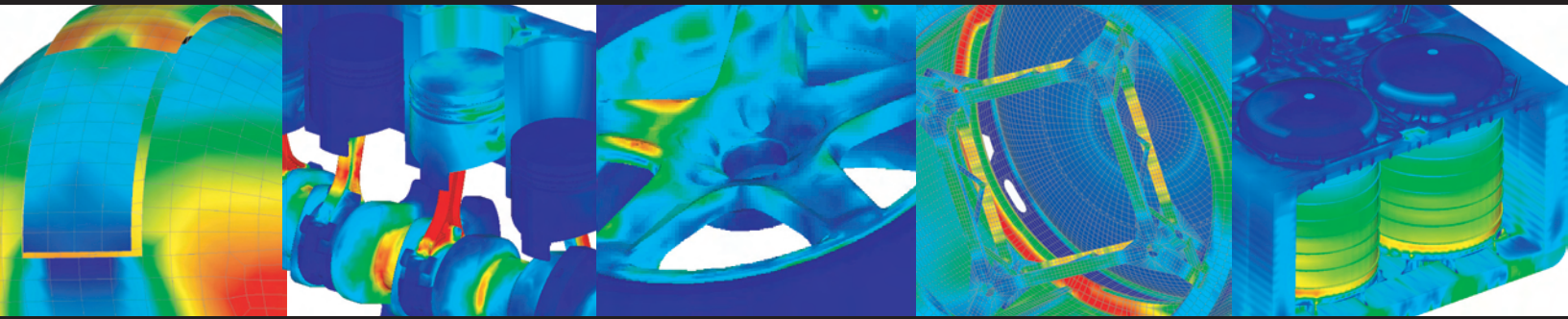
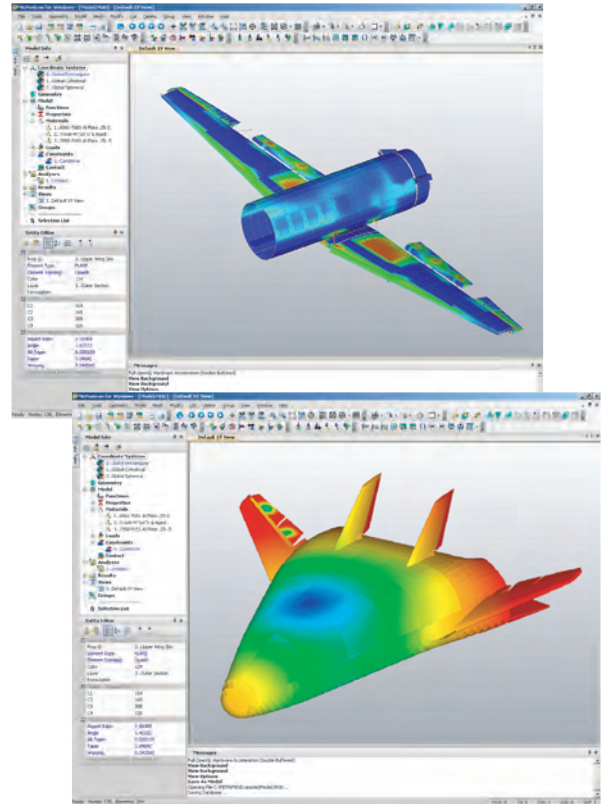


Femap®



Overview

Femap® is a Windows-native pre- and post-processor used by engineering organizations world wide to model various products, processes, and systems. Its graphical user interface provides streamlined, direct access to all Femap® functionality. This version runs on 32-bit and 64-bit operating systems. It also includes significant new functionality like composites layup modeling interface, expanded data surface capability, geometry thicken, custom tools toolbar, group operation Booleans, and post-processing contour display. It also has model info tree enhancements for load and constraint definition, and access to API utilities through Customer Tools toolbar. The pre- and post-processor is tightly integrated with NEI®Nastran and also provides integration to a wide range of industry-standard structural and thermal solvers.



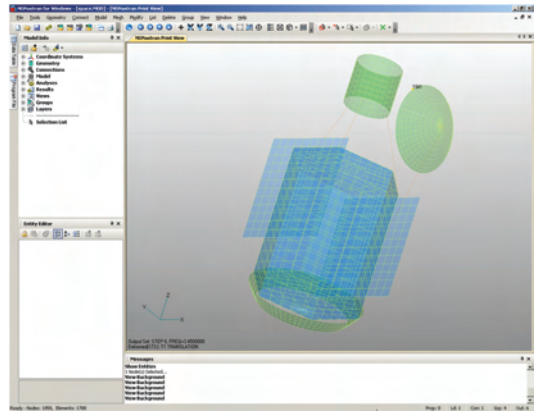
Capabilities

Geometry import/export:

- CATIA import with a new direct CATIA v5 translator (CATIA v4.1.9 to 4.2.4 and v5 r17)
- IGES import (IGES standards 4.0 to 5.3)
- IGES export (Parasolid geometry can be exported to IGES format)
- VDA import (direct access to VDA v2.0)
- I-deas import (access to .idi files generated by I-deas v9m2)
- Pro/Engineer import (Pro/E v16 - Wildfire 3)
- Solid Edge v19 import (access to Parasolid geometry in solid and sheet metal part files including assemblies)
- Unigraphics import (from Unigraphics v11 to v18 CAD models, and direct access to Parasolid geometry exported from all Unigraphics versions)
- ACIS v16 and Parasolid v19 import (each of these interfaces converts imported geometry to the other's solid modeling kernel format)
- STEP AP 203 and STEP AP 214
- Import DXF and IGES points, curves, trimmed surfaces and solids
- Import or export Stereolithography (.stl) data, ACIS (.sat) and Parasolid (.x_t) parts or assemblies

Loads and constraints:

- Geometry or finite elements based
- Nodal forces and moments
- Pressure loads
- Gravity and centrifugal
- Rotational acceleration and velocity
- Single and multipoint constraints
- Symmetric, antisymmetric, axisymmetric boundary conditions
- Multiple loading and boundary condition subcases
- Thermal loading and stress recovery
- Temperature dependent conductivity
- Isotropic and anisotropic thermal conductivity
- Temperature dependant internal heat generation
- Temperature dependent heat transfer coefficient
- Temperature gradient dependent heat transfer coefficient
- Radiation and convection loads
- Surface normal heat flux
- Nonlinear functional forms
- Initial starting temperatures for all transient analyses
- Inertial relief loading
- Transient dynamic, frequency and random vibration



Groups and layers:

- Easily subdivide your model for visualization or post-processing purposes
- Group by:
 - Coordinate Clipping
 - ID, Associativity, Material, Property and Type
- Automatically add new entities to active or user-specified group
- Automatic group creation based on properties, materials, and geometric constraints
- Create layers from groups

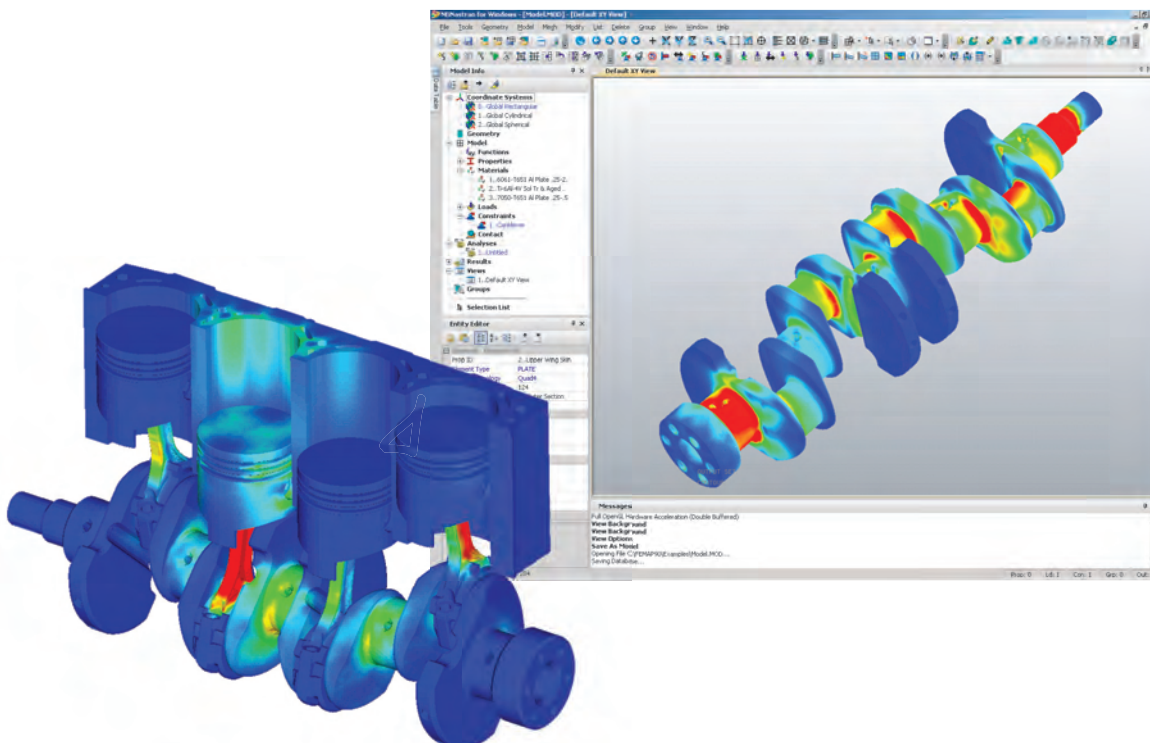
Femap® Capabilities

Geometry creation:

- Create point, line, circle, spline, surface, volume
- Boolean and extrude/revolve solid modeling
- Mid-surface extraction and extended surfaces
- Project curves onto surfaces
- Intersect surfaces to create curves
- Define regions by projecting curves onto solids
- Create curves based on U-V space on surfaces
- Shell solids with constant thickness
- Rule, revolve, extrude and loft surfaces
- Stitch surfaces into solids
- Break, trim, split, extend, join and fillet, offset and copy geometric entities
- Automatically adjust geometry scale factor option

Meshing:

- Subdivision and semi-automatic meshing of solids
- Automatic and mapped meshing (with quads or bricks), including biasing
- Free surface meshing (quads or triangles only)
- Tetrahedral and hexahedral meshing
- Extrude and revolve curves into shell elements and shells into solid elements
- Mesh refinement and smoothing
- Interactive mesh editing
- Full associativity between geometry and mesh
- Glued connection that joins interfacing surfaces of dissimilar meshes
- Auto Boundary Small Surface option – automatic grouping and removal of sliver surfaces
- Alternate method for calculating property values for a beam cross-section



Results processing:

- Deformations, animations, and vector displays
- Single and multi-load set animations
- Filled color contours and criteria displays
- Iso-surface and cutting planes, with dynamic control
- Shear and bending moment diagrams
- Error estimates
- Results across composite laminates
- Results data mapping – transfer displacements or temperatures and map them onto the same model with dissimilar mesh
- Extensive result sorting capabilities
- XY-Plots with multiple curves
- Text reports:
 - Standard
 - User-customized
- Interactive data query with mouse
- Free-body displays, including Grid Point Force Balance support for grouped entities
- Import/export in comma separated tables
- Internet publishing with VRML support
- Save animations with AVI support

Solution capabilities:

- Linear Static Analysis
- Inertial Relief
- Thermal Stress and Deflection Analysis
- Nonlinear Static Analysis:
 - Geometric nonlinear
 - Material nonlinear
 - Tension-only cables and shell elements with multiple loading conditions
 - Gap, slide line, and surface-to-surface contact with friction
- Linear and Nonlinear Buckling Analysis
- Natural Frequencies and Model Shapes
- Linear and Nonlinear Direct Transient Response Analysis with adaptive time stepping
- Direct Frequency Response
- Modal Transient and Frequency Response
- Modal Response and Shock Spectrum generation
- Dynamic Design Analysis Method DDAM
- Random Vibration
- Response Spectra Generation
- Complex Eigenvalue Analysis
- Enforced Motion
- Linear and Nonlinear Prestress
- Linear and Nonlinear Steady State Heat Transfer

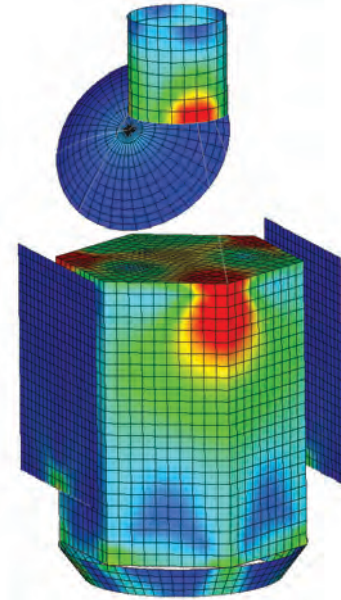
Femap® Capabilities

Element library:

- 1D (rod, tube, bar, tapered beam, pipe)
- 2D (quadrilateral and triangular plates -6DOF/Node, membrane, shells, shear panels, laminates)
- 3D (solids four, five, and six-sided up to 20 nodes)
- New quadrilateral element meshing option that improves mesh quality around critical boundaries and stress raisers
- Plane stress, plane strain
- Spring, mass and damper
- Coupled spring and damper
- Cable element:
 - Initial slack or tension
 - Failure (snap)
- Contact elements:
 - Gap element with friction
 - Slide line element with friction
 - Surface-to-surface contact with friction
- Automatic contact detection and creation (welded or contact)
- Weld option for contact surfaces
- Spot weld (WELD)
- Rigid general form, rigid rod, rigid bar, and rigid plate
- Interpolation
- Conduction
- Capacitance
- Boundary surface

Materials:

- Isotropic
- Orthotropic
- Anisotropic
- Hyperelastic
- Temperature and strain rate dependence
- Stress-dependent
- Creep and composites
- User extensible library with thousands of material properties included
- Nonlinear elastic, bi-linear and plastic



Analysis Set Manager:

- Stores solver setup data with your models
- The command List-Model-Analysis can be written to a file

User interface:

- Customizable, floating dockable toolbars, along with numerous standard toolbars
- Multiple models in the same session
- Multiple views per model
- Cut and paste from one model to another
- Model Information Tree with direct access to higher level entities
- Entity Editor which enables direct review and editing of the model data
- Cut and paste data into other Windows applications with complete retention of the formatting
- Data Table that provides grid-based tabular representation of the model data
- Toolbar Entity Selector that allows entities to be selected from the top level of the user interface
- Full, multi-level undo/redo
- Online Help with hypertext links
- Dynamic highlight during selection operations
- Box, circle, polygon, front, depth and query picking of geometric and FEA entities
- Select entities by associativity
- The Delete confirmation dialog box can be switched off
- Entity Display toolbar for entity display selection
- Updated dialog boxes (File Preferences, Materials, Connection Properties) to use tabbed style

Customization:

- BASIC Scripting Language
- Record, edit, debugging and playback of user-defined macros directly within the Femap[®] interface
- Full-featured, object oriented, OLE/COM-based programming API. API can be directly accessed from a fully integrated, VB-compatible BASIC development environment. Environment includes interactive program editing, stepping and debugging including watch variables. Full context sensitive help linked directly to the API documentation. Dynamic type-library checking and input assistance for completing Femap[®] functions, arguments and constants. The BASIC engine is also fully compliant with programs like Word and Excel. The Data Table can be accessed directly via the Femap[®] API.
- Neutral file: fully documented binary and ASCII file formats

Graphics:

- Dual Windows GDI (Vector-Based) and OpenGL graphics
- 3D dynamic pan, zoom and rotation
- Hidden line and wireframe display
- Free edge and free face display
- Light source shading and transparency
- Transparent view makes the model transparent without changing entity colors
- Complete beam and plate displays, including orientation, axes, offsets, etc.
- Dynamic highlight during selection operations

NEi Software Global Headquarters

5555 Garden Grove Blvd., Ste 300
Westminster, CA 92683-1886, USA
Phone: 1.714.899.1220
Fax: 1.714.899.1369
E-mail: info@neisoftware.com
Website: www.NEiSoftware.com



NEi, NEi Works, NEi Fusion, and the NEi logo are trademarks of NEi Software, Inc. NASTRAN is a registered trademark of NASA. All other trademarks are the property of their respective owners.
Copyright © NEi Software, Inc. 2009. All rights reserved.