SIEMENS

Siemens PLM Software

Femap version 11

Benefits

- Significant time savings with results data management and postprocessing
- Faster model manipulation with improved graphics performance
- Easier confidential transfer of model structural data using the external superelement method
- Increased productivity when updating or remeshing legacy shell element models
- Better results comprehension through improved XY plotting

Summary

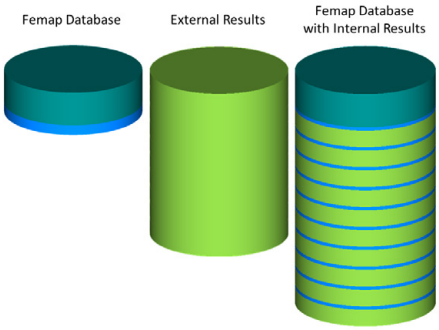
Femap™ software version 11 from Siemens PLM Software is the latest release of the robust finite element modeling pre- and postprocessor for engineering simulation and analysis. Femap works in combination with a wide variety of finite element analysis solvers, including the industry-leading NX™ Nastran® application, also from Siemens PLM Software.

The latest release of Femap, version 11, helps improve your productivity through more efficient postprocessing by allowing separate external results files to attach to the Femap database for data access. Graphics performance has also been improved significantly particularly when displaying and rotating large models. Preprocessing now includes the ability to create surface geometry from legacy shell element models facilitating modeling updates and changes. Postprocessing now includes a streamlined XY plotting capability that expedites results review and comprehension of model behavior.

Version 11 extends the scope of simulation with support for external superelement creation and subsequent assembly analysis that allow suppliers to transfer model data confidentially. Femap 11 ships with NX Nastran 8.5 and brings numerous enhancements that serve to strengthen NX Nastran integration. In addition, there are many other customer-driven enhancements included in this latest version of Femap.

External results files

NX Nastran, MSC Nastran OP2 and NEi Nastran FNO results files can now be attached to the Femap database and accessed directly rather than reading and importing the results data into the database in a separate step post analysis. With externally attached data files, data access is



Femap version 11

Features

- External results data file attachment to the Femap database
- Use of graphics memory that increases performance of graphical data access and display
- Support for external superelement creation and assembly analysis
- Surface geometry creation from a shell mesh
- Streamlined XY plotting display

much faster particularly for analysis types such as nonlinear or transient response that generate incremental results steps. Consequently, productivity for post-processing tasks is significantly improved. Also, multiple results files can be attached to the Femap database. Maintaining external results data in this way minimizes the Femap database size and makes results data handing much more manageable.

Graphics enhancements

Graphics performance in Femap version 11 is significantly improved for dynamic rotation of large models. Femap now uses more of the graphics memory to increase performance for graphical access and display. This is achieved through the support of Vertex Buffer Objects (VBOs). VBO is an OpenGL feature that allows vertex data to be uploaded to the graphics card. The vertex data can therefore reside on the graphics card memory rather than the system memory, and can be rendered directly by the graphics card.

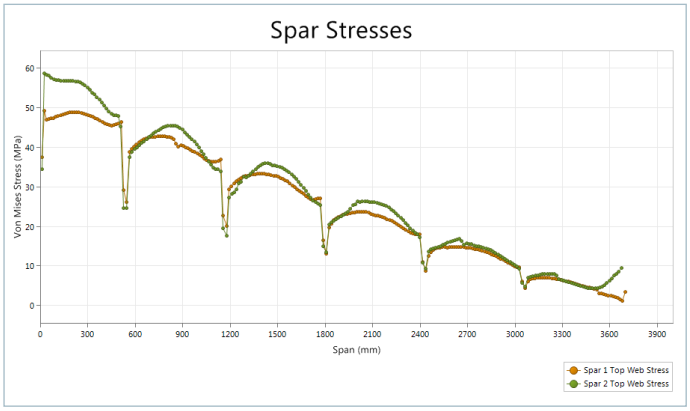
Another graphic enhancement is up to an order of magnitude performance improvement in the dynamic rotation of large models that can be realized depending on the graphics card, the amount of graphics memory and he model.

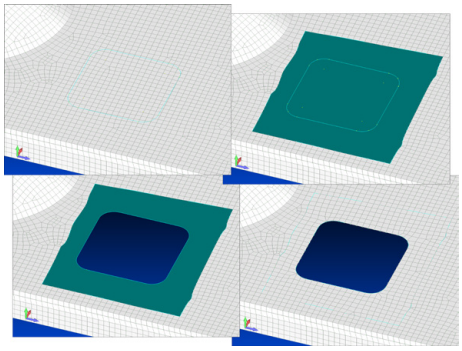
External superelements

Nastran superelements can be used to substructure a model for various reasons. External superelements, an extension of superelement technology, provide analysts or suppliers a method of transferring structural data in matrix form only to maintain confidentiality. Components of a model can be passed from supplier to OEM, for example, in matrix form and assembled to build the complete model. However, because the model data for the external superelements exists only in matrix form, no FE model detail or geometry is included, thus maintaining complete model confidentiality. In Femap version 11, the external superelement creation and subsequent assembly analysis are now supported with all the requisite Nastran file management and case control entities created automatically. This makes the process of creating and assembling external superelements less prone to error and much easier to manage. Note that superelement assembly analysis requires a superelement software license, however external superelement creation does not.

Geometry from mesh

The task of having to modify legacy finite element models, for which there is no geometry available, can be difficult and time consuming. Femap now delivers a method that helps you create surface geometry straight from an existing shell mesh, allowing existing FE models to be modified, updated or remeshed much more easily. You can take an existing shell model and convert it into surface geometry, then remesh to create a new shell model that is topologically identical to the initial model.





XY Plotting

In Femap version 11, the XY plotting capability has been streamlined, facilitating XY plot creation to aid results data review and comprehension of model behavior post analysis. Femap functions can also be viewed as well. The graphics have been extended, giving you complete control over colors, labels, formatting, etc. You can create multiple charts and save them to the Femap database, and multiple curves can be displayed per chart. Furthermore, all of the charting setup and formatting capabilities are accessible in context with right-mouse-button clicking, through the menus or via icons on the top of the charting pane.

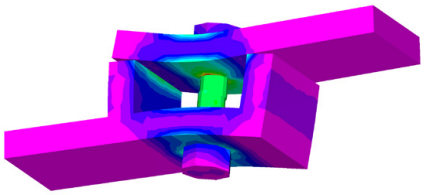
The XY plots are based on series of data taken from the analysis results, and can be filtered down to particular vectors or incremental steps for nonlinear, transient or frequency results and displayed for regions or subcomponents of the model. You can set up different groups for different data series, and for incremental data, you can set the desired beginning and ending output set. Also, once the charts have been set up it's easy for you to copy and paste them into a Word document to help create a report of the analysis results.

NX Nastran support

Femap 11 includes version 8.5 of the NX Nastran solver, and supports several new NX Nastran capabilities, including:

Solid composite element Composite structures can be represented by solid hexa and penta elements that reference the PCOMPS physical property and the MATFT definition for failure theory material constants.

Bolt preloads for solid elements
Preloaded bolts can be represented by solid hexa and penta elements to provide more detailed stress recovery in and around the bolt locations.



Nonstructural mass regions
Nonstructural mass regions allow a mass definition (based on NSM entities) that is independent of the structural model and can be defined on geometry (curves or surfaces) or elements (beams and shells).

Material angle definition by material coordinate system ID The material angle can now be defined using a material coordinate system ID, making it easier to specify a material angle across many elements regardless of element orientation.

Acceleration load definition Static acceleration loads can be defined using the ACCEL entity, allowing different acceleration loads to be applied over different parts of the model.

Customer-driven updates

In addition to the main enhancements mentioned above, there are also numerous customer-driven enhancements in the Femap 11 release, including:

Mesh edge split Femap 11 introduces a new method of splitting a mesh with the mesh edge split capability. Simply identify two nodes on an element edge and the mesh will split propagating through the model splitting 1D, 2D and 3D hexa and penta elements. You can also identify certain nodes to limit the split region.

Multiple output vector display
Results on multiple element types (line, shell and solid) can be displayed simultaneously. Also, results on different element types with the same topology such as beam, bar or rod can be displayed in one plot.

Postprocessing Data dialog updates
Filters have been added to the postprocessing data dialog to facilitate output set and vector data selection.

Database cleanup You can clean up a Femap database during a save operation to release any unused space and reduce the overall database size.

Add connected element selection
A new selection method for elements has been added to the entity selection dialog that allows you to select picked elements and all elements to which they connect.

Contact
Siemens PLM Software
Americas +1 314-264-8287
Europe +44 (0) 1276 413200
Asia-Pacific +852 2230 3308

www.siemens.com/plm

32348-Y3 8/14 A