



Siemens PLM Software

World-class finite element analysis (FEA) solution for the Windows desktop

Benefits

- Significantly speed up the design process by bringing simulation closer to design and reducing time-to-market
- Reduce the need for costly prototypes and testing, saving time and money
- Perform failure analysis that improves product performance and reliability, reducing costly recalls
- Evaluate and optimize designs to minimize material use, investigate use of alternative materials and perform trade-off studies to evaluate differing designs
- Standalone engineering analysis environment that can exchange data with any CAD system and simulate using all major commercial solvers

Summary

Femap™ software is an advanced engineering analysis environment for simulation of complex engineering problems. Using Femap engineers can simulate the performance of their products virtually to determine their performance and behavior, reducing the need for testing and prototypes.

Geometry interfaces

Femap offers latest geometry interfaces access to an extensive range of major CAD systems, including:

- ACIS import (CAD geometry is converted to a Parasolid® software format on import) and export
- Autocad DXF import
- Catia v4 (v4.1.9 through v4.2.4) and v5 (R8 to V5-6R2012 SP3) model file import
- NX™ I-deas™ software import (access to IDI files generated by I-deas 9.2, v10, v11 and beyond supported through I-deas Parasolid export) and universal file export through 9.0
- IGES import and export of Parasolid geometry to the IGES format through v5.3

- NX software import for Unigraphics v11-v18 and Parasolid geometry exported from all NX versions
- NX geometry accessed directly through PLMXML
- Parasolid import and export – Femap is a native Parasolid application, and Parasolid geometry is accessed through the same Parasolid modeling kernel that any Parasolid-based CAD program uses providing full direct access to your geometry
- Pro/Engineer import of model files (v16 to Creo 2)
- Solid Edge® and SolidWorks geometry import – provides direct access to Parasolid geometry for Solid Edge in solid and sheet metal part files including assemblies
- STEP import and export – AP203 and AP214 Class II, III, IV, V and VI entities
- Stereolithography import and export
- Teamcenter® visualization software export – JT™ format including mesh and results data
- VRML export through 2.0/97

www.siemens.com/plm/femap

World-class finite element analysis (FEA) solution for the Windows desktop

Features

- Interactive geometry preparation and meshing tools that facilitate high quality finite element model creation
- CAD-independent and can import CAD data from just about any source
- Solver-independent, supporting all of the major commercial solvers
- Scalable solution offering a range of capabilities up to specialized high-end analysis functionality
- Windows-based intuitive Windows interface that is easy to use and quick to master
- Cost effective solution offering the best functionality-to-cost ratio in the industry
- Postprocessing versatility allowing quick and efficient understanding of the behavior of the system being analyzed
- Versatile modeling and visualization tools to aid FE model creation and verification prior to analysis
- Scalable solutions that can be used to simulate everything from simple solid components to entire aircraft assemblies
- Powerful customization tools that allow integration with other programs

Geometry modeling and editing

Femap provides extensive geometry creation and editing tools that enable engineers to effectively, efficiently create geometry for analysis, including:

- Points
- Lines – modeled by scaling, editing start/end points, break, trim
- Arcs and circles – create by center and radius, points on arc or circle, start/end points. Fillets can be created with the modify tool
- Modify curves – modify by scaling, editing, start/end points, break, trim, extend
- Fillets and chamfers
- Splines – created by Control Point or Point on Spline, Parabola, Ellipse and Hyperbola. Can be defined by equations, midspline and offset. Can be created by Curves on Surface (intersect, project, slice, parametric space curves), and edited by moving control points
- Surfaces – 3 or 4 cornered or edge curves, ruled, bounded plane, extruded, revolved, sweep, aligned, analytic (plane, cylinder and sphere), slice or trim, remove holes
- Midsurface (midplane extraction) – manual and automatic tools, automatic generation and assignment of shell properties to midsurfaces
- Solids – create blocks, cylinders, cones and spheres, extrude and revolve, Boolean operations, explode and stitch, fillet and chamfer, shell thicken and remove face
- Volumes – specify 4, 5, 6 or 8 corners. Select 4, 5, 6 or 8 surfaces. Extrude or revolve, create between two surfaces or by surface and a point. Analytical volumes (cylinder and sphere)
- Geometry splitters – split surfaces between points and edges
- Pad and washer – offset curves around stress raisers to segment geometry for improved mesh creation
- Combine curves – create composite curves to simplify geometry for improved mesh creation

- Combine surfaces – create boundary surfaces to simplify geometry for improved mesh creation
- Surface from mesh – create surfaces from shell mesh

Element library

A complete element library is provided to increase finite element modeling versatility and accuracy, and includes:

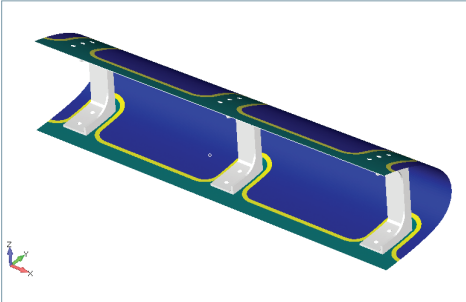
- Line elements (1D) – rod, tube, curved tube, beam, curved beam, bar, link, springs, dampers, d.o.f. spring, gap. Full support of arbitrary and standard cross-sections including all property calculations
- Planar (2D) – plane strain or stress, shear panel, membrane, bending only, plate, laminate, axisymmetric; linear and parabolic quadrilateral and triangular elements
- Solid (3D) – tetrahedral, pentahedral (wedge), hexahedral (brick), pyramid (read and write only) – linear or parabolic
- Miscellaneous – mass, mass matrix, rigid, stiffness matrix, slide line contact, weld and fastener

Materials

Femap provides comprehensive material property definition, including:

- Isotropic – temperature- and frequency-dependent properties, nonlinear and creep models, electrical and optical (for Thermal and Advanced Thermal modules) supported
- 2D orthotropic – temperature- and frequency-dependent properties, nonlinear and creep models, electrical and optical (for Thermal and Advanced Thermal modules) supported
- 3D orthotropic – temperature- and frequency-dependent properties, nonlinear and creep models, electrical and optical (for Thermal and Advanced Thermal modules) supported
- 2D anisotropic – temperature- and frequency-dependent properties, nonlinear and creep models supported
- 3D anisotropic – temperature- and frequency-dependent properties, nonlinear and creep models supported

- Hyperelastic
- Fluids – gas and liquid fluids
- Solver-specific material models for Abaqus, Marc and LS-Dyna
- User extensible library with thousands of material properties included



Boundary conditions – constraints

Boundary condition constraint definitions include:

- Constraint sets – unlimited number of constraint sets per model; you can copy and combine constraint sets
- Nodal constraints – nodal constraints for six d.o.f; you can assign by global or local coordinate systems
- Constraint equations – relate the motion or displacement of two or more nodal degrees of freedom with multi-point constraints
- Geometry constraints – create constraints on geometry including fixed, pinned or zero. You can assign constraints by selected d.o.f and coordinate system. Constraints on arcs including constrain radial growth and constrain radial rotation about center axis

Boundary conditions – loads

Comprehensive loading definition provided by Femap includes:

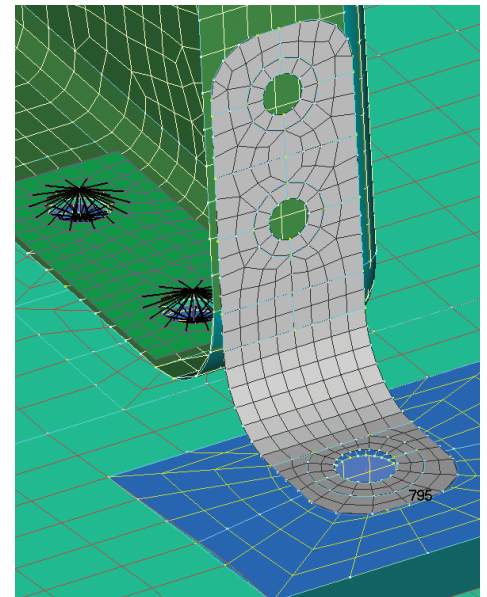
- Load sets – unlimited number of load sets per model; you can copy and combine load sets
- Body loads – unlimited number of load sets per model; you can copy and combine load sets
- Structural loads – force, moment, displacement, velocity, acceleration, distributed loading, bearing load, bolt preload, pressure – geometry- or finite element-based
- Thermal loads – temperature, heat flux, heat generation, convection, radiation – geometry- or finite element-based

- Fluid loads – fluid pressure, scalar, stream quality, relative humidity, fluid height, slip wall, fan curve, periodic – geometry- or finite element-based
- Advanced loads – function-based, variables and equations or data surfaces
- Data surfaces – advanced loading by interpolation between points, parametric evaluation along selected points on surfaces, mapping between models and equations
- Dynamic, heat transfer and nonlinear loads – setup analysis parameters for individual load sets. For dynamics, specify type of dynamic analysis; for heat transfer, specify inputs for free and forced convection and radiation; for nonlinear, specify type of nonlinear analysis
- Nonstructural mass regions

Mesh control

Femap provides an extensive set of meshing control tools and parameters, including:

- Default size
- Mesh sizing on point – sets the mesh size on surfaces attached to a point
- Mesh sizing on curve – length-based sizing, equal length, parameter-based spacing and bias control
- Custom mesh sizing on curve – set the mesh size on curve by projecting existing nodes or mesh sizes on selected nodes onto a curve
- Mesh sizing on surface – includes control of angle tolerance, sizing on small features, growth factor, curvature-based refinement and quad mesh layer control for edges
- Mesh sizing on solids – tetrahedral or hexahedral meshing. Includes control of angle tolerance, sizing on small features, growth factor, curvature-based refinement, quad mesh layer controls and multi-solid sizing for assemblies
- Interactive mesh sizing on geometry with optional auto-remeshing
- Interactive node positioning – move node in free space or slide on surface or curve
- Meshing points on surface – can specify “hard points” on surface
- Approach on surface – specify free, parametric, projection or mapped approach on surfaces

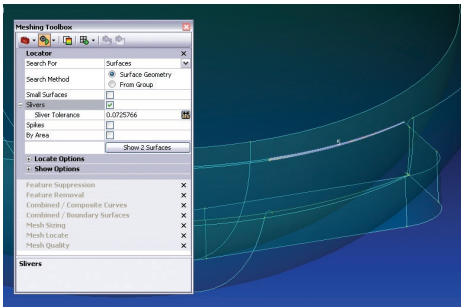


- Attributes on geometry – pre-assign property to selected geometry entities
- Feature suppression and removal – automatic or manual interactive feature suppression
- Geometry and mesh re-association – orphaned geometry and mesh of similar topologies can be re-associated together

Meshing and mesh editing

Benefit from flexible mesh controls and extensive editing capabilities:

- Geometry meshing – points, curve, surface, solids (tetrahedral and hexahedral), volume
- Solid mesh – from closed surfaces, solids from planar elements
- Mesh between – mapped mesh between nodes, lines and surfaces
- Mesh region – ruled mesh between nodes, lines and surfaces
- Mesh connect – disconnect coincident nodes, connect nodes with line, rigid or gap elements
- Mesh transition – between two free edge boundaries
- Mesh edge members – creates line elements on the free edges of planar elements and planar elements on the free faces of solid elements
- Transform meshes – copy by translation, rotation or mirror
- Extrude, revolve, sweep – extrude, revolve or sweep curves, elements or element faces



- Mesh connection regions – automatic or manual creation of contact areas and meshes for face and edge contact modeling. Create and manage preload bolted connections, fluid regions for fluid mass matrix elements and rotor regions for rotor dynamics
- Edit node – project, translate, rotate or align existing nodes; edit node position; and modify permanent constraints
- Edit element – change element connectivity, modify order of element. Scale, translate, rotate or align elements. Modify offsets, orientation and reverse normal of planar elements. Modify element property or type. Modify planar element material angle, plate thickness and shell properties. Adjust beam warping. Add or remove coefficients of thermal expansion for rigid elements
- Planar mesh editing – interactive or automatic splitting of triangular or quadrilateral elements
- Remesh – refine or unrefine an existing mesh, clean up option without resizing boundaries, convert facets to triangular meshes and cleanup slivers
- Mesh splitting – split hexa and penta solids, shell and beam elements
- Merge coincident nodes – preview and merge coincident nodes with ID and grouping control
- Element mesh quality display – aspect ratio, taper, internal angles, warping, Jacobian and combined

Interactive geometry preparation and meshing

The Meshing Toolbox contains several individual tools to help identify and clean up problematic geometry as well as set up the mesh to create a high quality finite element model dynamically. These

interactive tools can modify the underlying geometry, change the mesh parameters automatically updating the resulting mesh, and display mesh quality.

- Feature Suppression – temporarily suppress features such as loops, curves and sliver surfaces; suppressed geometry can be restored
- Feature Removal – permanent removal of geometric entities to simplify geometry
- Feature Editing – move and rotate surfaces to update geometry and mesh
- Geometry Editing – modify (split, extend) underlying geometry and apply pads and washers
- Combined/Composite Curves – combine several smaller curves along the edge of a surface for mesh sizing purposes
- Combined/Boundary Surfaces – combine several surfaces into a boundary surface to improve mesh topology
- Mesh Sizing – set mesh sizing and node spacing on curves
- Mesh Surface – set and modify shell meshing attributes
- Mesh Locate – move and reposition nodes in a mesh
- Mesh Quality – graphically view element quality values similar to a contour/criteria plot

Analysis interfaces and solutions

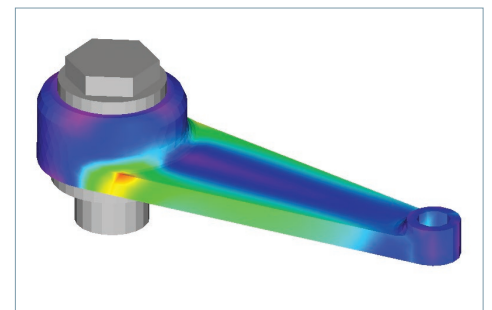
As a solver independent pre- and postprocessor, with Femap you can interface with all major commercial solvers:

- Import and export – solvers supported include Nastran® (NX, MSC/MD, NEi), Abaqus, Ansys, LS-Dyna, Marc, Patran, Sinda/G, I-deas, CAEFEM
- Nastran – capabilities supported include linear statics, normal modes, transient dynamics, frequency response, response spectrum, buckling, design optimization, topology optimization, nonlinear statics, nonlinear transient, nonlinear explicit, aeroelasticity, steady-state heat transfer, transient heat transfer, computation fluid dynamics, superelements, external superelement creation and assembly
- Analysis set manager – create analysis sets that define the analysis parameters, boundary conditions and output

Postprocessing

Post analysis, Femap offers a wealth of postprocessing functionality to let you quickly and easily interpret results data:

- Analysis results – results attached to remain external or imported into Femap database
- Dynamic cross section contours, shear and bending moment diagrams – output for beam and bar elements
- Error estimate output
- Global Plies – results across composite laminates
- XY plotting – Femap functions or results data series including output set results vs. node or element ID, entity output set result vs. selected output sets, entity output set result vs. output set values such as time or frequency, entity output set result vs. x, y or z coordinate
- Deformed shapes – includes control of percent of screen size for maximum value of selected deformation value or scale of deformed vector. Animation with control of number of steps and type. Multi-step animation for transient analysis. Vector plots showing direction and magnitude of selected deformation vector
- Contour plots – filled contour and criteria plots with control of colors and number of levels. Multiple selection cuts supported; and beam diagrams, isosurface and vector plots for displaying magnitude by color and direction of stresses and strains. Control display of average, min or max results on entity



- Freebody diagrams – includes control of resultant vector or component vector display, force and/or moment and freebody on entire model. Freebody forces can be applied as loads into the active load set

- Advanced output control – controls include dynamic animation control (speed, pause etc.), dynamic isosurface and cutting planes for solid contours
- Output capture – bitmaps, jpeg, avi's of animations; JT files can be saved with results contours, internet publishing with VRML support
- Stress and displacement results transformations
- Results reporting – data tables, lists and formatted text reports. API script available to format, annotate and report solid results
- Interactive data query with mouse over
- Import and export of comma-separated tables

Interactive postprocessing

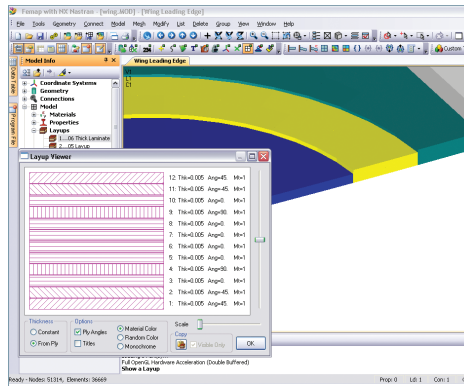
Much of the commonly used post-processing functionality resides in the Postprocessing Toolbox allowing results to be reviewed interactively:

- Deform – controls deform style, options and results
- Contour – controls contour plot style, options and results
- Freebody – controls free body, interface load and view properties

Views, tools and miscellaneous

Part of the power of Femap lies in the attention to detail and the myriad capabilities that collectively make engineering simulation much easier:

- Coordinate systems – includes support for Cartesian, cylindrical and spherical
- Workplanes – create and edit a 2D plane for geometry and FE entity creation and editing
- Views – control view background colors and layers
- Dynamic highlighter
- Light source shading and transparency
- 3D dynamic pan, zoom and rotation
- Hidden line and wireframe display
- Dynamic clipping plane – view the interior of geometry and FE models
- Free edge and free face display
- Element displays – orientation, beam cross section, plate thickness, axes and offsets
- Libraries – support includes saving and recalling of views, materials, properties, layouts, analyses and results output formats



- Functions – define tables of values by XY pairs, linear ramps, equations and periodic functions. Data can be pasted into or from spreadsheets or tab/comma-delimited files. Functions are displayed as XY plots
- Measure tools – measures distance (including nodal deformation), angle, surface area, solid volume. Checks mesh properties and sum of forces
- Check coincidence – check for coincident points, nodes, elements and element normals and distortion
- Layers – full control of active and display-only layers
- Groups – manual or automatic creation of groups of geometry and FE entities, materials and properties. Can be modified with Boolean type operations to create new or combined groups. Active or selected group can be displayed, and results viewed by group
- Model info tree and entity editor window – display model objects including highlighting and editing in entity editor window
- Select toolbar – allows selection of individual or multiple entities
- Toolbars to access frequently used commands to facilitate modeling processes
- Multiple graphics windows – allow access to multiple models simultaneously
- Full multilevel undo and redo
- Online help with hypertext links
- Cut and paste images into Windows applications
- Native Windows look and feel

Customization

Femap offers a powerful customization capability:

- Macros – record, edit, debug and playback user-defined macros directly in the Femap user interface
- API (application programming interface) – full-featured object oriented, OLE/COM-based programming API. API can be directly accessed from a fully integrated Visual Basic 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
- Neutral file – fully documented ASCII file format interface to the Femap database

Contact

Siemens PLM Software

Americas +1 314 264 8287

Europe +44 (0) 1276 413200

Asia-Pacific +852 2230 3308

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, Intosite, 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 used herein are the property of their respective holders.
12288-Y5 7/14 C