The most powerful, Windows-based finite element analysis software is here!

CAEFEM

Finally, the answer to the design engineers looking for a powerful analysis software.

From the massive structures (the largest bronze statue in the USA at the entrance to the MGM Grand Hotel, Las Vegas) to the smallest parts (dental crown model, Cicero Dental Systems, Netherlands), stress analysis using CAEFEM is a snap.

Developed in C/C++, CAEFEM is a true Windows application. Metal plasticity, contact, large deflections, steady state and transient heat transfer, natural frequencies, buckling, dynamic response, frequency response, and response spectrum analyses are some of the features of CAEFEM.

Solution of a Million node problem in Eight minutes!

Imagine! Solution of a hard disk housing modeled with 3,006,864 degrees of freedom in just 8 minutes using Pentium IV, 2.53GHz PC. Solution of a million node eigen value problem in 24 minutes.

Lion Statue at MGM Grand Hotel

Finite Element Model

CAEFEM was used to perform the stress analysis of a 90000 lb bronze lion statue at the entrance to the MGM Grand Hotel in Las Vegas. This massive statue is 70 feet tall including its pedestal. MGM Lion Statue was cast in 1,600 individual half inch thick plates

Analyzed for thermal, wind gravitational, and seismic acceleration loads.

Solution of complex and huge real life problems on an ordinary PC !

The new standard in FEA solutions

Powered with FEMAP

CAEFEM comes powered with FEMAP, the industry's leading pre and post processor. FEMAP, a true Windows application, has interfaces to a number of CAD (AutoCAD, IDEAS, Solid Edge, SolidWorks, Pro/ENGINEER etc.,) and FEA software.

Based on both ACIS and Parasolid engines, FEMAP has bi-directional interfaces with many CAD packages. Its high speed automatic mesher generates about 150,000 elements per minute. Being a true windows application CAEFEM is the only solver seamlessly integrated with FEMAP using Dynamic Data Exchange interface.

Cicero Dental Systems, Hoorn, Netherlands uses CAEFEM to perform the stress analysis of ceramic dental crowns subjected to chewing forces.

Just imagine the combined power of
FEMAP & CAEFEM.

Highly accurate ... and bug free!

CAEFEM is virtually bug-free software that delivers extremely accurate solutions. Check our web site for details on the benchmark problems.

BAE Systems, Chrysler, DuPont, Mitsubishi, NASA, OTIS elevators, Panasonic, Raytheon, SONY, Toshiba, Universal Studios and Xerox are just a short list of CAEFEM users.

The Daytona Prototype built by FABCAR, demonstrated flawless performance during its maiden win in the Nextel Grand Prix of Miami in March 2003. Designed in Soild Edge and analyzed suspension and roll bar using CAEFEM by Fin-el, LLC.

Powerful performance,
Ease of use, Great pricing

The right
Finite Element Analysis
software for you.

Concurrent Analysis Corporation
50 Via Ricardo
Thousand Oaks, CA 91320, USA
Tel : (805) 375 1060
Fax: (805) 375 1061

Web: http://www.caefem.com
Email: info@caefem.com

CAEFEM v9.0 Technical Specifications

Linear Static Analysis	Any combinations of load and constraint cases can be analyzed in a single run.
Natural Frequency Analysis	Inverse, Jacobi, Lanczos and Subspace iteration techniques, Automatic calculation of rigid body modes, Sturm sequence check, User specified frequency value shift, Stress stiffening, Calculation of eigen values within a given range, Lumped and Consistent mass matrices, Mode participation factors. Ability to store only user specified mode shapes in the database to reduce the disk space requirement.
Buckling Analysis	Jacobi, Inverse and Subspace iteration techniques, Automatic calculation of rigid body modes, Sturm sequence check, User specified eigen value shift, Stress stiffening, and Multiple buckling modes.
Steady State Harmonic (Frequency) Response	Calculates steady state (harmonic) response due to external sinusoidal loads. Both magnitude and phase of external loads can be functions of frequency. Based on Mode superposition method. Rayleigh, overall structural and modal dampings. Frequency dependent magnitude, phase, damping factor, structural damping, quality factor, and mode number. Supports all types of loads (including nodal, pressure and line loads on beams). Response at user selectable frequencies. Automatic calculation of normal modes needed for this analysis.
Response Spectrum Analysis	Response Spectrum analysis finds the response of a structure due to external loading such as earthquakes, wind loads etc., Displacement, velocity, acceleration, force, and Rocking spectrums. Base Excitation. Automatic calculation of normal modes needed for this analysis. Rayleigh, Structural and frequency dependent modal damping options. Mode combination methods: CQC, Double Sum, Grouping, SRSS and NRL
Linear Dynamic Analysis	Both Direct time integration and Mode superposition methods. Accepts Rayleigh, Spring/Damper element, Modal (frequency dependent, Critical damping ratio, Quality factor, Structural damping), Overall Structural and Viscous dampings.
Nonlinear Static Analysis	Contact (surface to surface contact), Gap with friction, Large deflections, Large rotations, Large strains, Nonlinear material models (von Mises, Drucker-Prager, Mohr Coulomb, Mooney Rivlin), Isotropic & Kinematic hardenings, Bilinear stress-strain curve, User specified stress-strain curve Regular and Modified Newton Raphson, Updated and Total Lagrangian, Line search option. Automatic time stepping. Deformation dependent pressure loads. Ability to interrupt and change parameters during execution.
Nonlinear Dynamic Analysis	Newmark Beta Method and all of the options mentioned for nonlinear static.
Steady State Heat Transfer Analysis	Linear and nonlinear, Automatic switching to nonlinear analysis depending on the nonlinearities. Time and Temperature dependent properties. Time and temperature dependent nodal temperatures, nodal and element heat generation, nodal and element heat flux, convection and radiation.
Transient Heat Transfer Analysis	Euler Backward implicit scheme and all the options mentioned for Steady state heat transfer analysis. Allows steady state solution to be used as an initial condition. Phase change effects (during melting/freezing) are also supported.

Solution Techniques	Skyline Solver : Direct solver Sparse Matrix Solver: Direct solver which is very efficient for large problems. Iterative Solver : Based on Pre-Conditioned Conjugate Gradient Method
Element Library	Rod, Cable, Bar, Beam (tapered, hinges, unsymmetric, shear deformation, nodal offsets), Tube, Gap with friction, Surface to surface contact, Membrane, Plane Stress, Plane Strain, Axisymmetric, Solid, Tetrahedron, Wedge, Shell (thin and thick), Laminate, Honeycomb, Spring, DOF Spring, Damping, Rigid, Stiffness, Mass Matrix and Nodal Mass. Supports both linear and parabolic elements. Full and Selective Reduced Integration and Incompatible formulation are available for all continuum elements. Nonstructural mass is supported for all appropriate elements. All elements are available for both structural and heat transfer analyses with the following exceptions: Rigid, Gap, Spring, DOF Spring, Damping, Laminate, Mass, Stiffness and mass matrix elements are available only for structural analyses.
Boundary Conditions	Prescribed Nodal Displacements and Single Point Constraints in any user defined coordinate system. Multi Point Constraints (Constraint equations) are also supported.
Loads	All structural loads can be time dependent and can be specified in any user defined coordinate system. Nodal forces, Moments, Velocities, Accelerations, Pressure loads, Distributed line loads, Gravity and Centrifugal loads and Deformation dependent pressure loads are some of the available loads. CAEFEM also supports time and temperature dependent thermal loads like Prescribed temperatures, Nodal and Elemental heat generations, Nodal heat flow, Convection, Radiation and Heat flux from element faces.
Thermal stress calculation	Automatic transfer of temperature results to structural analyses.
Print / Save / Group Options	Groups of nodes/elements/time steps can be created for printing/saving into database of different entities. Stiffness and Mass Matrices (both global and elemental matrices)
Miscellaneous Options	Stress Stiffening Effects for Natural Frequency and Buckling Analyses. Natural Frequency, Static and Dynamic Analyses of Cable elements. Restart of an analysis (available for both Structural and Heat transfer analyses) Batch Option: Analysis can be performed in a batch/command line mode. Access to CAEFEM database through Dynamic Link Library (DLL). Ability to interrupt and change parameters during execution. 64 bit file addressing to handle file sizes greater than 2GB

Concurrent Analysis Corporation
50 Via Ricardo, Thousand Oaks CA 91320, USA
Tel. 805 375 1060, Fax. 805 375 1061
Web: http://www.caefem.com
Email: info@caefem.com