

Femap visualization and Nastran integration aids modeling and analysis definition

fact sheet

Siemens PLM Software

www.siemens.com/plm/femap

► Summary

Femap® version 10.1 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 a part of the Siemens PLM Software Velocity Series™ portfolio. Femap v10.1 extends a more than 20-year history of productivity and functionality improvements in FEA modeling and postprocessing for engineers. The v10.1 release focuses on modeling visualization and NX Nastran solver integration to help finite element modeling and analysis setup.

Benefits

- Improves productivity with flexible and interactive model display and grouping
- Speeds up analysis model definition with improved analysis setup workflows
- Easier to postprocess composite structural models
- Easier to manage combination load and constraint sets

Features

- Interactive multi-group display with combination groups
- Interactive element and FE model entity display
- Enhanced analysis manager and internal job queuing facility
- Enhanced load and constraint set managers and combination sets
- Support for global composite ply
- Support for 2D tensor plot
- Load and constraint enhancements including new torque load option

Model visualization

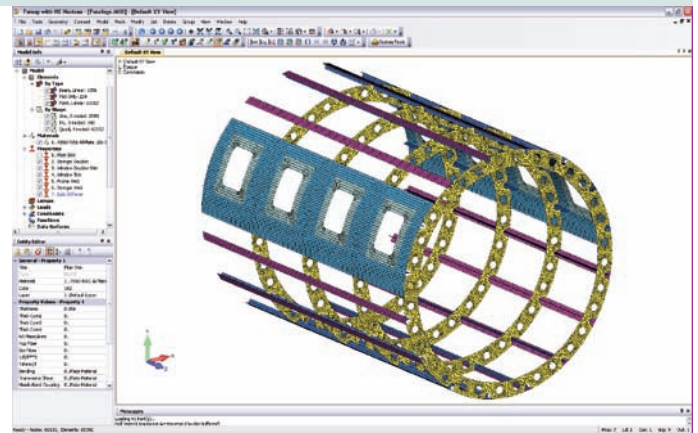
Model visualization management and control has been extended in Femap 10.1 with the support of multiple group and combination group display. The model information tree in the Femap user interface provides the basis for interactive control through new check boxes, which update the graphics display of the model instantly. This way you can easily control the entities that are viewed on the screen, adding or removing entities and creating combination groups on the fly as required.

Further enhancements to the model information tree add new element display controls with new check boxes that interactively switch on element display by shape or type as well as by association with materials and properties. The graphics display can be interactively controlled without having to create any groups or layers. This can also be combined with the extended group display functionality to add even more control over entity display.

Besides the model information tree, access to these new display controls, along with all of the existing visualization controls, has been consolidated into one dialog box – the visibility dialog box – giving a single point of access to all the visualization and display options.

NX Nastran support enhancements

Femap 10.1 brings new functionality that can be used to interactively define combination load sets. These are load sets that combine multiple loading conditions with a possible scale factor setting, and conform to standard Nastran definitions. For existing Nastran analysis files, it is also possible to read in the load case definitions, including all combination load sets into Femap, which means that they don't have to be manually rebuilt inside the program.



In the Femap analysis manager, support for analysis control and setup has been extended to encompass more advanced analysis types including dynamics and nonlinear. Also, existing Nastran input files can be read in and all advanced analysis setup definitions will be preserved in Femap.

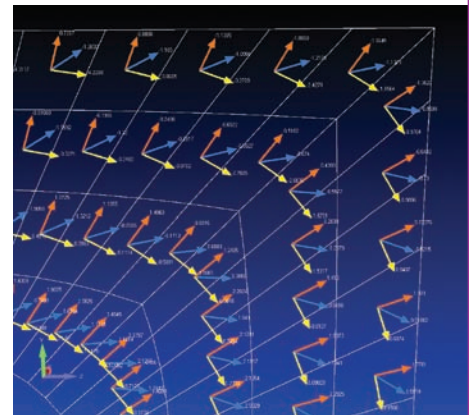
Femap 10.1 makes the task of job submission easier with the expanded job queuing functionality. Multiple jobs that can come from multiple models within Femap can be submitted to be run. Femap will set each to run in turn on the local machine and provide status feedback. As each one finishes, Femap ensures that the results are loaded into the relevant model file ready for postprocessing. Using the job queuing functionality, you can submit multiple jobs to be run and leave Femap to manage each one, and the associated results data accordingly.

Graphics updates

Femap 10.1 also sees some improvements to the graphics database for geometry-intensive models. Solid geometry facets are removed from the database during a save, so for models with many geometry facets, you will see a significant reduction in the model file size needed to store the model. However, while this might lead to a slight increase in model load times, save times will also be reduced.

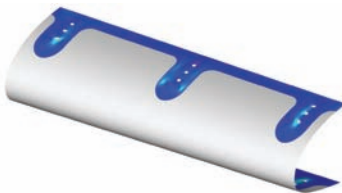
2D tensor plot

In the area of postprocessing, Femap 10.1 introduces a new 2D tensor plot. These plots can be used to display positive X and Y scalar values as well as the direction of positive shear flow. Typically 2D tensor plots of forces are used in the aerospace industry to display running loads through the structure.



Graphics display

Femap 10.1 adds new graphics to represent body loads such as gravity or rotational acceleration if they exist in the model, so that they are no longer hidden. Graphical representation of these loads can be by the model axes, or for rotational acceleration loads around the axis of application.



Composite global ply

In any composite structure it is likely that there will be areas of ply drop-off as the composite laminate changes thickness. In a finite element analysis model of a composite structure, this means that adjacent elements may contain differing numbers of plies, as the composition of the laminate changes through the structure. This can make the task of postprocessing composite structures laborious and difficult.

Femap 10.1 introduces support for global composite plies. For a composite structure, continuous plies can be determined across different elements and continuous results data displayed. This allows results for the same ply to be viewed across the finite element model, making the task of postprocessing the results data much more straightforward.



Femap performs this calculation independently of any global ply solver computation, such as PCOMPG in Nastran. As such, Femap can be used to view global plies for any composite structure, and results calculated by any solver.

Loads and constraints

With existing load and constraint definitions, it is now possible to update and change the position of the load or constraint. This makes it much easier to work with existing loads and constraints making any necessary updates without having to go back and create them from scratch.

Also, Femap 10.1 introduces a new torque load definition for surfaces, allowing a torque load to be applied directly to geometry.

Some Nastran-specific load and constraint sets have been renamed in Femap to conform to the Nastran naming convention.

Femap 10.1 now supports through thickness gradients for plate elemental temperature loads.

For loads that are applied to more than one geometry entity, the user is now given a choice to allow either the full load definition to be applied to each geometry entity individually, or dispersed over all geometry entities, with the new total load option.

► For more information, contact your local Velocity representative:

► **Contact**
Siemens PLM Software
Americas 800 807 2200
Europe 44 (0) 1202 243455
Asia-Pacific 852 2230 3308
www.siemens.com/plm

SIEMENS