

The Siemens logo is displayed in a bold, teal, sans-serif font.

Ingenuity for life

Femap Version 11.3

Benefits

- Easier model viewing and handling
- Faster connection definition and setup
- Faster and easier mesh refinement process
- More accurate meshes with minimal triangle element creation
- Easier analysis setup and control with improved solver integration
- Easier arrow results plot creation

Features

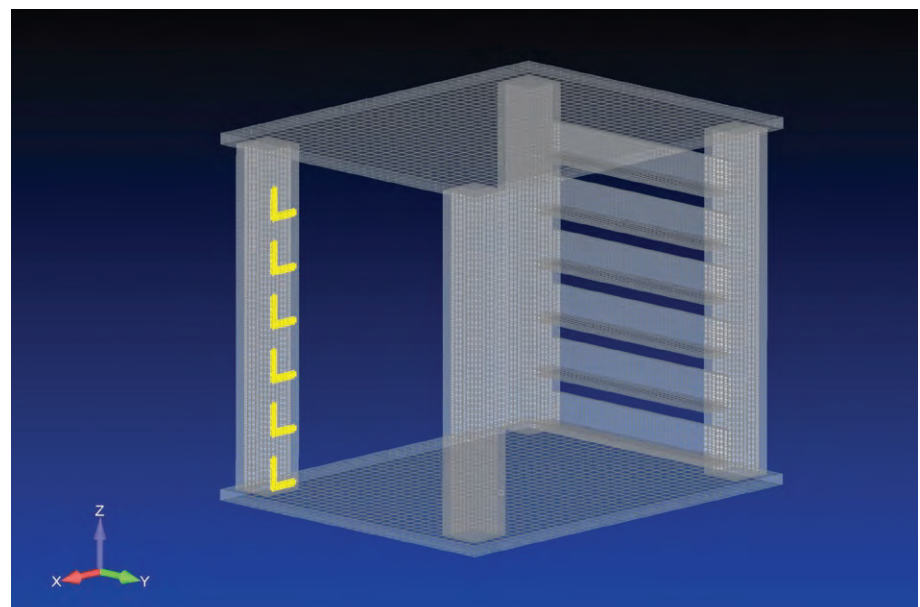
- Enhanced view controls and model visualization
- Direct connection manager
- Interactive mesh refinement
- Max quad meshing option
- Solution launch control
- Automated arrow plot setup

Summary

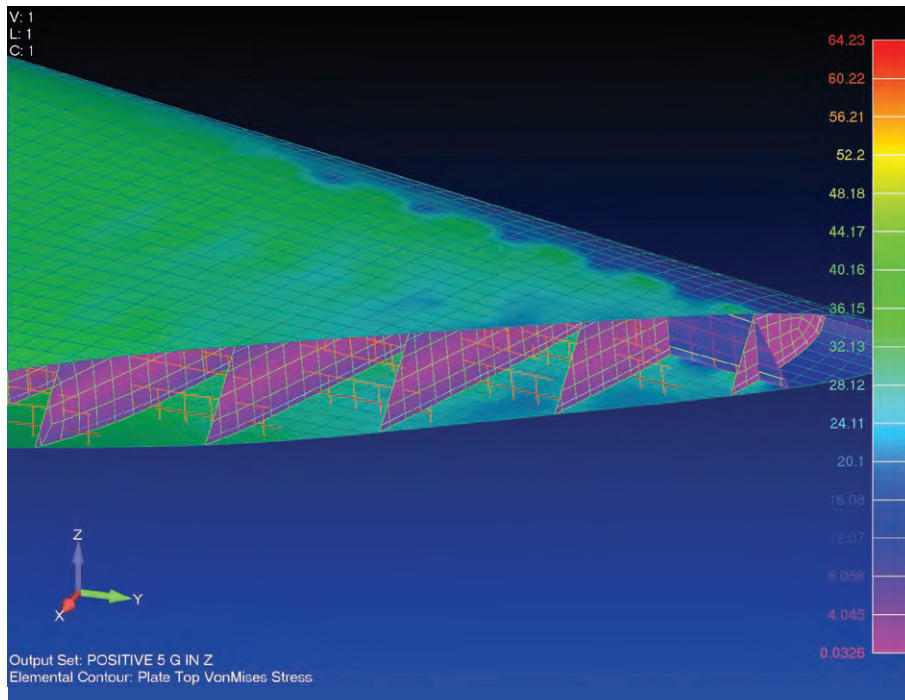
Femap™ software version 11.3 is the latest release of the standalone finite element modeling pre- and postprocessor for engineering simulation and analysis. Femap is CAD-independent and can import geometry from all major CAD platforms and supports most CAD formats. Femap also works in combination with a wide variety of finite element analysis solvers, including the industry-leading NX™ Nastran® software.

Femap 11.3 provides a number of preprocessing enhancements including

greater control of view rotation and zooming, improved beam modeling and model visualization with a new draw/erase capability and a roll-through mode. Connection manager enhancements facilitate a more direct approach to connection definition and setup. Meshing enhancements include a new max quads option minimizing triangle creation, and more streamlined mesh refinement and smoothing processes. You'll see improved solver integration with version 11.3 also, including Abaqus® ODB file attachment, launch control improvements, and increased support for CBUSH element types.



Femap Version 11.3



Preprocessing

Model visualization control

New to version 11.3 is a new draw/erase toolbar that provides an easy way to control entities that are drawn on the screen and those that are erased, allowing more versatile model viewing control. A new roll-through viewing mode allows you to move inside the model and view it from the interior, greatly facilitating model checking and results review.

View controls

New view rotation options allow you to rotate the model about view axes, model axes or a selected a coordinate system. You can also switch preferences to rotate the model about the cursor position, giving you greater control of model rotation and preventing it from swinging out of view. Similarly there is also an option to zoom the model about the cursor position.

Element face picking

Element face picking has been streamlined, enabling you to pick an element face from the elements shown on screen. Femap then collects all of the connected elements based on the default adjacent faces method.

Connection manager

The connection manager has been enhanced to allow a more direct approach to connection definition and setup with direct editing, sorting and filters. When used with element face picking and the draw/erase toolbar, the job of setting up connections on FE entities is much easier and requires significantly fewer mouse clicks.

Beam modeling

Femap version 11.3 also includes improvements to beam modeling with improved cross section documentation and annotation, a basic cross section calculation that is performed automatically, and a full calculation that includes shear center and torsional properties performed at the click of a button. Supported cross sections include the standard Nastran sections as well as arbitrarily shaped sections. For reporting purposes, you can also copy the cross section information to the clipboard or save it to a file.

Pick visible entities

With Femap 11.3 you can now front pick multiple visible entities at a time using box or polygon picking methods, a very fast approach that depends on visible pixels on the screen.

Meshing

Max quads

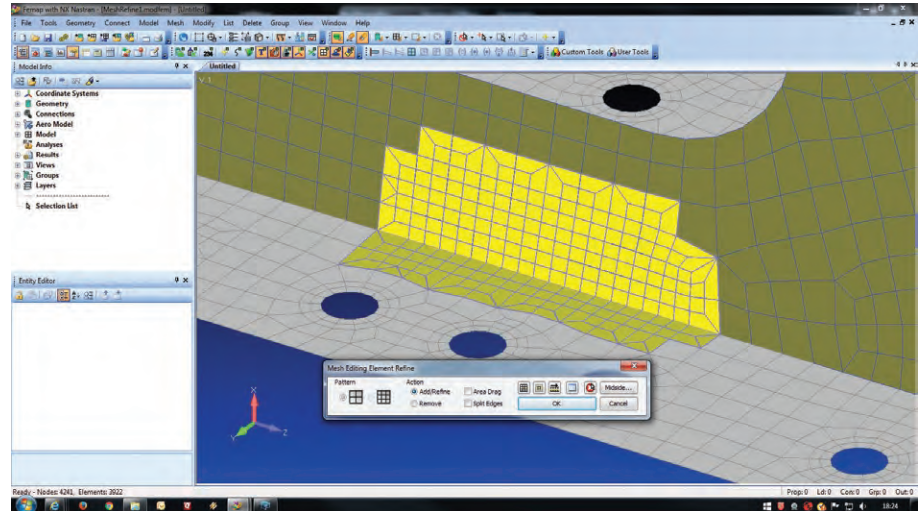
A new max quads option has been added to Femap 11.3 that minimizes the number of triangles produced in a shell mesh within the specified allowable element aspect ratio setting. By modifying the aspect ratio and mesh size, you can obtain all-quad meshes, if the geometry topology allows. Max quads can also be combined with other meshing tools such as quad layers and element growth to produce attractive and accurate quad-dominant meshes.

Mesh smoothing

Mesh smoothing has been improved for surfaces that are highly curved, producing more accurate and better-looking meshes.

Mesh refinement

You can now interactively refine an existing mesh in a specific region to better capture a stress concentration, for example by using the new element refine command. With this streamlined workflow, simply select a few elements and immediately see the mesh refine to four or nine elements per original element. You can also add to or remove from the selection, and employ selection grow and shrink tools. In the process of mesh refinement, Femap will automatically refine any internal beam elements to match the new refined mesh density, propagating any loads, constraints or laminate properties automatically.



Quad mesh growth

Previously, internal mesh growth only applied to triangular element meshes; in version 11.3 the capability has been extended to include quadrilateral element meshes as well. You can elect to have quad elements grow in size in the middle of a mesh away from external boundaries and internal details such as holes. This capability serves to keep the mesh size to a minimum without compromising accuracy.

Multiple element splitting

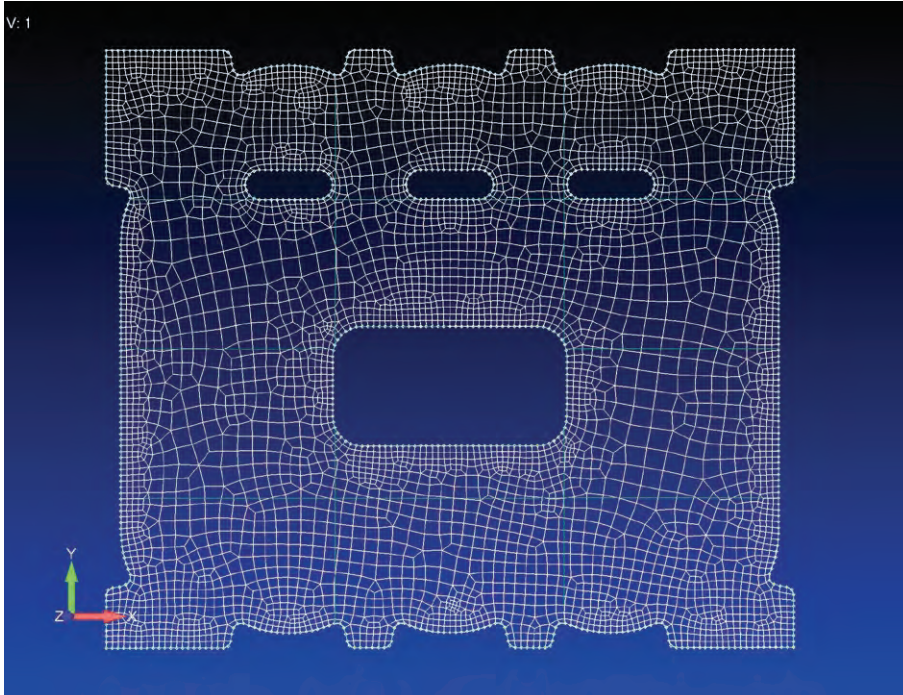
In previous versions of Femap, only a single element width could be selected for the element splitting command. Now in version 11.3, multiple elements can be selected and split in a single command, streamlining the mesh splitting process and allowing meshes to be refined more efficiently.

Meshing hard point management

Handling and management of meshing hard points has been improved in the meshing toolbox with automatic re-association of hard points to surfaces following geometry editing and updates.

Geometry cleanup

Geometry cleanup actions, including removal of small curves and extraneous points, can now be permanently applied at the geometric (Parasolid® software) level, creating new clean geometric entities.



Solver support

Abaqus ODB support

Version 11.3 extends high-speed results attachment to include the Abaqus ODB file. Automatic versioning is supported, and you can also save results data internally to the Femap database if desired.

Nastran analysis study titling

Titling of analysis studies has been enhanced to allow you to optionally add other meaningful attributes such as file name and subcase IDs. Also, if results data sets are read in more than once, Femap will automatically manage the versioning.

Nastran element quality checking

Element quality checking has been extended to include all of the Nastran element quality checking methods.

Nastran frequency input entities

All of the frequency input (FREQ, and FREQ1 - 5) entities are now supported in Femap 11.3. You can apply the frequency input across the model or across specified analysis sets.

Nastran nonzero constraint definition

Femap supports the direct method of defining nonzero constraints in Nastran, so that you can now define constraints directly on the single point constraint (SPC) entity without requiring a load definition.

Nastran grounded CBUSH and CELAS2 definition

The concise method of defining a grounded CBUSH or CELAS2 element when the second node definition is left blank is now supported. With this capability you can use a shorthand method to easily set up spring-to-ground spring elements in your model.

NX Nastran solution control

A new command line access allows you to specify parameters for each job that you run individually at the time of running the job.

Also, use of graphics processing unit (GPU) computing can significantly accelerate certain parts of solution, and new controls are available that enable you to apply GPU computing to specific parts of the solution.

Solver launch control

The Femap user interface now directly supports the environment variable settings that are used to control solution parameters, making it much easier to control the analysis solution.

LS-DYNA enhanced support

The launch control and job setup process for LS-DYNA® has been enhanced in version 11.3. Also, a user-defined material has been added, enabling you to manage and set up any LS-DYNA material.

ANSYS command input

The ANSYS® interface in Femap and the job control methods have been updated and modernized, and a new ANSYS analysis monitor has been added that allows you to monitor any locally running jobs.

Abaqus and ANSYS CBUSH equivalent element support

Support for CBUSH type elements has been added into the Abaqus and ANSYS solver interfaces for fastener modeling. For ANSYS the CBUSH formulation is written as MATRIX27 input representing an equivalent element. For Abaqus, an equivalent MATRIX INPUT definition is created.

Postprocessing

Contour arrow plots

Generation of contour arrow plots has been overhauled for version 11.3 and includes automatic vector selection, so that when you select a particular force output vector, the corresponding components for the arrow plot are automatically retrieved. Also, you can now transform the output data to another coordinate system directly with minimal setup.

Femap automatically determines the arrow styling according to the type of vector displayed, and contoured arrow results can be listed in the data table, where you can sort them and highlight specific entities.

For component display, with Femap version 11.3 you can elect to show output quantities such as displacements or forces as individual components or resultants. For example, fastener shear forces can be displayed as separate y and z components or as a single resultant with the associated x axial force.

Free bodies

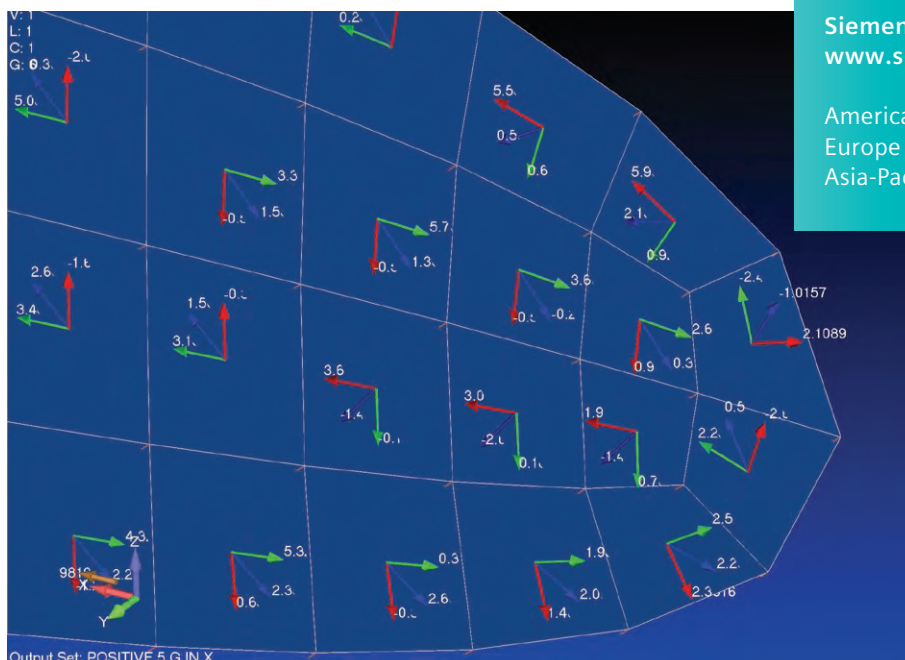
Glue and contact support has been added and glue and contact results are now treated as discrete free body contributions. The free body listing options have also been consolidated, and you can now control these options from a new list force balance dialog box. Also, you can direct output to the message window, data table or clipboard to facilitate pasting into other

applications such as Excel. When output is directed to the data table, the format mimics that of the Nastran f06 output file.

If you have a setup preference for free body diagram creation, you can now control and set these options in the file preferences results tab.

Chart creation

You can now create new data series directly from the model information tree area to streamline the process of creating charts for translations, accelerations and temperature output quantities.



Siemens PLM Software
www.siemens.com/plm

Americas +1 314 264 8499
 Europe +44 (0) 1276 413200
 Asia-Pacific +852 2230 3308

© 2016 Siemens Product Lifecycle Management Software Inc. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Fibersim, Geolus, GO PLM, I-deas, JT, NX, Parasolid, Polarion ALM, Solid Edge, Syncrofit, Teamcenter and Tecnomatix are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other logos, trademarks, registered trademarks or service marks belong to their respective holders.
 55470-A3 4/16 C