Chapter 1

Overview

Chapter 1 Overview

VisualFEA is an integrated software for finite element analysis, which is an advanced numerical technique to solve and analyze physical problems arising in many fields of science and engineering. Numerous commercial or academic programs for finite element analysis have been developed and distributed all over the world so far. However, most of them are not much accessible for many potential users of the method, owing to various reasons: complexity of usage, high expenses, restricted portability, functional limitations and so on. The main objective of VisualFEA is to overcome such barriers between the user and the software, and make itself easily accessible and affordable for everyone who needs finite element analysis not only for practical use but also for educational purposes.

The greatest advantage of VisualFEA is its ease of use and user friendliness. Its usage is simple, natural and intuitive for all the diverse and ample functionality it provides. This makes the software most unique and attractive, because finite element analysis involves complicated procedures, and accordingly finite element programs in general are far from user-friendly.

Another advantage of the software is its functional integration. All the functions necessary for finite element analysis are integrated into a single executable module of VisualFEA. Thus, the whole procedure from preprocessing to analysis, and to postprocessing can be completed on the spot without launching one program after another, or without pipelining data from one program to another.

VisualFEA is a full-fledged software with many easy-to-use but powerful features of finite element modeling. Its preprocessing capability includes the most advanced 2 and 3-dimensional mesh generation techniques. Finite element modeling is facilitated by new and creative methods of user interfaces devised in this software. Its preprocessing capability is flexible and resourceful. VisualFEA can process linear static, dynamic and nonlinear analysis of structures, and steady state analysis of heat conduction. In addition, the software is open for external solvers or processors so that they can be easily attached to VisualFEA and can exploit its pre- and postprocessing capability.

VisualFEA has many useful and spectacular functions of postprocessing. The data generated by the finite element analysis are visualized in various forms: contour, isosurface, vector, animation, and so on. Furthermore, the model can be freely split or transformed for more elaborate and understandable visualization.

Another uniqueness of VisualFEA is its educational capability. VisualFEA has a number of functions which are useful for teaching and learning finite element analysis. Many concepts and processes in finite element methods can be understood easily by the aid of these educational functions.

VisualFEA is aimed for the masses in finite element method, ranging from serious practitioners to students. VisualFEA will make the finite element method not only a useful analysis tool but also a fun for many users.

Finite Element Analysis and VisualFEA

The outline of finite element analysis procedure is briefly introduced, and characteristics of VisualFEA usage are described in this section. The differences between VisualFEA from other finite element analysis software are stressed in view of their software constructions and modeling procedures.

Procedure of finite element analysis using VisualFEA

The whole procedure of finite element analysis can be processed consecutively by a single executable module of VisualFEA. However, there is an alternative to split the procedure into a few stages and process each stage separately using different programs, especially employing external solver for main computation of the analysis.

■ Stages of finite element analysis

The general procedure of finite element analysis can be split largely into 3 stages: preprocessing for preparation of modeling data, processing for assembly and solution of the equations, and postprocessing for visualization of analysis results.

- Preprocessing: The stage of preparing input data for finite element analysis is called “preprocessing.” The geometry of the finite element model is defined in this stage. The attributes and the various conditions are also applied to the model.
  - definition of outlines and surface primitives
  - division of the outlines to adjust the density of the mode
  - finite element mesh generation
  - assignment of element properties.
  - assignment of boundary conditions.
  - assignment of load conditions (only for structural analysis).
  - node number or element number optimization.
- Processing: This is the kernel of finite element analysis. The finite element equations are assembled and solved at this stages, and thus, the analysis results are obtained. This computational procedure is automatically handled by the computer, and requires user interactions. The computation is carried out in the following sequence.
  - computation of element stiffness matrix and equivalent nodal forces
  - assembly of system equations
  - solution of the assembled equations
  - recovery and smoothing of secondary variables
- Postprocessing: Postprocessing is the stage visualizing graphically the

analysis results and presenting them in more understandable forms.

- additional processing or transformation of analysis results
- visualization of analysis results
- handling of various output

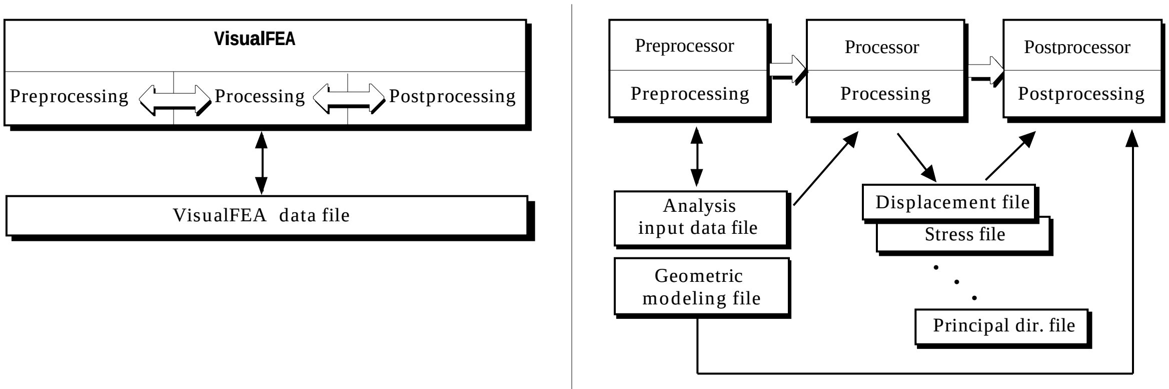
Some stages and procedures may be added or dropped depending on the analysis subject and the project contents.

■ Finite element analysis using VisualFEA

Finite element analysis systems are generally composed of separate programs for each of the above 3 stages, and they are called preprocessor, solver and postprocessor respectively. On the other hand, VisualFEA has all the components integrated in one executable program. Thus, the whole procedure of finite element analysis can be completed by a single execution of VisualFEA. It is not necessary to pass the data from one program to another. There is only one data file for a project. All the modeling and analysis data are stored to and retrieved from the file. Once the file is opened, all data related with the finite element analysis are available for internal processing.

In some cases, the stages are not clearly separated, but intermixed with each other. It is possible to go back and forth freely between two different stages. In frame analysis, for example, the computed results are displayed graphically right after its geometry or attribute data are altered. In such cases, users can hardly distinguish the stages.

The software and data constructions are compared between VisualFEA and other programs. VisualFEA has a monolithic construction for both software and data, while most of other programs have complex constructions with multiple programs and a number of data files. Such a simple construction makes usage of VisualFEA simple and efficient.



intuitiveFEM

Other finite element softwares

< Comparison of software and data construction >

■ Mixed use of VisualFEA and external solvers

Although the entire procedure of finite element analysis can be completed by a single execution of VisualFEA as described above, there is an alternative procedure in which an external solver is used together with VisualFEA .
VisualFEA has powerful pre- and postprocessing capability, but can handle only relatively limited types of analysis. Therefore, it is sometimes desirable to accomplish pre- and postprocessing using VisualFEA, but adopt an external solver for analysis. In this case, input data for the analysis is created by VisualFEA, and supplied to the external solver, and the results computed by the solver are again retrieved to VisualFEA for graphically visualization. If existing third party software is used, an additional program for data interface between VisualFEA and the solver may be necessary. In case the solver is developed by the user, the computed results may be written directly in the VisualFEA file, so that neither extra files nor interface data files are necessary. The data interface with external solvers is described in detail in Chapter 10.

Creating a finite element model using VisualFEA

A finite element model designates an aggregate of data as a subject of finite element analysis, and typically consists of the following data items:

- node data : node identification, nodal coordinates
- element data : element-node relations, connectivities
- attributes : element properties, integration rule
- boundary conditions : boundary constraints, initial states
- external effects : applied forces, temperature

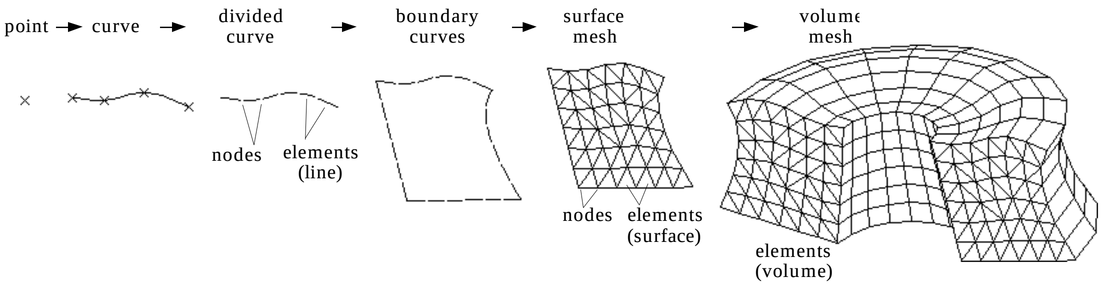
The above items are in fact the input data items used at the processing stage. There are other data items which are necessary only for pre- and postprocessing purposes in VisualFEA.
A finite element model is completed by creating nodes and elements and assigning attributes, boundary conditions and external effects.

■ Node and element data

The geometry of a finite element model is defined by nodes and elements in VisualFEA. An element has a specified number of nodes. The node-element relations are established as a part of a finite element mesh. Some nodes are created by dividing curves used as a seed for mesh generations, and the others are created by mesh generation.
There are line elements for frame analysis, surface elements for plane or shell analysis, and volume elements for 3-dimensional solid analysis. Line elements are

created by dividing a curve, or by direct line drawing. Surface elements are created by surface mesh generation. A surface mesh is generated using boundary curves or seed curves. Volume elements are created by volume mesh generation which are usually derived from surface mesh.

A mesh has a hierarchical structure in VisualFEA. The hierarchical construction of a mesh is schematically represented in the following figure.



< Hierarchical construction of line, surface and volume elements >

■ Other data assignment

A finite element model is geometrically embodied by nodes and elements. But it is not complete unless proper data are assigned to nodes and elements. The finite element model becomes ready for analysis, only when the following data items at least are assigned, depending on the analysis subject.

- structural problem: element properties, boundary conditions(initial displacements), load conditions.
- heat transfer analysis: element properties, heat boundary condition.

Visualizing analysis results using VisualFEA

The data obtained as the result of finite element analysis are very extensive, and therefore can hardly be grasped unless they are graphically visualized. VisualFEA has a number of functions to visualize the analysis results by graphical image. There are two types of images. One is surface images for 2 or 3 dimensional continua, and the other is diagram images for trusses and frames. The continuum data can be split into two types: scalar data which have only magnitudes, and vector data which has directions as well as magnitudes. Different methods of visualization are applied depending on the type of the data.

Visualization of analysis results is the most spectacular part of VisualFEA.

■ Visualization of scalar data

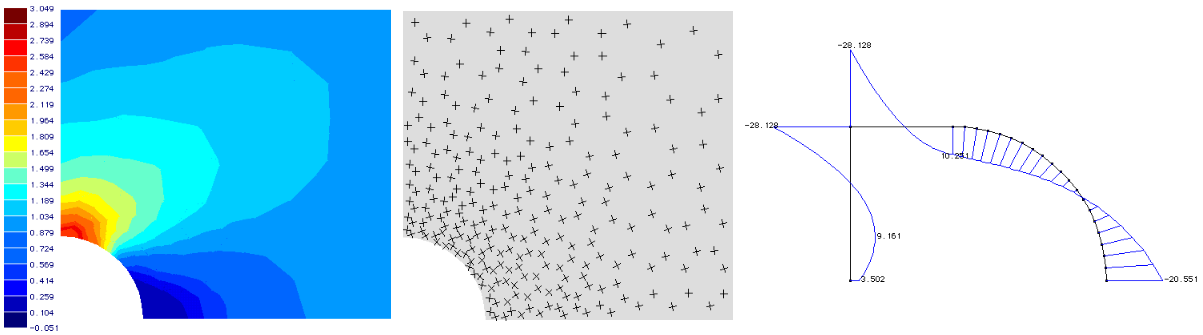
Stresses, strains and temperatures are examples of scalar data obtained from finite element analysis. The most widely used method of visualizing scalar data is contouring. The data distribution is represented by a number of contour lines or bands. However, contouring is limited to representing the data on planes or surfaces. Data distribution within 3 dimensional volumes cannot be represented directly using contours, but can be visualized properly with the aid of volume visualization tools. Volume visualization can also be achieved by iso-surface rendering. There are other methods of visualizing scalar data such as graph and level surface representation.

■ Visualization of vector data

Examples of vector data are displacements, reaction forces, heat fluxes and principal directions. The most widely used method of visualizing vector data is arrow representation. Different types of arrow representation are available for planes, curved surfaces, or 3 dimensional volumes. Vector data like displacements can be represented effectively by deformed shape of the model.

■ Visualization of truss and frame data

The analysis results of trusses and frames are best expressed by diagram images. Member forces such as bending moments, shear forces and axial forces are represented by diagrams. Diagrams are drawn in continuous curves to represent the variation of the values across the frame members.



Visualization of scalar data

Visualization of vector data

Visualization by diagram

< Different methods of visualizing analysis results >

Framework of VisualFEA

Functions of VisualFEA can be classified largely into 5 groups in relation with various stages of finite element analysis : basic user interface, preprocessing, processing, postprocessing and educational aids. The functions of each group are split into a number of sub-groups by their relationship and similarity. The framework of VisualFEA has been designed on the basis of this grouping, and the tool palette and menu commands are also arranged as such. The contents of this manual are also written largely following the framework.

Basic user interface

Functions of basic user interface are those related chiefly with basic usage of VisualFEA, such as inputting geometric data, setting work environments, interactive viewing control, file handling and other functions related to model handling.

■ Project and file

Functions of this category are related with project and file such as starting and quitting VisualFEA, and opening, closing, saving files, printing, setting preferences and so on.

- Starting and ending VisualFEA
- File opening, closing, saving
- Printing
- Preference setting

■ View control

View control functions handle the view of finite element model rendered on the screen. The screen view of the model can be controlled interactively. The user can also set the rendering modes and states as desired.

- Zoom in/out (instant zoom, gradual zoom), fit-to-window, rubber-band zoom
- Panning, scrolling, centering
- View rotation (by virtual trackball, by bounding box), setting view direction
- Perspective/parallel projection
- Mono/stereo projection
- Setting light source
- Hiding and showing specified objects
- Aerial view

■ 3-D geometric data input system

The 3-dimensional grid and cursor system in VisualFEA is implemented as a highly user-friendly and effective way of inputting 2 and 3-dimensional geometric data. These functions are related with inputting the geometric data and setting the grid and cursor system.

- Setting grid, grid on / off, moving grid planes, resizing grid planes, sub-grid
- Simultaneous keyboard / mouse input conversion
- Snap mode on / off(to node, to grid point, to control point, to curve end)
- 3-D cursor

■ Model selection and control

Various objects composing a finite element model need be selected for various reasons. The following functions are provided for selection, modification and other operations of the model and its component object.

- Selection of curve, element, node, surface mesh, volume mesh, surface primitives
- Curve-curve intersection, surface-surface intersection
- Projection, duplication, mirroring
- Reshaping curve and surface primitive
- Moving, resizing, rotating mesh

■ Model rendering

The finite element model can be rendered in a few different forms chosen by the user. This rendering modes can also be applied for displaying the analysis results such as deformed shape. VisualFEA supports the following rendering modes.

- Wireframe
- Hidden line removed wireframe
- Shading
- Transparency shading
- Outline-only display
- Broken mesh

Preprocessing

Preprocessing functions are to create, edit and check the modeling data necessary for finite element analysis. The data are constructed in a few steps: creation of boundary curves and surface primitives, mesh generation and data assignment. The preprocessing functions are sub-grouped based on these steps.

■ Curve and surface primitive modeling

To construct the geometry data of finite element analysis model, curves and surface primitives must be created first. They are used as constraining objects for mesh generation. There are several types of curves and surface primitives as listed below. Each type is associated with an independent function activated by a tool command, and has a number of options which appears as items in the corresponding menu.

- Straight line
- Circle / circular arc
- Ellipse / elliptic arc
- Parametric curve: cubic spline, B-spline, Bezier, polynomial
- Segmented curve
- Sphere
- Cylinder / cone / truncated cone
- Torus
- Parametric surface: plane, B-spline, Bezier surface

There are also functions related with devising curves which is necessary for mesh generation.

■ Mesh generation

The functions of mesh generation are the most sophisticated, and the most important part of VisualFEA. Each of the mesh generation methods is associated with an independent menu command. The methods are classified into a few groups based on the type of user interaction and the type of generation algorithm:

- Automatic mesh generation
  - Planar mesh by triangulation (triangular , quadrilateral , or mixture)
  - Surface mesh by triangulation (“)
  - Triangulation on surface primitives(“)
  - Tetrahedronization of volume(4 node or 10 node tetrahedron)
- Coordinate mapping
  - Surface mesh by 2 edge mapping (lofting)
  - Surface mesh by 3 edge mapping (triangular mapping)
  - Surface mesh by 4 edge mapping (transfinite mapping)
  - Volume mesh by 9 edge mapping
  - Volume mesh by 6 edge mapping
  - Volume mesh by 12 edge mapping
- Sweeping
  - Extrusion
  - Translation

  - Revolution
  - Twisting
- Mesh Editing
  - Dragging node
  - Remeshing
  - Crack tip element

■ Data Assignment

A finite element analysis model is completed by assigning necessary data to nodes and elements. There is an independent function to assign each type of data. The data can be assigned, removed, edited and checked interactively. There are also independent auxiliary functions related with data assignment.

- Boundary condition
  - Fixity defined in local or global coordinates
  - Initial displacement
  - Spring defined in local or global coordinates
- Element property
  - Material property
  - Type definition (for mixed structures)
- Load condition
  - Nodal, point, uniform, trapezoidal, parabolic, moment, body force, hydrostatic
  - Thermal load, self-straining
  - X,Y,Z axis direction, normal direction, tangential direction
  - Load combinations
- Dynamic motion
  - Displacement
  - Velocity
  - Acceleration
  - Nodal dashpot
  - Nodal mass
- Member joint
- Heat boundary condition
  - Temperature
  - Heat flux
  - Heat source, sink
  - Convection
- Seepage boundary condition
  - Open head
  - Confined head

  - Flux
  - Nodal source
  - Initial water table

■ Others

Various objects created in preprocessing stage can be manipulated by object operations as listed below. Node and element numbering can also be optimized to improve the computational efficiency of processing. The procedures can be done prior to processing.

- Object operations
  - Duplication
  - Mirroring
  - Rotation
  - Scaling
  - Projection
- Number optimization
  - Node number optimization
  - Element number optimization

Processing

Processing is the kernel of finite element analysis. Processing does not require user interaction and proceeds with various stages of computing element equations, assembling system equations, solving them, and executing other related computations. The solver of VisualFEA can process structural analysis and heat conduction analysis.

■ Structural analysis

VisualFEA can process the following types of elements. Different types of elements can be mixed within a project. The computed results includes displacements, stresses, strains, bending moments, member forces and etc. Actual items vary depending on the analysis subject.

- Plane stress / strain
- Axisymmetric
- Plate
- Shell
- 3-D solid
- 2-D truss
- 3-D truss

- 2-D frame
- 3-D frame
- Interface
- Embedded bar

The current version of VisualFEA can process the following types of structural analysis:

- Material nonlinear analysis
- Dynamic analysis
- Sequential analysis
- Adaptive analysis

Multiple load combinations can be supplied as input data, and the results of arbitray combination can be obtained.

■ Heat conduction

VisualFEA can process stead-state heat conduction analysis in 2 or 3 dimensional space. Temperature distribution and heat fluxes are obtained as the analysis results.

- 2-D plane
- Axisymmetric
- 3-D volume

■ Seepage

VisualFEA can process unconfined ground water model, termed here as seepage problem. Open or confined head boundary conditions, initial water tables can defined and assigned to the model. The analysis results include water head distribution, pore pressure, phreatic surface, flow path, velocity and discharge.

- 2-D seepage
- Axisymmetric
- 3-D volume seepage

Postprocessing

Functions of this group are used for graphically visualizing or further processing the computed results of the analysis to facilitate their interpretation and understanding. Data in a 3 dimensional volume are effectively visualized with the aid of volume visualization tools. There are also other functions for handling and printing the visualized images.

■ Data visualization

VisualFEA can visualize the data in various forms as listed below. The most appropriate form of visualization can be determined in consideration of the spatial dimension and other characteristics of the data. Two or more methods may be used together to represent two or more data items in a single image.

- Contour
- Graph
- Diagram
- Data level surface
- Iso-surface
- Vector
- Deformed shape
- Nodal resultants
- Animation

■ Visualization aids

There are tools to produce more effective visualization of 3 dimensional data. Volume visualization tools are useful for visualization of data inside 3-dimensional volume. These visualization aids can be set interactively as desired, and are used to cast the images.

- Parallel planes
- Cross planes
- Slicing
- Capping

■ Image handling

The images visualizing the data are sometimes necessary to be stored or retrieved for later reviewing or processing such as overlaying with other images. Screen capturing is also an efficient and simple way of exporting the image to other software.

- Saving image
- Reading image
- Overlaying image
- Capturing image

■ Printing

The text of the computed results as well as their graphic images visualized on the screen may be printed in hard copy. The quality of the printed image depends on the color capability and the resolution of printer.

- Text
- Screen image

■ Others

The contour band can be marked in more detail up to the screen resolution. The data value can be obtained at any point within the model by specifying the coordinates of the point in 2 or 3 dimensional space.

- Contour marking
- Data sampling in 2-D or 3-D space

Educational aids

VisualFEA has functions for education in finite element analysis and in structural mechanics. These functions can be used as a tool helpful for teaching and understanding various concepts of finite element analysis and structural mechanics.

■ Element stiffness computation

The contents of the element stiffness matrix and other related computation are expanded either in texts of numerical data or in symbolic expressions. The information is organized in hierarchical tree form, and can be expanded selectively.

- Hierarchical expansion of the information tree
- Matching with the global stiffness matrix
- Toggling numerical expression and symbolic expression
- Real time update of element stiffness information in response to geometry or other assigned data of the model.

■ Assembly and solution process

These functions allow the user to examine interactively the procedure of assembling and solving the system equations in a few different forms: full square matrix, upper triangular matrix, band matrix, skyline assembly and frontal solution form. There are functions to display or animate interactively the assembly and solution process of the system equations.

- Manual display of assembly procedure in the order of element numbering, or in the order specified by the user
- Animation of assembly and solution procedure
- Toggling text and graphic mode
- Matching the part of the model and the part of the matrix

- Displaying the overall status of the system equations
- Matrix partitioning, decomposition, back substitution

■ Shape function and interpolation

These educational functions are to visualize graphically the shape functions and their derivatives employed in the finite element modeling. Various characteristics and behavior of interpolation models can be examined interactively. They can be used as a training tool in teaching or learning the concepts and the behavior of element modeling.

- Creating an interpolation model
- Assigning nodal values to the interpolation model
- Displaying a shape function or interpolated values
- Displaying derivatives of a shape function or interpolated values
- Sampling the numerical value at a specified point or integration points
- Setting rendering modes of shape functions and their derivatives
- Manipulating the view of shape function display

■ Eigen mode

The eigenvalues and eigenvectors of the stiffness matrix can be interpreted as a strain energy and corresponding displacement mode, respectively. An eigenvector can be graphically visualized as a displacement mode. The characteristics and the validity of an element can be examined by such a display. The meaning of eigenvalues and eigenvectors can be explored through interactive manipulation of eigen mode display.

- Creating a stiffness model for eigen mode display
- Animation of eigen modes
- Contouring strain-equivalents of an eigen mode
- Changing integration order and instantly updating eigen modes
- Displaying integration points
- Setting rendering modes of eigenvector
- Manipulating the view of eigen mode display

■ Stress recovery and smoothing

Solving the finite element equations gives the values of the primary variables such as displacements in structural analysis or temperatures in heat conduction analysis. Stresses, strains, or heat fluxes are the secondary variables computed from the primary variables. The secondary variables are usually obtained first at integration points and then evaluated at nodal points. This process is called “stress recovery and smoothing.” VisualFEA has an educational function to simulated and visualize graphically the stress recovery and smoothing procedure. The computational aspects and characteristics of the procedure can be studied and understood easily using this educational function.

- Simulating the procedure of stress recovery and smoothing
- Getting details of stress and strain computation
- Comparing the methods of stress smoothing
- Examining intermediate results of stress recovery and smoothing

■ Adaptive analysis process

VisualFEA has the capability of adaptive analysis, and has an educational function to visualize this process. Intermediate stages of the adaptive process can be saved and visually reproduced for step by step examination under user's control. This function is intended to help understanding the principles and methods of adaptive solution procedures.

- Animation of adaptive mesh refinement process
- Comparing computational error and mesh refinement density
- Examining the distribution of energy norm error
- Manipulating the visualization of adaptive process

■ Structural behavior

Structural behavior of trusses and frames can be examined by displaying the real time responses to any modification in the structural model. Structural responses such as deformation and member forces are updated immediately after the geometries, element properties, boundary conditions, or load conditions are changed. Thus, the relation between variations in modeling data and their responses can be examined interactively, and this will help understanding the structural behavior.