory.

Visibility

The underlying code for the visibility dialog has been updated to speed up population of the property, material and group selection boxes.

Load expansion

Load expansion from geometry to a mesh has been enhanced for elemental face loads on surfaces such as pressure, heat flux, convection and radiation. Load expansion times for large models with many loaded surfaces are reduced from minutes to seconds.

Postprocessing

Results files

A more efficient method of accessing results data was introduced in the previous release of Femap, and has been extended with version 11.1 to include XDB format results files commonly created by using NX and MSC Nastran solvers. Support also extends to user-created comma separated variable (CSV) files, which allow significantly enhanced user set control.

You can now also export results data to a binary .FNO format and include the model data or a reduced model of just the area of interest. This can act as an aid to help demonstrate and communicate the analysis results.

Charting

The Femap charting capability now includes a new data series type, value versus value, which allows plotting of quantities, such as applied force versus displacement at a specified location. The chart data series dialog has been reorganized to display only relevant items, and the user now has more control over fonts, labels and colors for chart entities. Also, rendering has been improved when copying charts to the clipboard at non-screen resolutions.

Free body

The free body capability has been enhanced in Femap 11.1 as the user has the added ability to force free body calculations to consider only applied, constraint and multipoint constraint forces from Nastran's GPFORCE data block. There is also a new validation tool to check that all requested values exist in the result set, and the free body listing tools have been updated to provide more relevant output.

NX Nastran support

Pyramid element

Femap 11.1 supports the pyramid element for read or write operations. Models containing existing pyramid elements can be imported and exported, but element creation remains a manual task.

Restarts

Support of Nastran's restart technology has been enhanced in Femap 11.1. You can opt to perform read only restarts, and specify the model version and subcase identification (ID) from which to restart.

Design optimization

Femap 11.1 includes support of design optimization with normal modes analysis so designs can be optimized to modify or avoid certain critical natural vibration frequencies. You can specify a mode frequency or an eigenvalue to be set up as a design constraint with an option for mode tracking.

Dynamic response output

You can now request RMS Von Mises stress output directly for random vibration analyses. Composite ply results are available for transient response so you can get stress/strain output for individual laminae.

Load and constraint set combination display

In Femap 11.1, you can visualize load set and constraint set combinations. When you set up a new load case, such as a Nastran LOAD combination, or a new constraint case, such as a Nastran SPCADD/MPCADD combination, the combined loading or constraint sets can be displayed.

Customer-driven enhancements grouping

The group generation command has been enhanced to create groups according to geometry T-junctions through the non-manifold edge functionality. This allows easy break up of multiple geometry features into groups facilitating model management.

Modify/project

Some changes have been made to the way points and nodes are projected in Femap version 11.1, standardizing the method of projection and allowing selection of multiple entities to project on to as well as projection along a vector and to the closest entities.

User interface

The tooltip display now optionally displays group and layer information appearing in both the tooltip and entity information pane to aid model organization and identification.

Create line or measure between geometry

You can measure the minimum, maximum or both distances between multiple selected geometry entities. In a related capability, you can create a line between the maximum, minimum (or both) points between multiple selected geometry entities.

Load set combination data surface

Femap 11.1 allows spreadsheet access to the combination of existing load sets to create new linearly combined load sets. Resulting load sets can be used to create actual scaled and combined individual loads, or create a virtual NX Nastran LOAD CARD combination. You can cut and paste the entire load set combination data table to and from Excel. Any updates to the load combination table can either update previously created combinations, or create new ones. While editing the load set combination table, you'll have continual right click access to the load summations for the constituent sets and/or the new combined sets.

Tools check extensions

A new option has been added to coincident nodes check that allows you to merge nodes with differing output coordinate systems.

Options to sum forces checking have expanded to sum all loads, loads on selected nodes and elements and selected load definitions. Body loads can optionally be included in the load summation as well.

The element quality checking dialog includes new all-on and all-off options to aid quality check selection. There is a show button that will highlight elements that fail any of the checked quality check options. Some listing options are also available for quality check details and a summary.