SIEMENS

Femap Version 11.2

Benefits
- Faster results comprehension
- More intuitive geometry editing
- Easier contact model setup and visualization
- More versatile meshing functions
- Faster graphics display with improved graphics performance

Features
- Enhanced postprocessing capabilities including analysis studies and free body section cuts
- Direct interactive geometry creation methods
- Contact manager data surface

Summary
Femap™ software version 11.2 is the latest release of the standalone 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® software.

Femap 11.2 provides extensions to the postprocessing capabilities of Femap including analysis studies and free body section cuts that speed up results comprehension. There are new geometry modification tools that allow faster direct interactive geometry editing, and extended preprocessing functionality including a contact manager data surface that greatly facilitates contact model setup and visualization. Meshing enhancements include improved control of through-thickness elements for solid meshes and mesh splitting for increased meshing versatility. In addition, there are significant graphics performance improvements and many more miscellaneous and solver-specific enhancements.

Postprocessing enhancements
Analysis studies
With the new analysis study manager, new analysis studies can be set up to help organize output into more orderly and understandable data sets. You can group output sets from different analysis types into separate analysis studies and operate on them collectively.

Results set processing data surface
Selected output sets are placed into the data surface editor where an operation is chosen and individual scale factors entered for any or all of the output sets. One command is then used to create or update all of the processed output sets defined in the data surface.

Use of analysis studies with the new results set processing data surface lets you postprocess results data on-the-fly, enabling you to:

- Envelop analysis studies to collect output maxima and minima across collections of output sets
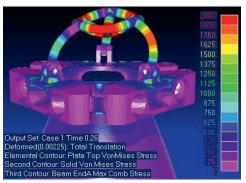
www.siemens.com/plm/femap

Femap Version 11.2

Features *continued*
- Improved meshing tools including through-thickness meshing, mesh splitting and washer and pad extensions
- Increased use of graphics memory for improved graphics performance

- Easily create factored linear and RSS combinations of datasets

- Determine whether the extra output calculated is stored temporarily or saved permanently to the database

Element contour plots
You can view solid, shell and line element results together in the same contour plot using a multiple contour vector option, which helps postprocessing and visualization of models containing multiple element types. You can identify three independent vectors to contour the differing 1D, 2D and 3D element topologies separately.

Output Set: Case 1 Time 0.26
Deformed(0.00225): Total Translation
Elemental Contour: Plate Top VonMises Stress
Second Contour: Solid Von Mises Stress
Third Contour: Beam EndA Max Comb Stress

Free body section cuts
Free body section cuts can be used to gain a faster understanding of load paths through the model. The section cut is defined using a cutting plane where the free body total/nodal summation is calculated on-the-fly and displayed for all the contributing nodes and elements. You can move the free body section interactively along a predetermined path through the model using a slider in the postprocessing toolbox.

Complex results
You can now postprocess complex results on-the-fly using the synchronize phase option to set a display phase angle for contours and/or a phase increment for animation. These settings can be applied to either the whole model or a particular view.
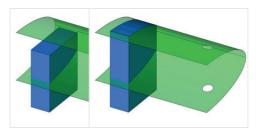
Linear and nonlinear results
Expanded use of vector IDs in Femap allows both linear and nonlinear results to be recovered within the same output set.

Geometry tools
Geometry modification tools
Femap version 11.2 offers more control over meshing around potential stress raisers in the model, such as holes or slots, with more robust and enhanced geometry modification tools based on offset curves, including the washer, pad and combined washer/pad options. The offset size can be specified as a factor of the original selected curves or as an offset distance.

This release also introduces new geometry editing tools that enable you to modify geometry entities interactively and perform operations such as deleting edges and moving points, curves and surfaces. Also some new mid-surface extend tools have been introduced such as curve-to-surface joining that enable you to connect geometry following mid-surface extraction operations much faster and more easily. You can more easily repair or correct gaps in geometry by moving points to other points interactively.
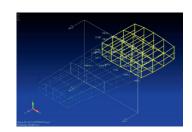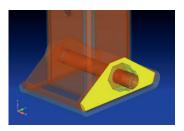

Smart snap geometry picking
Version 11.2 introduces a new smart snap geometry picking method that facilitates geometry selection and location. Femap can automatically determine the closest end point, midpoint or arc center in the process of selecting geometry. The smart snap method is accessible through the Select toolbar or Quick Access menu and can be set as a preference.

Geometry solid slice
The geometry solid slice commands available in previous versions of Femap have been consolidated to provide simpler execution and expanded functionality, including parallel planes options, as well as an option for repeated slicing. Also, you can now slice solids using surface or curve

geometry as cutting tools, optionally retaining or deleting the tools after command execution.

Preprocessing
Contact manager data surface
A new contact manager data surface greatly facilitates the setup of contact between components in a solid model. Show, hide and highlight options give you improved visibility of contact surfaces in the model, and you have direct access to the contact property definitions.

Element support enhancements
Enhancements to element and property definitions include:

- More efficient definitions of tapered plate structures using shell elements – corner thickness definitions can now be defined on the connectivity entity (for example CQUAD4) instead of the property definition (for example PSHELL) so that many tapered elements can reference a single property definition.

- More efficient spring/damper coordinate system definitions – CBUSH coordinate systems can now be defined directly on the element definition instead of the property definition so that many spring damper elements with differing coordinate systems can now reference a single property record.

- Constraint equations are now defined using an enhanced dialog offering several creation methods, interactive editing, and bulk update of coefficients and/or degrees of freedom (DOF). Also, up to 6,000 terms can now be defined for a single constraint equation allowing for more complex constraint definitions.

Meshing
Multiple tet through thickness
You can increase the accuracy of solid tetrahedral models with the ability to control the number of though-thickness elements when meshing the models with tetrahedral elements. You can define up to ten element layers through the solid thickness.

Mesh splitting
You can now define the number of splits for the edge split command to split an element edge into multiple element edges. There is also a bias control to bias the element widths across the edge that is being split. New elements retain geometric associativity of the original mesh. All solid, planar and line elements can be split in this way, and mesh-based loads and boundary elements are distributed appropriately.

Creation and modification of rigid elements
Edits to rigid elements (RBE2 and RBE3) are now easier to achieve with improvements to rigid element connectivity definition, notably the ability to simply click on an unconnected node to connect it to the element or similarly click on a connected node to remove it. The mesh connect rigid tool can be used to create rigid or interpolation elements by selecting any number of source nodes for which target nodes are automatically identified based on specified criteria.

Mesh quality checking
You can now check the model according to the NX Nastran element quality checking definitions to ensure that the model will not fail due to poor element quality before submitting for analysis.

Graphics enhancements
Graphics performance
Building upon performance improvements in Femap version 11.1, graphics

performance continues to increase in version 11.2 by taking advantage of graphics card hardware resources. Graphic representation support has been expanded to include coordinate systems, solid element material definitions, point and line elements, and nodal constraints. An example model containing over 1 million elements, nodes and nodal constraints (with labels displayed) sees model refresh (Ctrl+G) times reduce by a factor of 10, from 15 seconds in regular OpenGL® graphics to 1.5 seconds. Similarly, dynamic rotation response increases by a factor of over 80, from an average frame rate of .067 frames per second using regular OpenGL graphics to an average frame rate of 5.366 frames per second.

Solver support
Femap 11.2 ships with NX Nastran 10 and extends support of NX Nastran and other major solvers to provide closer integration between the pre- and postprocessing and solution components.

NX Nastran and Nastran
- New output for frequency response solutions now includes laminate ply-by-ply stress/strain, failure indices, strength ratios and von Mises stress

- Support for PCOMPG and bolt preload output is now available in advanced nonlinear solutions

- You can now import or attach strain results for beam elements form the .op2 file

- Support has been added to import or attach the DDAM summary results from the .op2 file

- Added read/write support for ACCEL1 entries which can now be created in Femap by creating an acceleration load and exporting a static analysis

- Added read/write of Femap comments as titles for connection regions, connection properties, and connectors

MSC Nastran contact
A new tab has been added to the define connection property dialog to support definition of both linear and glued contact for the MSC Nastran™ solver. MSC Nastran users can also take advantage of new options found in the analysis set manager to specify contact solver parameters, such as contact method, friction, and separation control. Additional options are available to control which connectors will be included and how they will be used in the contact table.

Other solvers
LS-DYNA: Proper definition and export is now available for many LS-DYNA material types

ANSYS: Reading and writing of linear and parabolic pyramid elements is now supported

Miscellaneous enhancements
Connection regions
Performance of connection region expansion has increased – in certain cases a performance increase of 135 times has been recorded.

Geometry
- You can now delete surfaces that are part of solids and Femap will automatically change the original solid into a stitched sheet solid.

- The commands geometry/midsurface single, single in solid, and trim to solid now work with non-manifold bodies.

Nonmanifold add
Performance of the nonmanifold add process has been greatly improved though the use of a single call to the Parasolid® software engine.

Spaceball
Improvements in driver performance now overcome limitations which previously caused Femap to become unresponsive, improving the overall performance of the Spaceball™ 3D mouse with Femap.