



# Femap with NX Nastran – Getting the most out of your trial

You can re-register as many times as required to download Femap with NX Nastran. The duration of the Femap trial remains at 45 days from your initial registration, and you must use the same email address that was used in the initial registration.

## Getting access to the Femap community site

To help you collaborate with other users and share tips and techniques, you can access the discussion forum on the Femap community site. You can access the Femap community site directly from the help menu in Femap – help/Femap user community, so visit this frequently to make the most out of interacting with other Femap users.

## General trial test plan

After installing the software, you can start to learn more about Femap with NX

Nastran by using the tutorials and user guide available in the online help system. The basic areas in the trial test plan cover the most common simulation tasks and should be used to ensure all areas of the simulation process are evaluated.

## Taking your first steps

**Tutorials:** There are many tutorials in the Femap help/examples menu that cover many aspects of modeling and analysis.

**Documentation:** All Femap and NX Nastran documentation is available through the Femap help/help topics menu should you wish to find out more about any specific command or capability.

Don't forget to make use of the discussion forum on the Femap community site – accessible straight from the Femap help menu.

## Trial test plan

Area	Key points to understand
Simulation process	The parts of the simulation process performed by Femap and analysis by NX Nastran
Interface navigation	Window look and feel, customizable, specialized panes
Model visibility	Control of entity and label visibility, and model highlighter and transparency
Computer-aided design (CAD) import	Wide variety of CAD types and formats supported
Geometry cleanup	The meshing toolbox, entity locator and automatic geometry preparation (version 10.3)
Creation of properties	Material property libraries and element property types
Meshing	Meshing toolbox
Analysis setup	Loads and boundary conditions, types of analysis, output requests
Analysis submission	Variety of solver support
Results viewing	Types of plot and variety of results available, data table

## Contact

Siemens PLM Software

Americas +1 314 264 8499

Europe +44 (0) 1276 413200

Asia-Pacific +852 2230 3308

[www.siemens.com/plm](http://www.siemens.com/plm)

© 2014 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, Quality Planning Environment, 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. Nastran is a registered trademark of the National Aeronautics and Space Administration. All other logos, trademarks, registered trademarks or service marks belong to their respective holders.  
25810-Y9 6/15 B