

Femap Version 10.2

Benefits

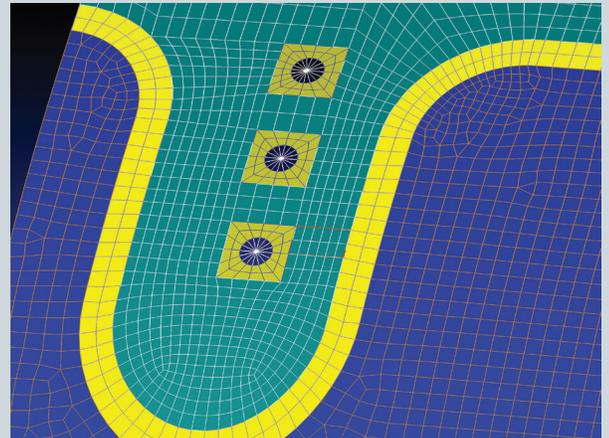
- Improves productivity through process improvement with interactive streamlined surface meshing, geometry preparation and feature editing tools
- Faster output processing response times and more versatile results data handling
- New conceptual designing for minimum weight starting from simple design area models
- Easier visualization and better comprehension of beam element results

Summary

Femap® version 10.2 software is the latest release of the robust finite element modeling preprocessor 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 v10.2 extends a more than 20-year history of productivity and functionality improvements in FEA modeling and postprocessing for engineers. The v10.2 release focuses on geometry preparation and meshing extensions, improved postprocessing capabilities and performance, further NX Nastran solver integration, a new topology optimization capability, as well as numerous other customer-driven updates.

Meshing Toolbox

The Meshing Toolbox has been extended in Femap 10.2 to include two new tools – Mesh Surface and Geometry Editing. There are also additions to the Mesh Sizing tool and a new Feature Editing capability.



Mesh Surface tool

With the new Mesh Surface tool, you can apply surface mesh sizing and mesh attributes interactively, and with automatic re-meshing set, see the effects of the changes immediately. There are also methods that automatically apply three or four corner meshing mapped meshing attributes on suitable geometry. The Mesh Surface tool also provides other surface meshing options including quad layers around a boundary, which can now be applied interactively with mesh updating.

Meshing Toolbox.

VELOCITY SERIES

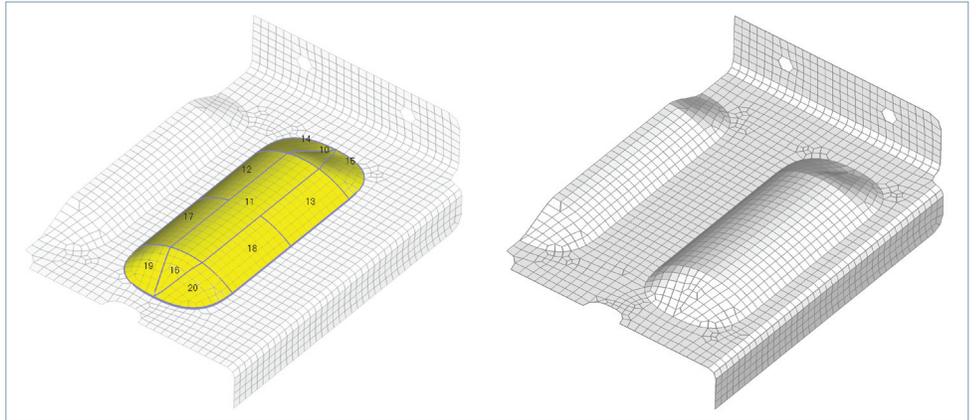
www.siemens.com/velocity

SIEMENS

Femap Version 10.2

Features

- New Meshing Toolbox tools and capabilities including Mesh Surface, Geometry Editing and Feature Editing
- Postprocessing enhancements including performance improvements, beam post-processing and UI dialog consolidation
- New Topology Optimization capability
- NX Nastran 7.1 support including edge-to-face glued connection, Nastran model checking support, improved frequency response support and rigid element extensions
- Customer-driven updates including improved graphical output, a new frequency response analysis capability, update node merge improvements, dynamic model clipping plane and node distance measuring tool



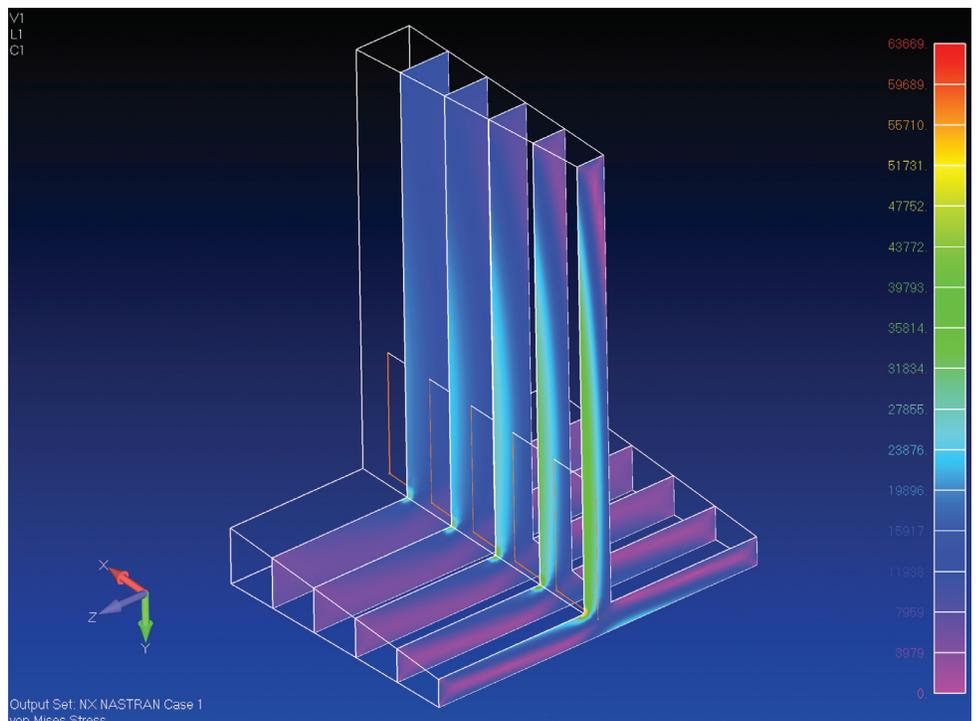
Feature Editing.

Geometry Editing tool The new Geometry Editing tool allows you to prepare the geometry for meshing by interactively splitting surfaces, applying pads, washers and extending surfaces. Curves can also be updated interactively by breaking extending or trimming.

Mesh Sizing tool additions More features have been added to the Mesh Sizing tool to match nodes and multiple curves between adjacent geometry entities. You can also increase or decrease the node count by a factor as well as by a number.

Feature Editing The new Feature Editing capability allows you to move and rotate surfaces interactively to change geometry. Note that if there is an underlying mesh, that mesh will automatically update as changes are made to the geometry. With Feature Editing, you can make changes to any surface or solid geometry features, such as moving holes or extending surfaces as long as geometric topology is maintained.

The enhanced Meshing Toolbox capabilities improve productivity by



Output Set: NX NASTRAN Case 1
von Mises Stress

Beam postprocessing.

replacing a process that previously was manual and required several steps to one that is now interactive and offers automatic re-meshing. Now, the effects of any changes made to surface mesh parameters or geometry updates can be seen immediately.

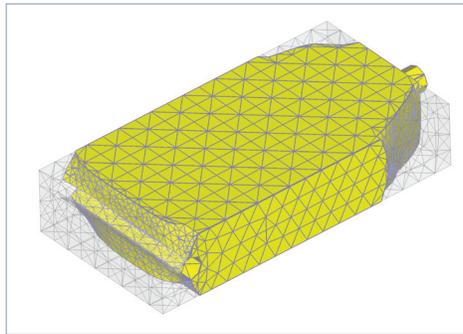
Postprocessing

Femap 10.2 brings several enhancements to the postprocessing functionality, including improvements to the processing of output data, new beam element results viewing capabilities and a consolidated and easier to use postprocessing dialog.

Processing output data Femap 10.2 includes significant performance improvements when processing output data. A speed improvement of 10 to 20 times can be seen with Model Output Process. There is also a new Model Output Delete functionality and an improved undo algorithm that instantaneously removes results data from the Femap database. There is also a new user interface for the Output Processing function with expanded options that allow for more versatile results data handling.

Beam postprocessing With Femap 10.2, you can now visualize stress distributions and shear flows directly on beam cross-sections. Femap offers complete control of the force components that are used to calculate the stresses; additional stress output can be calculated such von Mises.

Postprocessing Toolbox With the consolidation of the postprocessing user interface, Femap 10.2 is easier to use than earlier versions. Access is now possible to all popular postprocessing View options in a new Postprocessing Toolbox. You can now quickly change postprocessing options interactively.



Topology Optimization TorqueLink.

Topology Optimization

Femap 10.2 introduces a new Topology Optimization capability that offers conceptual designing for weight reduction. With Topology Optimization, you can find new designs, starting from simple design area models, and optimize design proposals earlier in the design process. Optimization capabilities include statics and normal modes for static loading and eigenfrequency optimization. The optimization analysis can also be based on manufacturing constraints to ensure that the optimal design proposals are realistic and can be manufactured.

Nastran support

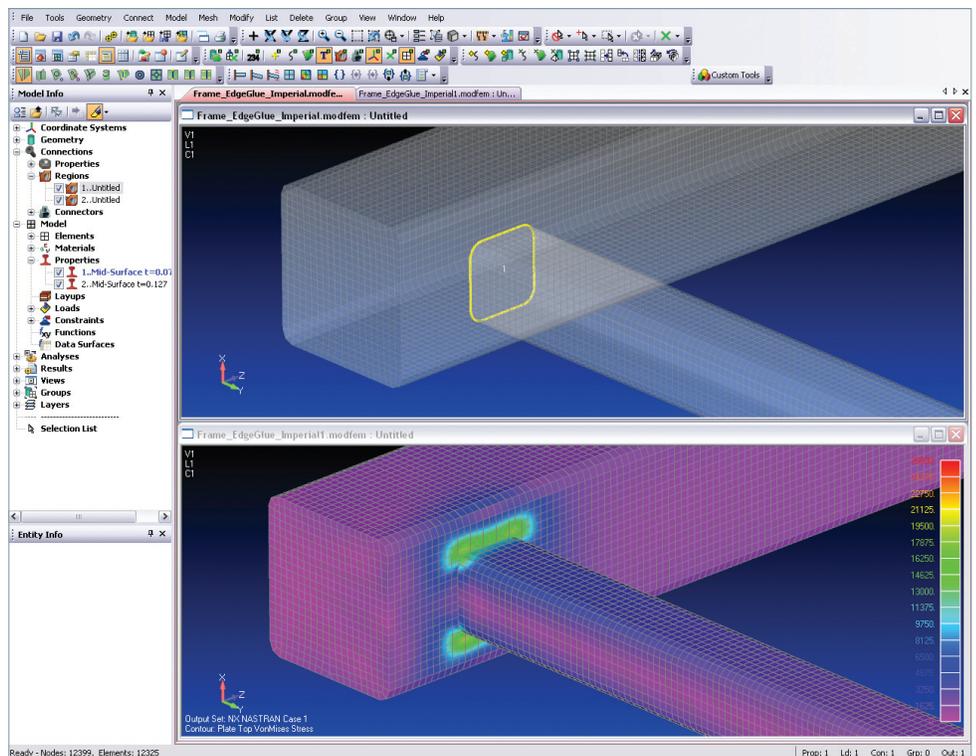
Femap 10.2 includes version 7.1 of the NX Nastran solver and supports several new NX Nastran capabilities including edge-to-face glued connection. Femap also adds support for several other general Nastran capabilities in this release.

NX Nastran 7.1 edge-to-face glued connection

In Femap 10.2, surface modeling and model assembly becomes much easier with support for edge-to-face glued connection in NX Nastran 7.1. To connect surface models together, you no longer need to ensure that a matching mesh exists across adjacent components. In Femap, contact areas and connection set up are automatic; the connection capability offers stiffness tuning for dynamic model correlation.

General Nastran capability support

Every Femap release sees further support for Nastran functionality in general and greater Nastran integration. With this release, Femap adds support for the Nastran rigid body motion grounding and the GROUNDCHECK and WEIGHTCHECK mass reduction model checking methods.



Edge glue frame.

Also, to facilitate set up of frequency response analyses, support for the Nastran FREQ input entities has also been added. For easier setup of rigid elements, Femap 10.2 introduces extended read/write support for the RBE1 type rigid body element definition and the RBE3 UM degree of freedom option.

Customer-driven updates

In addition to the main enhancements mentioned above, there are also numerous customer-driven enhancements that have been included in the Femap 10.2 release.

Update Node Merge Node merging has been simplified and extended in Femap 10.2; in addition, it is now easier to verify the desired action with a new interactive preview option. Update Node Merge now has a simpler user interface and listing; a number of merging options also have been extended. Femap 10.2 prevents over-merging and element corruption when aggressive merging settings with large tolerances are applied.

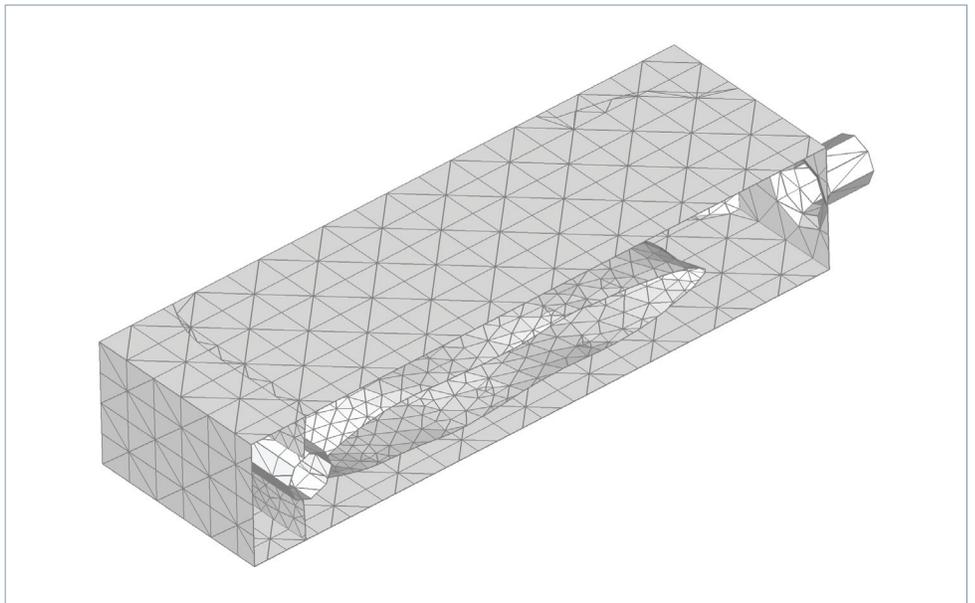
Graphical output Copy, Save and Print actions are now able to query your printer and send a full resolution image, allowing much higher quality images to be produced.

Frequency response analysis You can now run quick frequency response analysis right inside Femap based on previously calculated modal data and node or elemental loading.

Node distance measuring tool The distance measuring tool has been enhanced to measure distances between nodes, including the deformed state.

Quick model clipping plane New mouse-driven model clipping capability is available for geometry and FE model visualization.

New contour algorithm A new contour algorithm improves quality of contour plots for quad elements.



Cutting plane.

Contact
Siemens PLM Software
Americas 800 807 2200
Europe 44 (0) 1202 243455
Asia-Pacific 852 2230 3308

www.siemens.com/velocity

© 2010 Siemens Product Lifecycle Management Software Inc. All rights reserved. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, Jack, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series 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 used herein are the property of their respective holders.
X12 22506 9/10 C