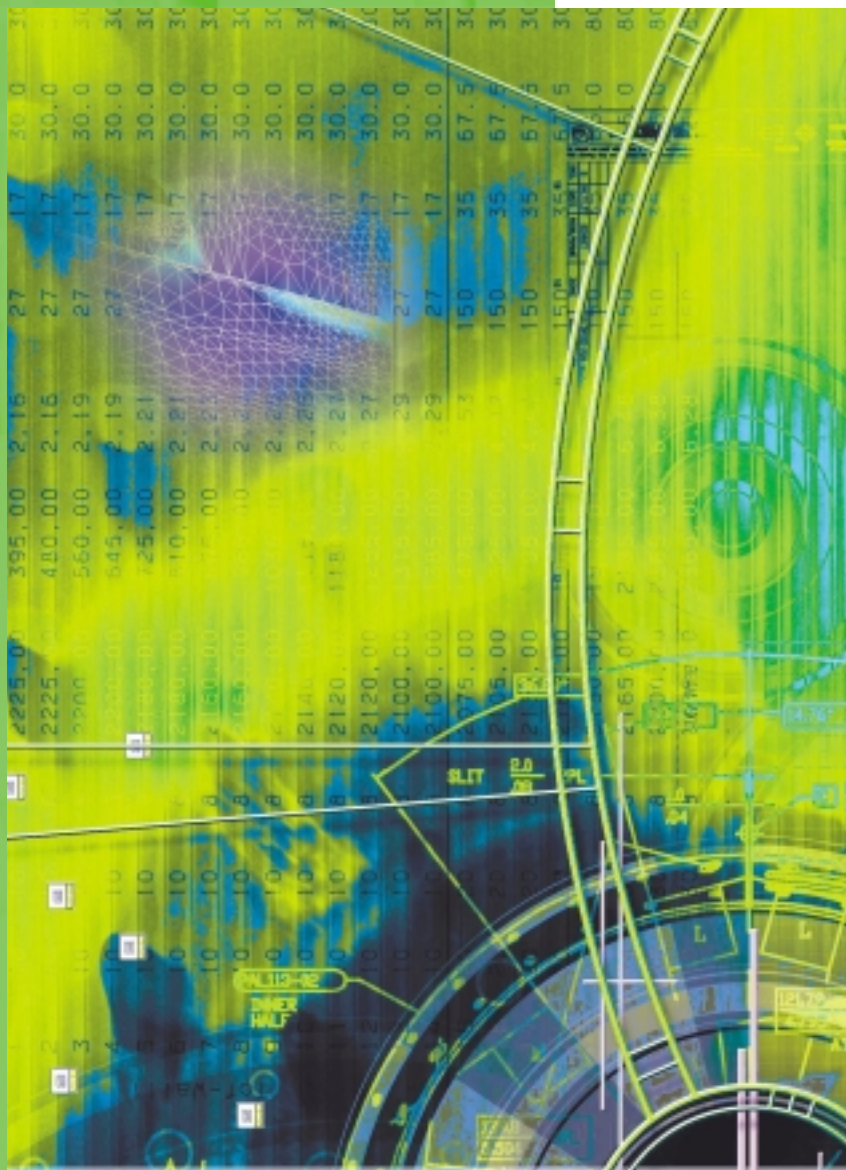
FEMAP®

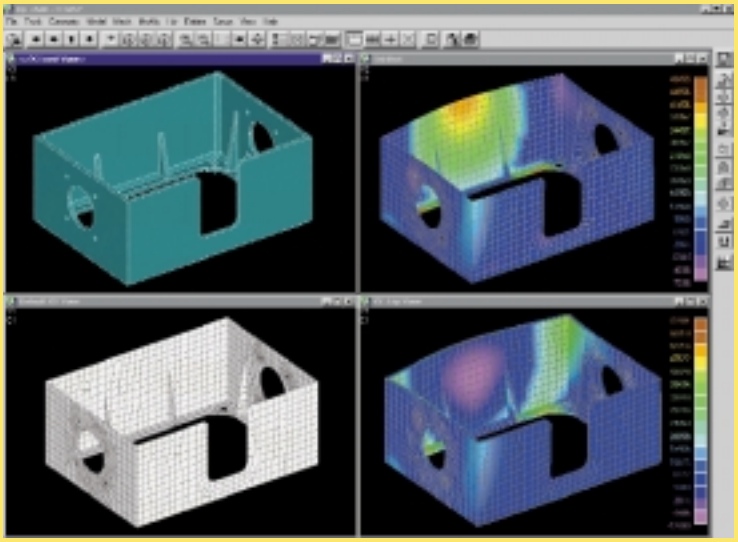
Real FEA Solutions Made Easy



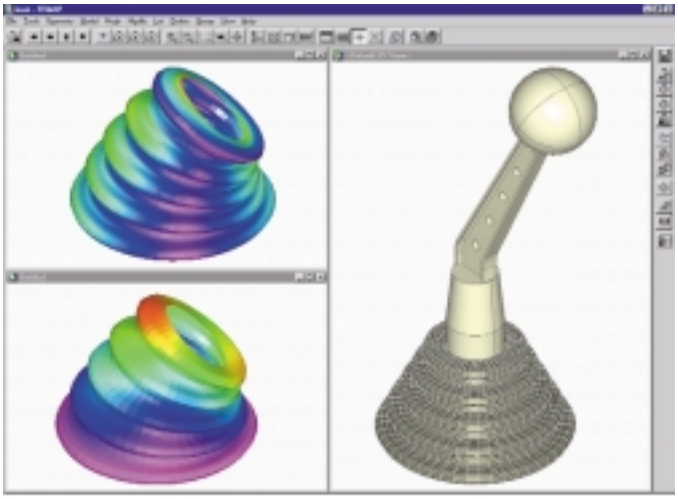
SDRC

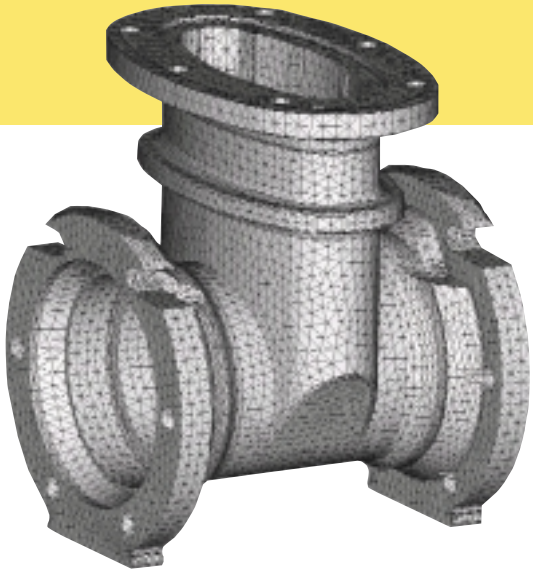
High-performance FEA Modeling with Windows-native Ease-of-Use

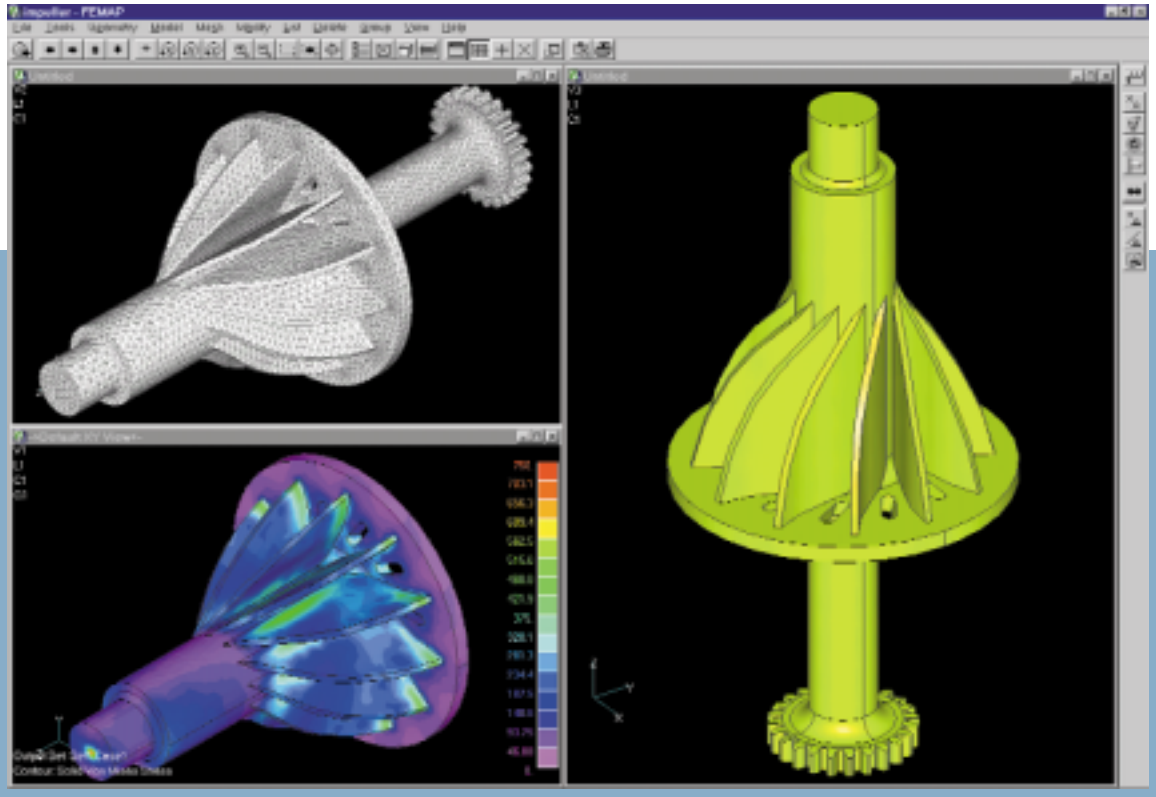
FEMAP® is the world's leading Windows-based pre- and post-processor for engineering finite element analysis (FEA). FEMAP provides engineers and analysts with an FEA modeling solution to handle complex tasks easily, accurately and affordably. FEMAP has focused on power and simplicity in finite element modeling for more than 15 years, and continues that focus today with integrated structural and thermal solvers. As an engineer or manager responsible for finite element analysis, you demand software that is easy to use, but with power to model the toughest analyses. FEMAP delivers just that–high-performance FEA modeling with Windows-native ease-of-use.



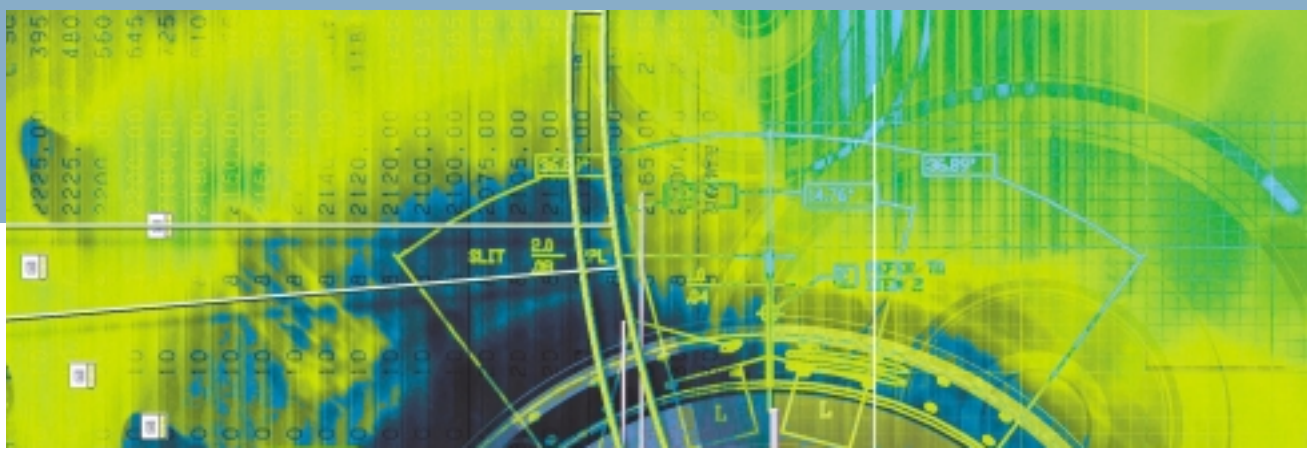
FEMAP products are used by the world's leading engineering organizations to model various products, systems and processes, such as satellites, aircraft, heavy lift cranes and high vacuum seals. From advanced beam modeling, mid-surface extraction and hex meshing, to robust CAD import and idealization, FEMAP gives you unparalleled model control. And with a broad range of loads, materials, analysis types and visualization options, FEMAP is a solid investment for those committed to excellence in finite element analysis.





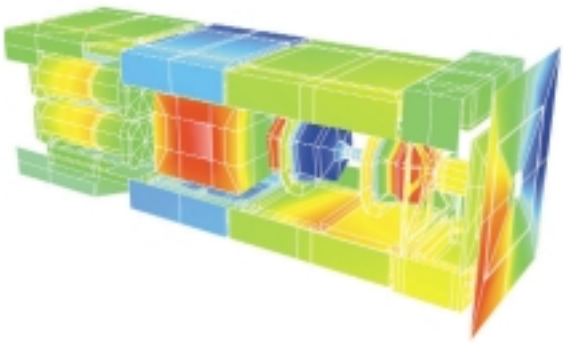


Easier to use than traditional tools and more successful than "push-button" tools, FEMAP provides engineers with a productive analysis solution that is tightly integrated with many CAD systems and FE Solvers, all in a Windows-native environment.



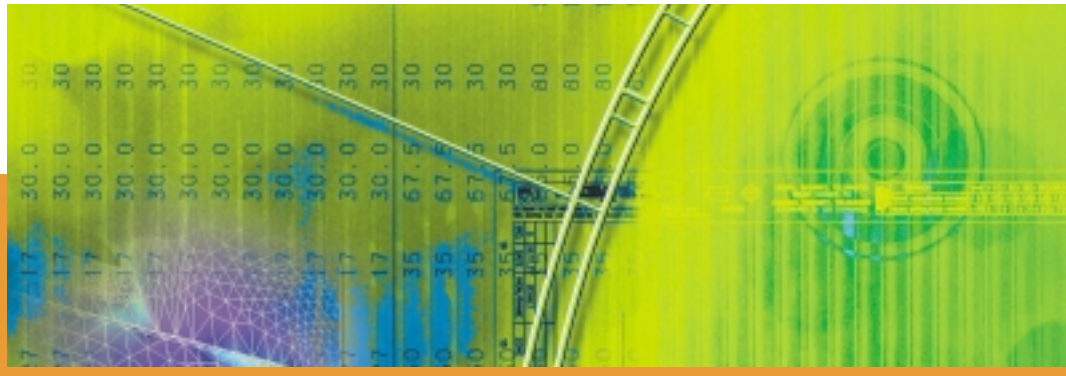
Product Configurations

For those who primarily exchange and modify existing finite element models, **FEMAP Basic** provides exceptional value and performance. FEMAP Basic is a high-performance, general-purpose finite element modeling, and post-processing product with Windows-native, ease-of-use. In addition to the integrated FEMAP Structural and Thermal Solvers, FEMAP supports over 20 industry standard solvers such as MSC.Nastran, ABAQUS, ANSYS and LS-DYNA.



FEMAP Professional includes full functionality and clearly offers the most productivity and value. From transferring CAD surfaces and solids via Parasolid, ACIS, IGES and STEP, to full solid modeling, mid-surfacing and solid subdivision, FEMAP Professional is simply the best way to create FE models from scratch, or from CAD geometry.

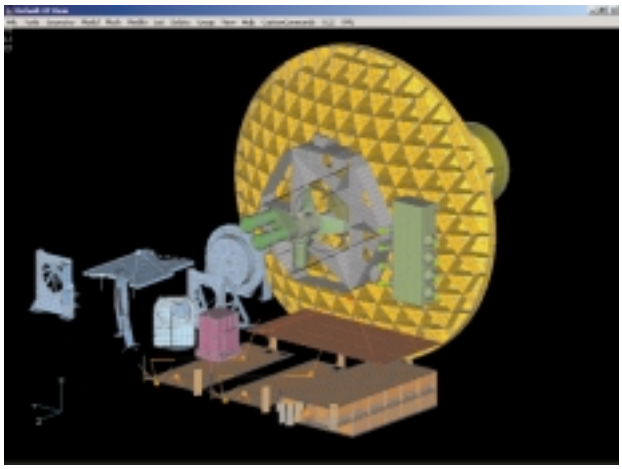
FEMAP Enterprise includes all the functionality of FEMAP Basic and FEMAP Professional, plus direct CAD geometry access to I-DEAS®, AutoCAD (ACIS), Unigraphics, CATIA, Solid Edge and SolidWorks (Parasolid), plus ACIS, IGES, Parasolid, STEP and VDAFS. Pro/Engineer data can be accurately accessed through IGES or STEP with FEMAP Professional or FEMAP Enterprise.

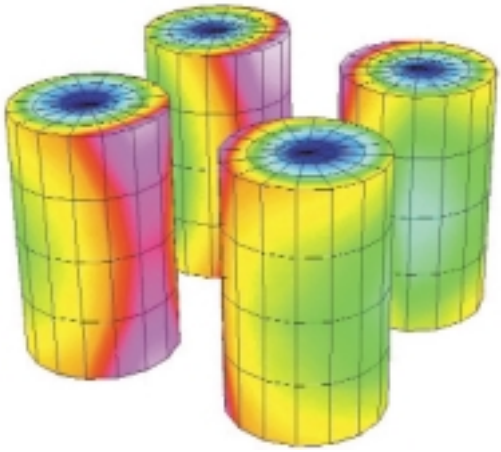


FEMAP Structural combines the world's leading Windows-native pre- and post-processor with time tested and proven solver technology from I-DEAS Model Solution™. FEMAP Structural includes Statics, Normal Mode Dynamics, Buckling and Steady-State Heat Transfer, and includes state-of-the-art sparse matrix and iterative solvers for fast analysis turnaround times.

FEMAP Thermal includes thermal simulation capabilities needed for most engineering applications for transient and steady-state thermal analysis. FEMAP Thermal includes modeling of conduction, convection, radiation and phase change. It provides a range of thermal boundary conditions and solver controls. It also provides a unique and powerful thermal assembly modeling tool called Thermal Couplings which allows you to create paths for heat to flow between parts in large assemblies.

FEMAP Advanced Thermal adds many advanced thermal and fluid flow modeling capabilities to the FEMAP Thermal package for the thermal analyst. In particular it adds fluid duct flow modeling, including coupled convection and fluid flow analysis. FEMAP Advanced Thermal provides an extensive set of tools for advanced radiation and spacecraft modeling, including solar and orbital heating, orbit modeling and display, specular reflections with ray tracing, and articulating structures. It also includes advanced solver features such as custom user subroutines, model simplification, substructuring and interfaces to industry thermal codes.

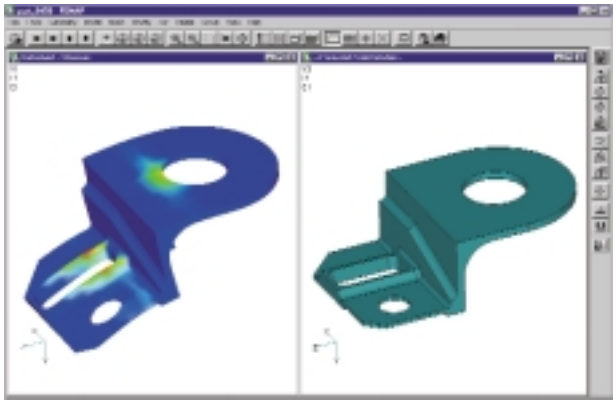




Key Benefits

Serious Engineering on Windows
FEMAP provides depth of functionality in geometry import, geometry creation, meshing, materials and properties, loads and boundary conditions, traditionally found in "high-end" UNIX solutions. FEMAP makes it possible to quickly create models that accurately predict the structural, dynamic and thermal performance of single components or complex systems.

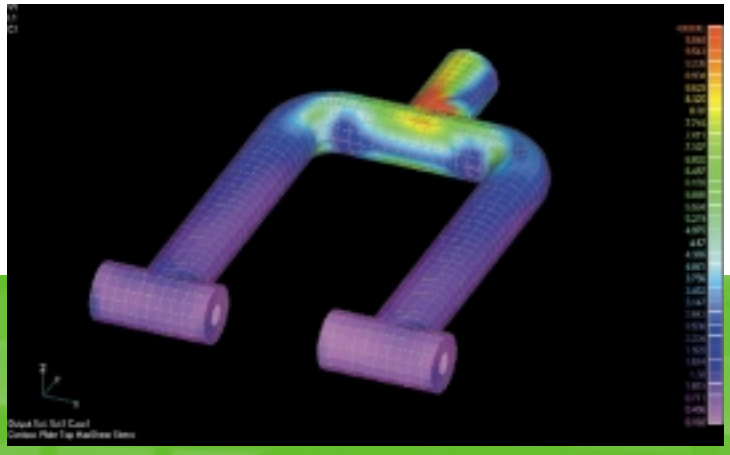
The Right Geometry for Analysis
Engineers are often confronted with geometry that is not the ideal geometry for analysis model definition; FEMAP provides geometry creation and editing tools for curves, surfaces and solids, feature suppression, and mid-surface extraction. Solids can be subdivided and automatically connected to represent dissimilar materials for semi-automatic hexahedral mesh generation. Multiple surfaces can be combined to improve meshing areas for higher quality shell meshes.



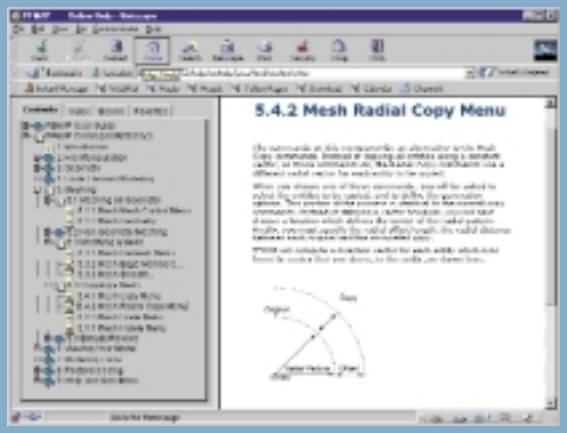
A Better Mesh, Faster Than Ever
FEMAP anticipates your meshing requirements, providing intelligent default mesh sizing. FEMAP also provides flexible mesh controls on points, curves and surfaces, with extensive options for biasing and mesh topology. Plus, fully automatic, high-speed tetrahedral solid meshing and quad-dominant surface meshing.

CAD Independent
FEMAP offers seamless geometry access with every major CAD system. FEMAP also offers extensive geometry creation tools. FEMAP Basic includes standard wireframe lines, arcs, circles and splines, and simple surface modeling. FEMAP Professional and FEMAP Enterprise can create complex solid and surface models using either the ACIS or Parasolid geometry engines. Powerful shelling, blending, boolean operations, surface imprinting and lofting combine to make FEMAP extremely effective at creating geometry for analysis.

Analysis Set Manager
The Analysis Set Manager allows you to store Solver Setup Data with your models, avoiding the need to complete numerous dialog boxes every time you edit your model and create a new analysis input file. The sets can also be saved in a Library for use with other models.



HTML Help System
To make FEMAP even easier to use, its Help system is published in HTML, allowing you to access it using either Netscape Navigator or the Internet Explorer web browsers. You can access the Help system through context sensitive help (the F1 key), the Help menu, or from outside of FEMAP. The Help system includes a collapsible table of contents, full-text search, index and favorites (bookmarks).



Integrating Analysis Technologies
Leading firms recognize it is unlikely that a single analysis technology will meet all their requirements. Moreover, by integrating multiple analysis technologies in a single modeling and visualization environment they can make better designs faster.

Multiple Solver Support
FEMAP provides in-depth and high-quality support for more than twenty solvers, including the popular and proven NASTRAN, ABAQUS, ANSYS and LS-DYNA. FEMAP provides the ability to reuse and integrate existing analysis models from legacy data, and from customers and suppliers. FEMAP has a complete element library with comprehensive support of physical and material definitions to take advantage of the advanced capabilities of these solvers, including dynamic, geometric and material nonlinear applications.

Leading third-party solvers use FEMAP to perform CFD, soil modeling, advanced thermal analysis and electromagnetic simulations.

Analysis Types and Solver Support Results

Analyses	ABAQUS	ANSYS	FEMAP Structual	LS-DYNA	MARC	MSC.Nastran	NE/Nastran	CAEFEM
Static	✘	✘	✘		✘	✘	✘	✘
Modal	✘	✘	✘		✘	✘	✘	✘
Buckling	✘	✘	✘			✘	✘	✘
Heat Transfer	✘	✘	✘			✘	✘	✘
Nonlinear	✘	✘		✘	✘	✘	✘	✘
Transient Response	✘	✘		✘		✘	✘	✘
Frequency Response	✘	✘				✘	✘	✘
Random Response		✘				✘	✘	✘
Explicit Dynamics	✘			✘				
Post Process	✘	✘	✘		✘	✘	✘	✘

FEMAP Basic

FEMAP Basic is the foundation for all FEMAP products. All versions of FEMAP include the following features.

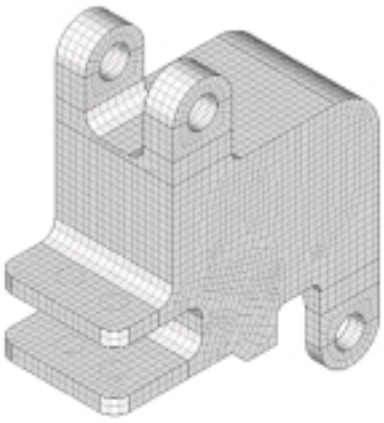
Geometry

- Create points, lines, arcs, circles and splines
- Break, trim, extend, join, fillet, offset and copy
- Import DXF and IGES points and curves
- Import or export stereolithography (SLA) data

Meshing

- Global and local controls with default sizing
- Define element size or spacing with bias
- Free surface meshing, quads or triangles only
- Mapped meshing with quads or bricks
- Direct generation of line, shell and solid elements
- Extrude and Revolve
  - Geometric curves or line elements can be extruded/revolved into shell elements.
  - Shell elements can be extruded or revolved to form solid elements.
- Connected shell elements can be extruded normal to themselves to turn thin-shell models into solid ones.
- Mesh refinement and smoothing

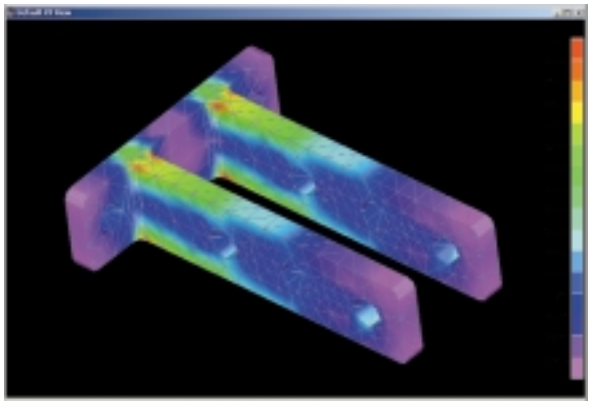
Element Library



- 1D: rod, tube, bar, beam, spring, gap. Full support of arbitrary and standard cross-sections including all property calculations
- 2D planar solid
- 3D surface or solid
- Planar and surface: quads and triangles
- 3D solid: tetra, wedge and brick
- All 2D and 3D elements: linear or parabolic
- 2D planar: plane strain or stress, axisymmetric
- 3D surface: shear panel or membrane
- Mass and general stiffness matrices
- Contact lines and surfaces, and slide lines

Materials

- Isotropic, orthotropic and anisotropic
- Nonlinear elastic, bilinear and plastic
- Hardening: isotropic or kinematic
- Hyperelasticity, creep and composites
- Temperature and strain rate dependence
- User extensible library included



Loads and Constraints

- Geometry or finite element based
- Associativity between geometry and mesh
- Load case definition and management
- Fixed (non-zero) displacements and rotations
- Constraint equations (MFCs)
- Nodal forces and moments
- Distributed loads on line elements
- Constant or variable pressure
- Velocities and accelerations
- Transient dynamic, frequency and random vibration
- Temperatures, heat generation or flux
- Convection and radiation

Customization

- Record, edit and playback macros
- BASIC Scripting Language
- Neutral file: fully documented ASCII file format

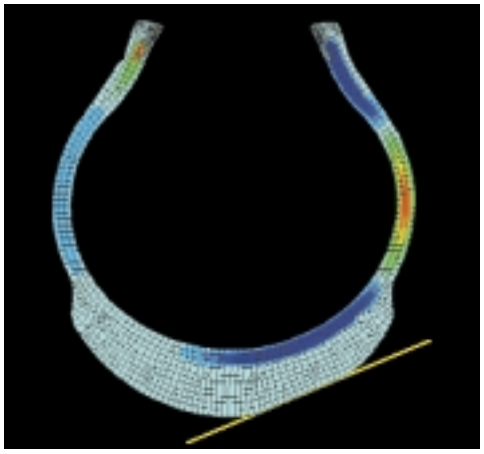
Groups and Layers

With FEMAP you can easily subdivide your model for visualization or post-processing purposes, grouped by –

- Coordinate Clipping
- Automatically adding new entities to active or user-specified group
- ID
- Associativity
- Material
- Property
- Type
- Automatic group creation based on properties, materials, and geometric constraints

Results

- Deformations, animations, and vector displays
- Single- and multi-load set animations
- Filled color contours and criteria displays
- Isosurface and cutting planes, with dynamic control



- Shear and bending moment diagrams
- Error estimates
- Results across composite laminates
- Extensive result sorting capabilities
- X-Y Plots with multiple curves
- Text reports: standard and user-customized
- Interactive data query with mouse
- Freebody 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

User Interface

- Native Windows look and feel
- Multiple graphics windows
- Full, multi-level undo/redo
- Online help with hypertext links

- Toolbars to access frequently used commands
- Cut and paste images into Windows applications
- Dynamic highlight during selection operations
- Box, circle, polygon, front, depth and query picking of geometric and FEA entities
- Select entities by associativity (all elements connected to specified nodes, all elements of a specified property)

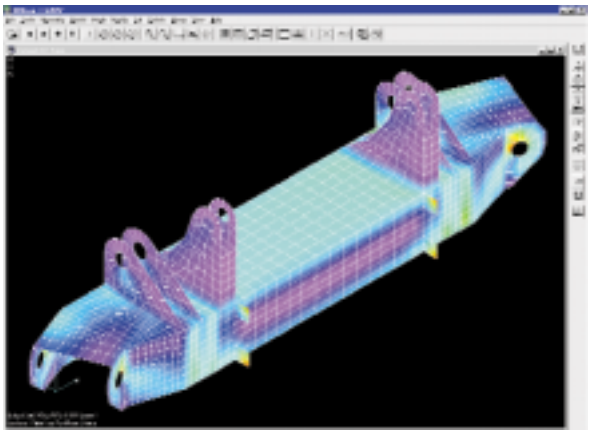
Graphics

- OpenGL support
- 3D dynamic pan, zoom and rotation
- Hidden line and wireframe display
- Free edge and free face display
- Light source shading and transparency
- Element displays: orientation, axes and offsets
- High-quality output–vector-based printing

FEMAP Professional

FEMAP Professional adds the following advanced geometry features to FEMAP Basic.

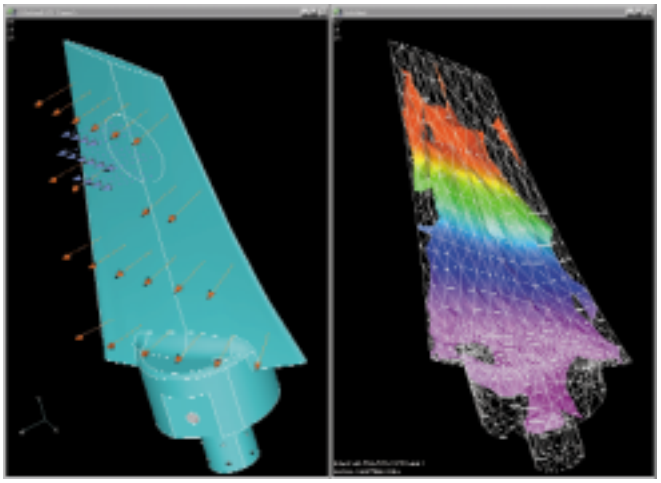
- Direct import and export of ACIS (.sat) Parasolid (.x_t) parts or assemblies
- Boolean and extrude/revolve solid modeling
- Midsurface extraction
- Project curves onto surfaces
- Intersect surfaces to create curves
- Define regions by projecting curves on solid
- Create curves based on U-V space on surfaces
- Shell solids with constant thickness
- Rule, revolve, extrude and loft surfaces
- Stitch surfaces into solids
- Import IGES trimmed surfaces
- Subdivision and semi-automatic meshing of solids
- Automatic solid meshing with tetrahedral elements



FEMAP Enterprise

FEMAP Enterprise adds the following CAD Geometry access capabilities to FEMAP Professional.

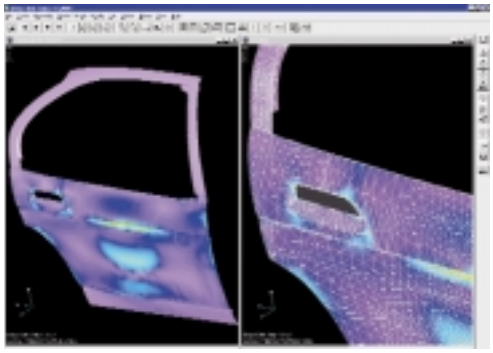
- CATIA Import–reads CATIA model files and Express files from CATEXP. (CATIA v4.1.x or v4.2).
- IGES Import–in addition to the basic IGES interface that is included in FEMAP and FEMAP Professional, an advanced IGES interface that supports many additional entity types is included in FEMAP Enterprise. This supports IGES Standards 4.0 to 5.3.
- VDA Import–this interface provides direct access to VDA files up to v2.0.
- I-DEAS Import–Provides access to IDI files generated by I-DEAS.
- PRO/E Import–Provides direct access to model files from PRO/ENGINEER v16 to v20.
- Solid Edge Import–Direct access to Parasolid geometry in Solid and Sheet Metal part files.
- Unigraphics Import–Direct access to Parasolid geometry from Unigraphics v11 through v15.
- ACIS and Parasolid Import–Both of these interfaces provide the ability to convert the imported geometry to the other solid modeling engine. If you normally work with Parasolid, but you have some ACIS geometry that you need to use, this lets you convert it to Parasolid so that it can fully interact with your other geometry.
- IGES Export–in addition to advanced IGES import, it is now possible to export Parasolid geometry to IGES format.



Solvers

FEMAP Structural

- FEMAP Structural is a general-purpose finite element analysis program.
- It is an integrated part of FEMAP, which means no additional input files need to be created for the analysis. All analysis control parameters are controlled by the Analysis Set Manager.



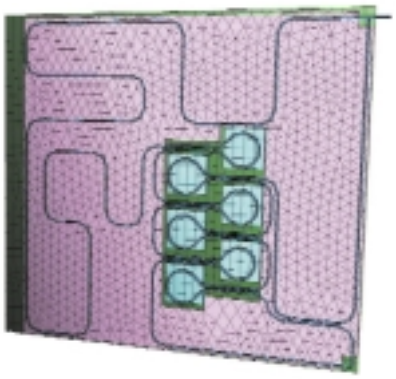
- FEMAP Structural offers a comprehensive set of finite elements to model a wide variety of structures, all with evolving state-of-the-art mathematical formulations to minimize computer time and maximize accuracy.
- FEMAP Structural includes the following solution methods.

- Static Analysis–Stress stiffening effects can be included.
- Normal Mode Dynamic Analysis–Lanczos, Simultaneous Vector Iteration(SVI) and Guyan reduction methods are available. Stress stiffening and Spin softening effects can by included.
- Buckling Analysis–Predicts the loads at which the onset of structural instability will occur.
- Steady-State Heat Transfer Analysis–Solves for the steady-state temperatures due to convection, conduction and heat generation. Material properties can be temperature dependent.

- FEMAP Structural uses a direct matrix solver based on an advanced sparse matrix strategy which minimizes solution time and disk space by taking maximum advantage of matrix sparsity. An alternative is an iterative solver that is especially effective for large models.
- FEMAP Structural is based on I-DEAS Model Solution that is in production use at thousands of sites around the world. SDRC tests FEMAP Structural against internationally recognized standards (e.g., NAFEMS, Soc. Francais de Mecanicians). An extensive suite of test problems, some with closed-form solutions, is used to verify the accuracy of the results.

FEMAP Thermal

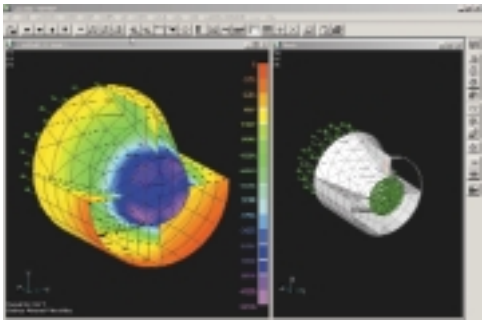
- Transient and steady-state solutions, linear and nonlinear
- Thermal boundary conditions including temperatures, heat loads, fluxes, intitial conditions and thermostats
- Thermal couplings to create thermal assemblies of disconnected FE models including couplings between surfaces, edges and points
- Conduction including isotropic and orthotropic properties, radial heat flow, phase change and time, temperature and direction dependent properties



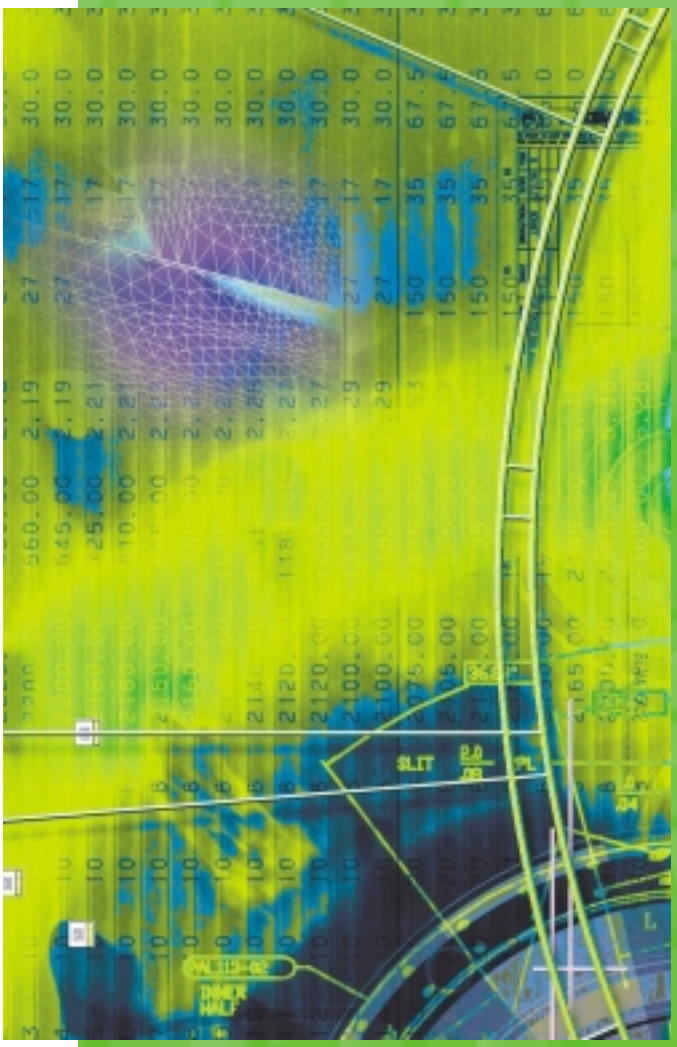
- Radiation including the calculation of view factors with shadowing effects for diffuse surfaces, variable surface properties, axisymmetric radiation modeling, and multiple radiation enclosures
- Convection by specifying boundary conditions using tabular data or formulae
- Additional solver features including axisymmetric modeling, table dependent parameters, non-geometric modeling and solution monitor

FEMAP Advanced Thermal

- Duct Fluid Flow network modeling with coupled forced and free convection simulation for multiple fluids and modeling both incompressible and compressible flows
- Solar and Orbital (spacecraft), diurnal and radiative heating, including orbit modeling and interactive orbit display
- Specular and Hemicube Radiation modeling with ray tracing and modeling of transmissive surfaces



- Articulating Structures for modeling including translating and rotating joints and spinning spacecraft
- Joule Heating simulating electric resistance circuits
- Temperature Mapping (across meshes)
- Interfaces to industry thermal codes such as SINDA, ESATAN, TRASYS and NEVADA
- Advanced Solver features including model simplification, substructuring and user written subroutines, batch solutions and editable input files



SDRC World Headquarters

2000 Eastman Drive
Milford, OH 45150-2740
+1-513-576-2400
+1-513-576-2135 fax
www.SDRC.com

SDRC European Headquarters

Milford House
Priory End
Hitchin SG4 9AL
England
+44-1462-445000
+44-1462-440522 fax

SDRC Asia/Pacific Headquarters

Odakyu Southern Tower 9F
2-2-1 Yoyogi
Shibuya-ku, Tokyo 151-8583
Japan
+81-3-5354-6700
+81-3-5354-6780 fax

Get There Faster®

SDRC

M-1130