Femap version 10.3

Benefits

- Significant time savings with finite element model creation
- FE models from complex 3D geometry are much easier to create and mesh first time
- Improved FE model accuracy with the removal of geometry imperfections that compromise element quality
- Easier, faster and more accurate aerodynamic statics and flutter analysis preprocessing and model creation and setup
- Easier and faster generation and regeneration of free body diagrams
- Greater flexibility and control of free body display

Features

- Automatic solid geometry preparation prior to finite element meshing
- User interface support for NX Nastran aeroelastic statics and flutter analyses
- A new free body tool extension to the postpro-cessing toolbox that allows easy creation of free body diagrams and investigation of interface loads

Summary

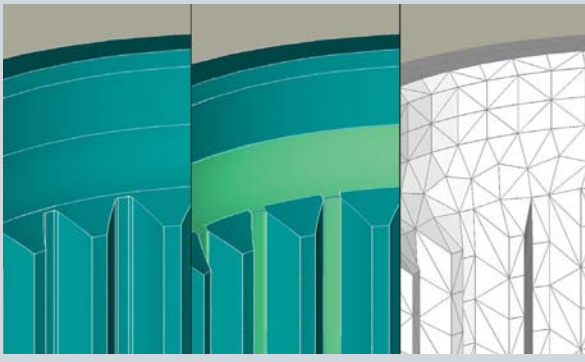
Femap™ version 10.3 software is the latest release of the state-of-the-art finite element modeling pre- and postprocessor for engineering simulation and analysis. Femap works in combination with a wide variety of finite element analysis solvers, including the industry-leading NX™ Nastran® application.

The release of Femap version 10.3 helps improve user productivity through a new automatic 3D geometry preparation capability. The postprocessing toolbox has also been extended to house an overhauled and extended free body tool that provides a detailed results reviewing capability for free body diagrams and interface loads.

Version 10.3 extends the scope of simulation with the introduction of user interface support for NX Nastran aeroelasticity, which allows aeroelastic model preparation for statics and flutter analyses. Femap 10.3 ships with NX Nastran 8.0 and brings numerous enhancements. Performance has also improved for forced dynamic response analyses, and for many preprocessing functions. In addition, many other customer-driven enhancements are included in this latest version of Femap.

Automatic 3D geometry preparation

The Femap meshing process is further enhanced with the new automatic 3D geometry preparation capability. This removes the need for manual model cleanup, which is typically very time consuming, and prepares the geometry so it can be easily meshed to create an accurate and efficient finite element model.



Femap reviews the entire geometry model and identifies potential trouble spots including items such as sliver surfaces, short edges, narrow regions, close points and internal voids. Geometry cleanup actions are carried to combine curves and surfaces where necessary, suppress small

FEMAP

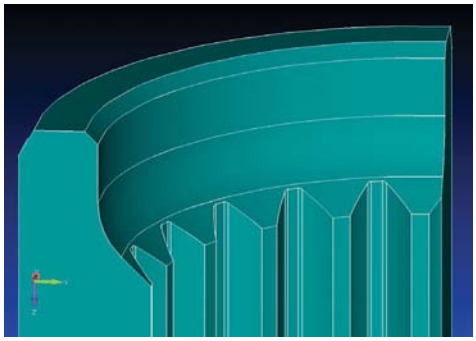
www.siemens.com/plm/femap

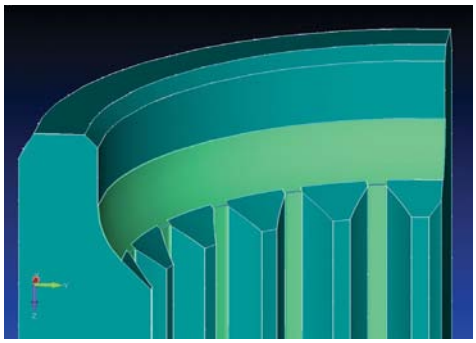
SIEMENS

Femap version 10.3

Features *continued*

- NX Nastran 8 support including new plane stress/strain elements in linear solutions with traction loading
- Customer-driven updates including the ability to embed multiple solids at once and set view alignment by selected surface





features, and split geometry entities at strategic locations to improve the geometry model for meshing – all completely automatically. User-defined tolerances control the extent of geometry changes, and entities that are loaded or constrained are ignored. You can also identify groups of entities to be ignored. This high level of user control avoids unintended loss of important features or load and boundary condition data, and prevents accidental or undisclosed model oversimplification.

Use of the automatic 3D geometry preparation capability allows complex geometry models to be repaired and meshed quickly and efficiently. Once the finite element model has been created, it is possible to continue to enhance and tune it using the Femap meshing toolbox.

Aeroelasticity

Aeroelastic analysis is a capability that enables the simulation of structural models in the presence of an airstream. NX Nastran calculates the loads and damping characteristics from an aerodynamic model and applies them to a structural model. Aeroelasticity is applicable in the design of airplanes, helicopters, missiles, high-speed trains and civil engineering applications including suspensions bridges, tall buildings, chimneys and power lines. Femap version 10.3 extends the scope of simulation with the introduction of user interface support for NX Nastran aeroelasticity, which allows aeroelastic model preparation for statics and flutter analyses.

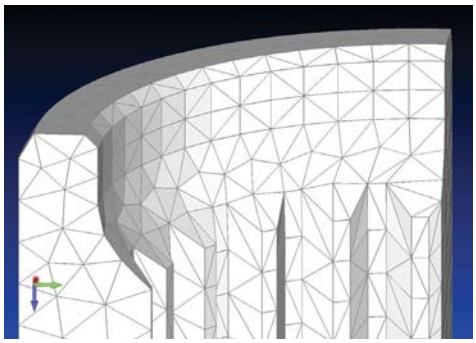
With static aeroelasticity, structural stresses can be obtained due to aerodynamic loading and control systems. With flutter analysis, you can simulate the interaction of aerodynamic forces with natural frequencies that produce potentially destructive rapid period motion. For the aerodynamic model, basic components such as panel and body components are defined, as well as splines that interpolate between aerodynamic and structural grid points. Specify basic flight characteristics within Femap including angle of attack, turn and pitch rates etc., as well as control surfaces and body-to-panel interference factors.

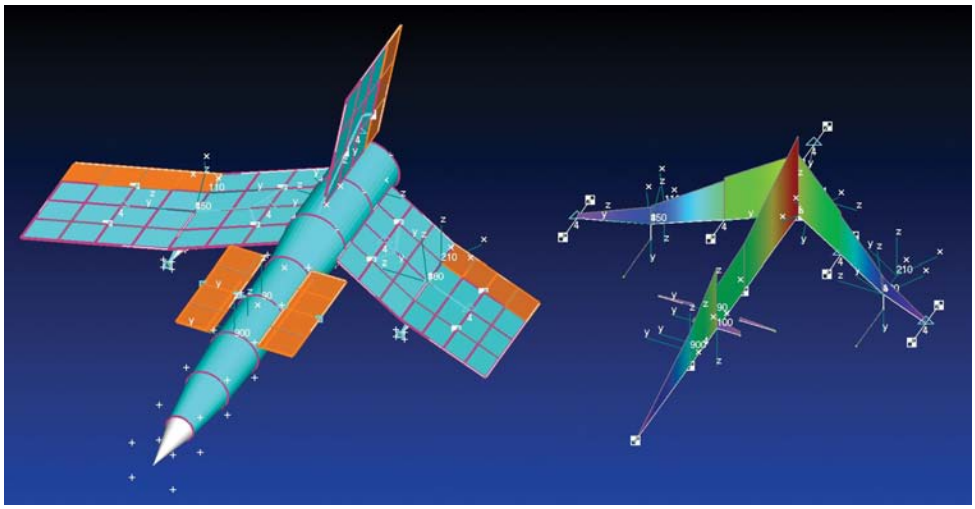
Aeroelasticity is an add-on module within the Femap with NX Nastran product line.

Freebody Tool

Femap version 10.3 extends the postprocessing toolkit and introduces an overhauled free body diagram capability. The Freebody Tool presents a results reviewing capability for free body diagrams and interface loads which provide detailed information about load distribution and transfer in a structure – key information for strength analysis and design.

There are two methods of creating free body diagrams: Freebody Only and Interface Load. The Freebody Only method is the simplest way to display free body diagrams – you select the elements, Femap picks up the appropriate nodes and displays the free body vectors. The Interface Load method provides more





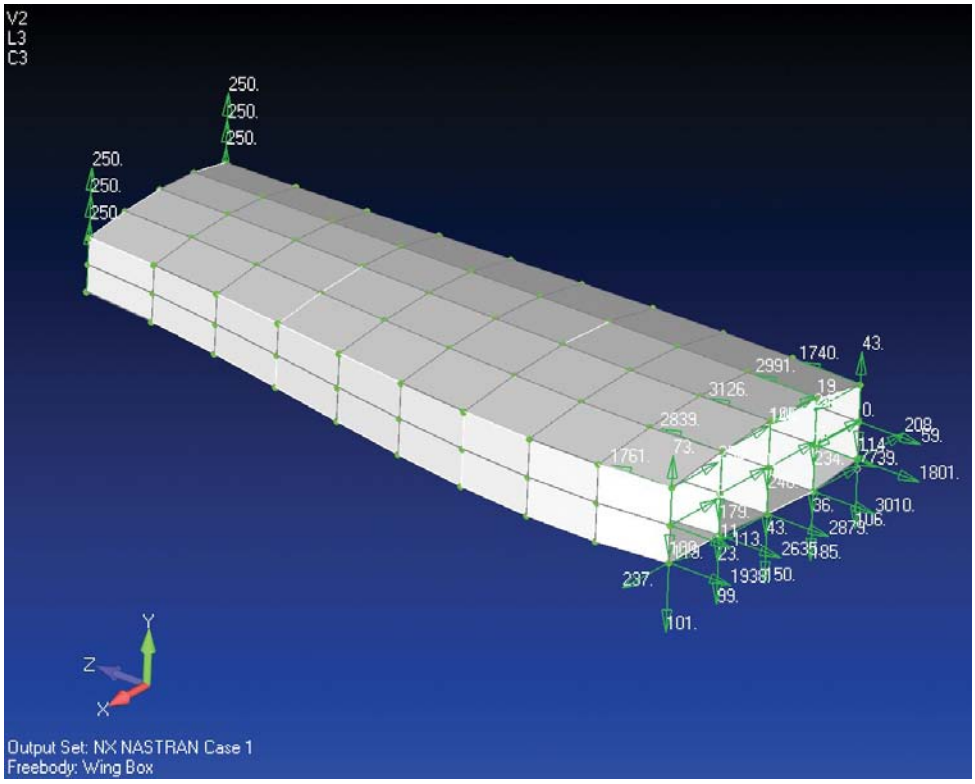
control over the free body display, and you can select both elements and nodes and obtain an "on-the-fly" interface load display. Also, a summation vector is presented, full control of load component contribution is available, and free body data can be listed to the messages pane or the data table and easily transferred to other products such as Microsoft Excel.

The Freebody Tool exists as an object in the Femap database which allows free body calculations to be stored, making free body diagram results easily reproducible. Also, the Freebody Tool introduces greater control of display options, supports multiple free body displays and brings enhanced API capabilities.

Nastran support
Femap 10.3 includes version 8 of the NX Nastran solver, and supports several new NX Nastran capabilities including new plane stress/strain elements in all linear structural solutions and edge traction loading. Femap also adds support for several other general Nastran capabilities in this release.

Performance improvements
Forced response performance has been improved for large models and analyses that require many frequencies

Preprocessing performance has improved for commands that involve the Entity Editor, Meshing Toolbox, and Model Info Tree.



Solid meshing
Solid meshing performance has also improved for the case where the solid already includes some premeshed surfaces.

Customer-driven updates
In addition to the main enhancements mentioned above, numerous customer-driven enhancements have been included in the Femap 10.3 release.

Spring/damper element updates
When spring and damper elements are created the user interface now displays "CBUSH or Other" to facilitate element creation with new symbols applied to distinguish more clearly between element types.

Geometry solid embed
The Geometry Solid Embed command has been modified to allow multiple solids to be embedded into one base solid simultaneously.

View alignment
The View Alignment command has been extended to allow the view to be aligned by a surface, and normal to a plane to facilitate view management.

Contact
Siemens Industry Software
Americas +1 800 807 2200
Europe +44 (0) 1202 243455
Asia-Pacific +852 2230 3308

www.siemens.com/plm/femap