

FEMAP®

Femap® Modeler

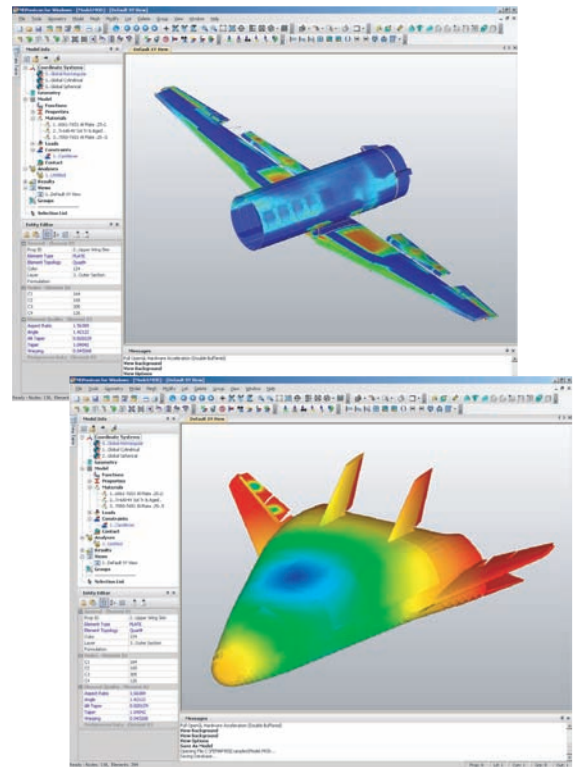
Overview

Femap V10.3 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.

The V10.3 release focuses more on improved user productivity through a new automatic 3D geometry preparation capability.

The post-processing 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.

Femap 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 (v4.1.x, v4.2, or v5r19)
- 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 (v16 – to v20)
- Solid Edge with Synchronous Technology 3 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 v21 and Parasolid v23 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

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

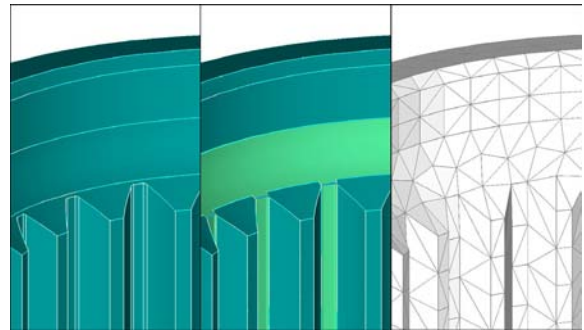
Femap® Modeler

- 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
- Feature and geometry editing tools
- Geometry Solid Embed command allows multiple solids to be embedded into one based solid simultaneously

Meshing:

- New automatic 3D geometry preparation capability
- 3D solid and surface meshers
- 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
- Offset-curves-on-surface feature that guarantees well shaped elements around holes
- Mesh refinement and smoothing
- Interactive mesh generation and verification
- Robust mesh improvement tools
- 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
- Mesh surface tool



Element library:

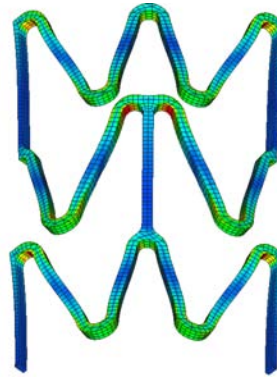
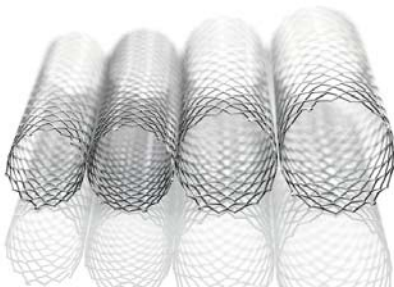
- 1D (rod, tube, bar, tapered beam, pipe)
- 2D (quadrilateral and triangular plates -6DOF/Node, membrane, shells, shear panels, laminates)
- 3D (solids four, five, and six-sided up to 20 nodes)
- New quadrilateral element meshing option that improves mesh quality around critical boundaries and stress 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

Femap® Modeler

- 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
- Rigid element support for thermal analysis in NEi Nastran
- Interpolation
- Conduction
- Capacitance
- Boundary surface
- Global ply support using PCOMPG

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
- Hardening: isotropic or kinematic
- Nitinol and Shape Memory Alloys supported for NEi Nastran



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

Femap® Modeler

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
- Linear and Nonlinear Transient Heat Transfer

Analysis Set Manager:

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

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
- Interactive element and FE model entity display
- Interactive multi-group display with combination groups
- Updated dialog boxes (File Preferences, Materials, Connection Properties) to use tabbed style
- Consolidated visibility options for layer, group, and element visualization

Femap® Modeler

- Recording of tab changes in dialog boxes
- View Alignment command allows the view to be aligned by a surface, and normal to a plane to facilitate view management
- “CBUSH or Other” displayed when spring and damper elements are created to clearly distinguish between element types

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

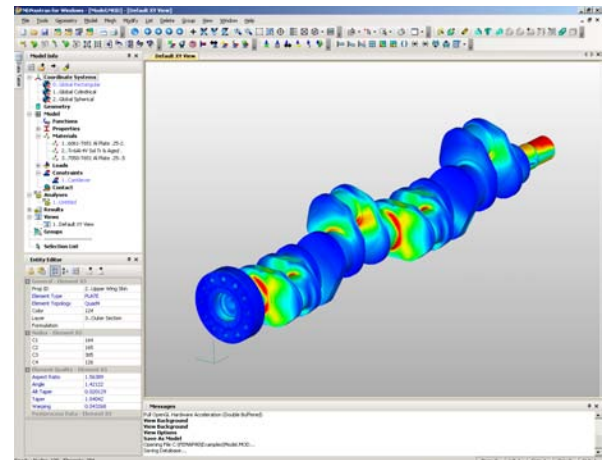
Customization:

- BASIC Scripting Language
- 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
- Record, edit, debugging and playback of user-defined macros directly within the Femap interface

General:

- Changed extension of Femap model files from *.MOD to *.MODFEM. *.MOD file may still be opened.
- New Tools, Measure submenu
- Moved Tools, Distance and Tools, Angle commands under Tools, Measure submenu
- Moved Tools, Mass Properties, Measure Curves and Tools, Mass Properties, Surface Area from Tools, Mass Properties submenu to Tools, Measure submenu
- Added Tools, Measure, Distance Between Nodes and Tools, Measure, Angle Between Nodes commands



About NEi Software

NEi Software is a world leader in Finite Element Analysis (FEA), engineering simulation, and virtual test software. The core product NEi Nastran is a powerful, industry-proven FEA solver that thousands of companies routinely use to perform linear and nonlinear structural stress, dynamics, and heat transfer analysis. In addition, NEi Software's portfolio includes products for impact, kinematics, fatigue, acoustics, optimization, aeroelasticity, and Computational Fluid Dynamics (CFD) with support for a full range of materials from composites to hyperelastic rubber. NEi Software covers the different needs of each stage of the product development process, from designers looking for affordable, easy-to-use, CAD-based simulation for validation and trade-off studies to dedicated FE analysts looking for high accuracy, productivity, and real world fidelity. The website features case studies in aerospace, automotive, maritime, military, civil, petroleum, medical, and consumer products with videos, webinars, tutorials, and options for evaluation.

Contact Information

5555 Garden Grove Blvd. Ste 300
Westminster, CA 92683-1886, USA
E-mail: info@neisoftware.com

Phone: +1 (714) 899-1220
Fax: +1 (714) 899-1369
Website: www.NEiSoftware.com

To see our reseller list, go to:

<http://www.nenastran.com/fea/reseller>

