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.

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

www.siemens.com/plm

44767-Y4 3/15 H