

## **Chapter 2**

### **Basic User Interface**



## Chapter 2 Basic User Interface

VisualFEA is a cross-platform software, which supports both Mac OS and Windows system. VisualFEA is based on the standard user interface of each operating system, and thus has native look-and-feel for each of them. Menus and tool buttons are used to issue commands for various functions, but they have separate and independent roles in VisualFEA, while menu and tool commands are usually overlapped in most of other software.

For ease and simplification of usage, all the functions are integrated in one executable module, and all the data for modeling and visualization are contained in one data file. Thus, the entire procedure of finite element analysis can be carried out using only one program and one data file.

VisualFEA provides 3-dimensional graphical user interfaces for inputting geometric data, viewing, and handling objects. Grid planes are used for convenience of inputting coordinates. There are 3 orthogonal grid planes, XY, YZ and ZX planes. The grid planes can be moved, resized, or subdivided as desired for data input. In addition to this, a user can define grid planes in arbitrary direction and at arbitrary position. Three-dimensional cursor is also available as another aid for inputting 3-dimensional coordinate data. An arbitrary point in 3-dimensional space can be set easily using the cursor.

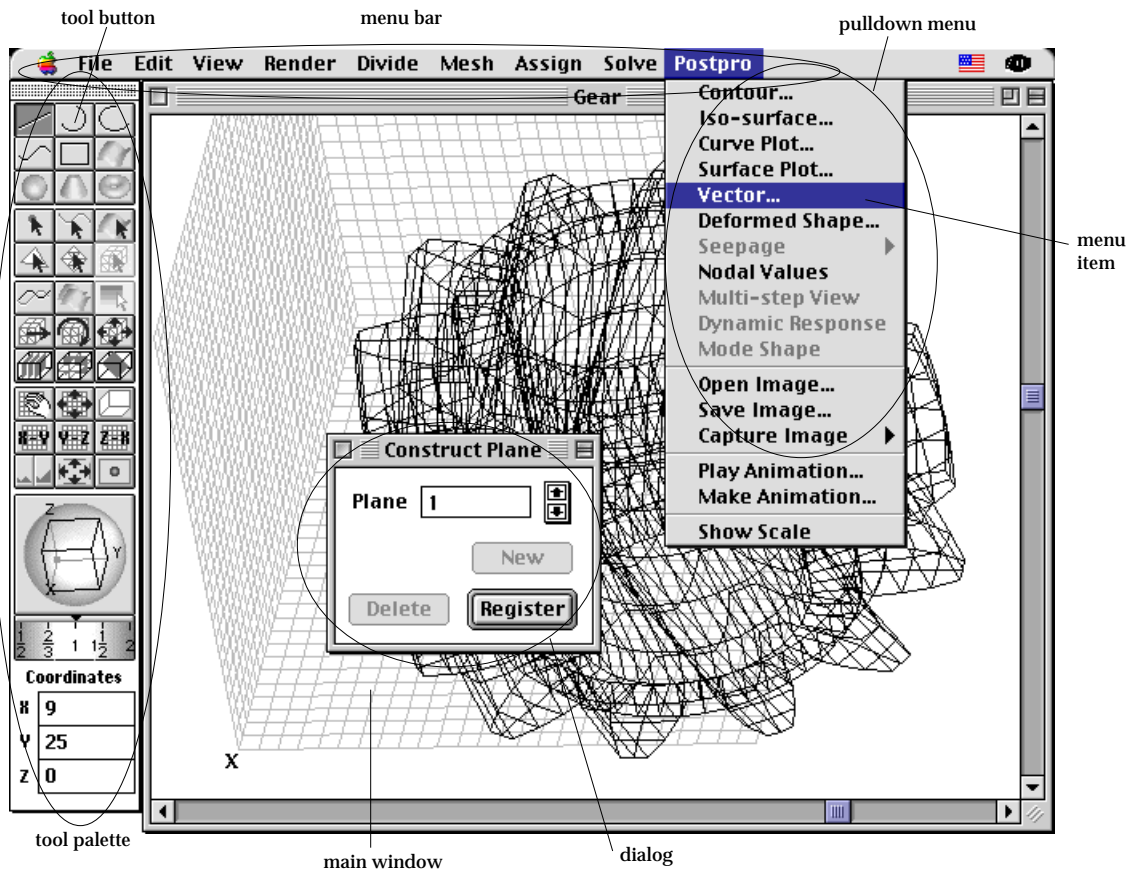
View of the model on the screen can be controlled and adjusted interactively either using the mouse or keyboard input. There are various tools for view transformation including view rotating, zooming and panning. In addition, the direction of view or the part of zooming can be specified directly. Usage of these tools is simple, intuitive, and easy. For example, the view can be rotated by virtual trackball, as if an actual ball were rolled by a finger. The aerial view gives the overall view and facilitates handling of a large and complex model, only part of which fills the whole screen. A desired view can be obtained by combined use of the view transformation tools. A certain view can be marked, saved, or imported. The model can be rendered in various forms: wireframe, hidden-line removed wireframe, outline, and shading with or without transparency.

A finite element model in VisualFEA consists of various objects: curve, primitive surface, node, element, and mesh. There is a selection tool for each type of object. Objects can be selected in 3-dimensional space using the relevant selection tools. Depth of the model or front-and-back relationship is recognized in object selection. There are many other CAD-like features of VisualFEA, which makes the software all the more convenient and easy to use. They are explained in detail in this chapter.

## Basic Elements of User Interface

There are 4 basic elements of user interface with the software: menus, tool buttons, windows and dialogs. The interactions between the user and the computer are taken through one of these elements. Menus and tool buttons are usually used to issue a command starting or executing a function. Windows and dialogs are used for inputting and visualizing data.

The following figure shows a state of VisualFEA under which all the 4 elements are displayed on the screen. There are a number of tool buttons on the tool palette. Menu consists of a menu bar and pulldown menus. A pulldown menu has a number of menu items. Only the main window is shown here, although there are several other windows which appear for certain functions. A type of dialog is shown. But, there are a few other types of dialogs. These are described in more detail in the next section.

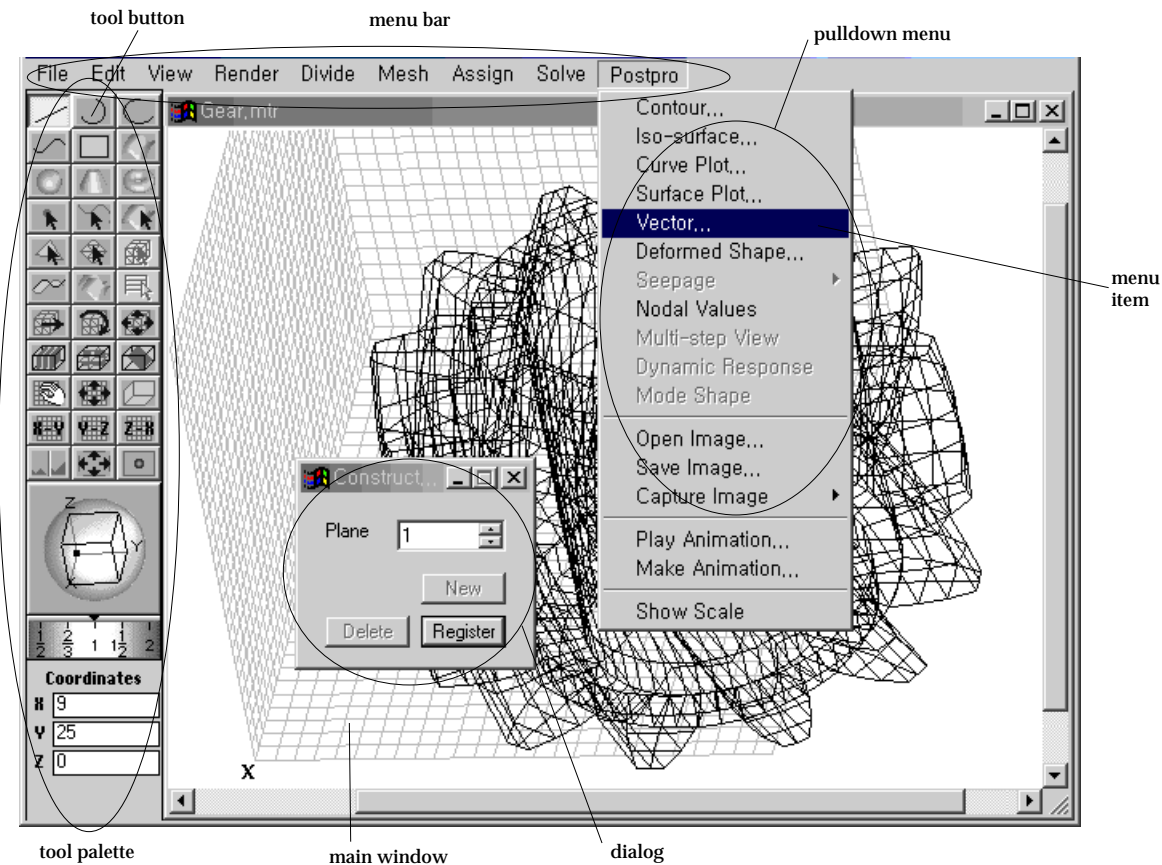


< Basic elements of user interface (Mac OS version) >

## Mac OS and Windows versions

Although Mac OS and MS Windows versions of VisualFEA are working with native processing and with native look-and-feel under respective operating systems, both versions have the same functions with the same usage. The user interfaces of the two operating systems are slightly different in their nature, shape and behavior. And not all the elements of the two have one-to-one equivalence. However, all the menus, tool buttons, windows and dialogs of the two versions are arranged in as much similar fashion as possible. The figures shown in this and the left pages illustrate one-to-one equivalence of the two versions.

In order to avoid overlapping, the illustrations are presented with only the Mac OS version for the rest part of this manual unless there are any worth-mentioning differences between the two versions.



< Basic elements of user interface (Windows version) >

## Menu and tool button

There are two types of user interfaces to launch or execute functions of VisualFEA: menu and tool button. They are used in most software with graphical user interfaces. Therefore, it is not necessary to explain what they are. Instead, it is worth mentioning that their characteristics and roles in VisualFEA are distinctively separated. Menu is generally used to start or execute a single operation, while tool palette is used for sustained operation. For example, intersecting two curves is a function achieved by one time execution, and is activated by selection of a menu command. On the other hand, creating a spline curve is a sustained operation requiring a continuous input, and thus launched by clicking a button in tool palette.

Roles of menu and tool palette are completely separated. That is, no menu commands and tool button commands are overlapped with each other. However, a tool palette command may be launched while a menu command is being executed, and vice versa. In fact, tool button and menu commands are separated chiefly in order to enable activation of multiple functions at the same time.

### ■ Menu

Menu consists of a menu bar and pulldown menus. The menu bar has a number of head menu items. The menu bar items are changeable depending on the command in action, but the following items are always shown:

- File : **File** menu has items related to opening, closing, saving, and printing files. This menu also has commands for getting information or updating the status on the current file.
- Edit : **Edit** menu has editing commands including duplicating, intersecting and linking objects. There are many other commands for inputting and editing data.
- View : **View** menu has commands related to grid setting and view transformation of the main window.
- Render : **Render** menu has the options related to the method of rendering the model, and the method of view projection.
- Divide : **Divide** menu has the command to divide curves for mesh generation, and items to set curve divisions and weights.
- Mesh : **Mesh** menu has the commands to generate surface and volume meshes by various mesh generation schemes.
- Assign : **Assign** menu has the commands related to assigning attributes and various conditions to the model.
- Solve : **Solve** has the command to execute the finite element analysis of the model. The menu also has other items related to finite element processing

and education of finite element method.

- Postpro : **Postpro** menu has the items related to visualization of the analysis results.

A menu command is issued by choosing the corresponding menu item. There are menu items with alternative commands, which can be invoked by pressing *option* key when choosing the menu item. Some menu items are disabled when the command cannot be issued. Some items have a submenu which also has menu items.

### ■ Tool button

The tool palette contains buttons and editable text boxes. Various functions or commands are executed or launched by pressing one of these buttons. The selected button is highlighted for indication of its pressed state. Always one and only one button is highlighted. There are 6 parts in the tool palette as follows:



- Input tools: create curves or primitive surfaces by inputting the coordinates of their control points.
- Selection tools: select objects including curves, primitive surfaces, nodes, elements, and meshes.
- Object operation tools: do various operations such as reshaping, moving, rescaling, or rotating the selected objects. Visual aids such as contour marks, parallel planes, cross planes, or cutting planes can also be set by using one of these tools.
- Grid tools: manipulate the grid system. Each one of the xy, yz, and zx grid planes can be moved, resized, or turned on and off by using the grid tools. Three dimensional cursor may be popped, and be set at the desired location in 3-D space.
- View transformation tools: change the view of the models in 3-dimensional space. Virtual trackball to rotate the view direction, zoom dial to zoom in or out, and instant zoom buttons are included.
- Data edit texts: are used for keyboard input of coordinates, angles, and other digital values relevant to the action in progress.

The tool buttons are displayed in one of the following 3 states:



- Pressed button : The command associated with this button is now in action. There is always one and only one pressed button in the tool palette.
- Active button : This button is not pressed, but can be pressed to launch the function associated with this button.
- Inactive button : This button is disabled, and can not be pressed.

Activation and deactivation of the tool buttons are automatically determined by VisualFEA, as appropriate for the current state of action.

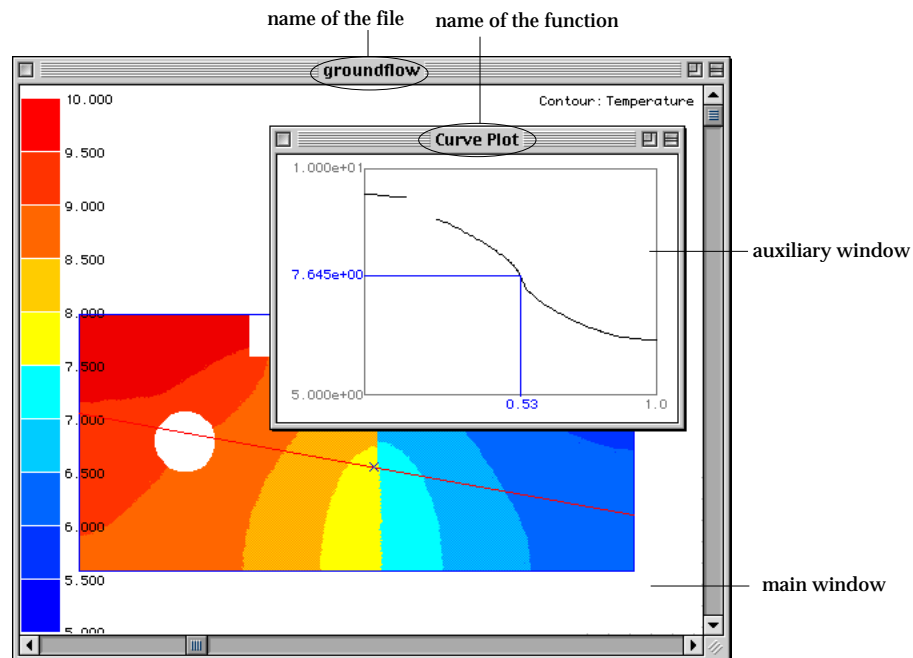
## Window and dialog

Window and dialog are another elements of user interface. Their characteristics are different from those of menu and tool button described in the previous section. Window and dialog are the media for data input and output, while menu and tool button are the means of issuing commands.

### ■ Window

Window is the work area for modeling as well as for visualization. User interactions such as inputting model data take place within the window, and the image of the model and the analysis results are also displayed in the window.

There are two kinds of windows: the main window and auxiliary windows. There is only one main window, and the main window is always opened while a process is going on. The main window has a title identical to the name of the file currently opened. There are a number of auxiliary windows. Each one of them is associated with a certain function, and opened only while the function is being used. An auxiliary window bears the title identifying its function. Auxiliary windows are shown always in front of the main window. Closing the main window closes the file in the current session. On the other hand, closing an auxiliary window terminates the function associated with the window. Terminating the function will also close the auxiliary window automatically.

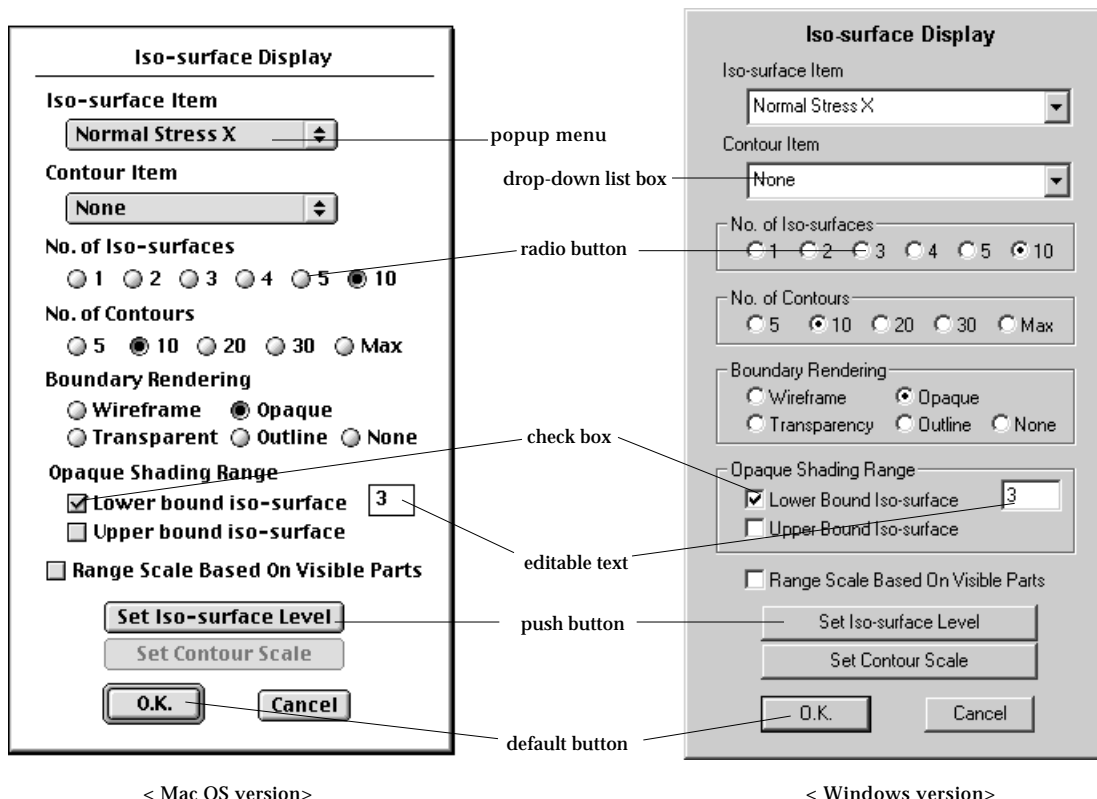


< Main window and auxiliary window >

## ■ Dialog

Dialog is similar to window, but different in its function. Dialogs are usually used for numerical data input or selection of options, while windows are used for graphical data input. A dialog has one or more items for user interaction, which can be classified into the following types:

- Radio buttons: occurs in groups. They are mutually exclusive. Exactly one button in the group is on at any given time.
- Check boxes: act like toggle switches. Use check boxes to indicate the state of an option that must be either on or off. The option is on if the box is checked.
- Editable texts: are used to input string or numerical data.
- Push buttons: are labeled with text. Pressing a push button performs the action described by the button's label. There is one default button which can be activated by pressing **return** key (Windows : **Enter** key). The default button is visually distinguished from the other buttons by its thick outline.
- Popup menus: are used for setting values or choosing from lists of related items. Popup menus are used only in the Mac OS version.
- Drop-down list box: are used only in Windows version and equivalent to





< Mac OS version >

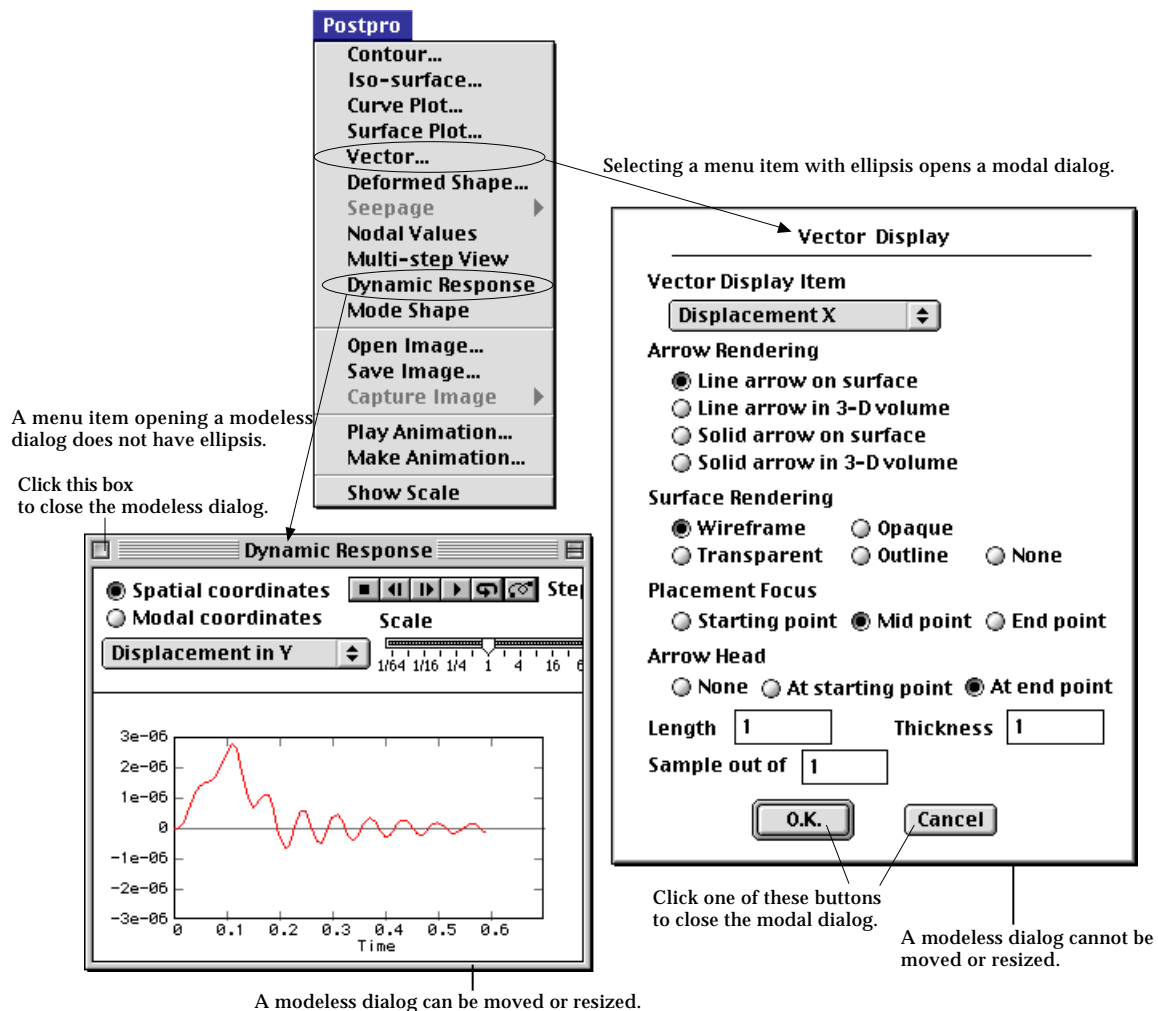
< Windows version >

< Dialog items >

popup menus of Mac OS version.

There are two types of dialogs : modal dialog and modeless dialog. Modal dialog is the one which demands exclusive input to the dialog. In other words, any other action cannot be taken except responding the dialog, while the dialog is open. Modeless dialog does not require exclusive response. Thus, modeless dialogs are used for sustained operation, such as mesh generation for example. Any action related with mesh generation can be taken while the dialog is open to receive user input.

A modal dialog is usually opened by a menu command ending with ellipsis(...), and closed by clicking a button such as  or . A modeless dialog is opened by a menu command without ellipsis, and closed by clicking the close box in the dialog, or terminating the associated function. Certain dialogs are opened by tool button commands. Functions with a modeless dialog always take sustained operations.



## Project and File

A “project” implies a finite element analysis session with a specific modeling and analysis data set created by VisualFEA. The data set for a project is stored completely in a single file. This file is termed here as VisualFEA file or project file. A project can be closed and resumed by saving and opening its project file. VisualFEA can work with only one project at a time.

### Working with a project

To work for a project, launch VisualFEA, and start a new project or open an existing project file. Then, a finite element analysis model can be created, modified, solved, or visualized. The modeling and analysis data worked for a project are saved on user's request or sometimes automatically for later use. Project and file related commands are provided in **File** menu of VisualFEA.

File		
New...	⌘N	Start a new project
Open...	⌘O	Open an existing project file
Import...		Import data from an external file
Close	⌘W	Close the current project
Save	⌘S	Save the project file
Save As...		Save a new project file or save the project file with new name
Save As Text...		Create a text list of the modeling and analysis data
Project Setup...		Setup the project
Update File Status	⌘U	Update the file status
Get Info	⌘I	Get information on the current project
Page Setup...		Setup the page for printing
Print	⌘P	Print the screen image or text
Quit	⌘Q	Quit the program

### ■ Launching VisualFEA

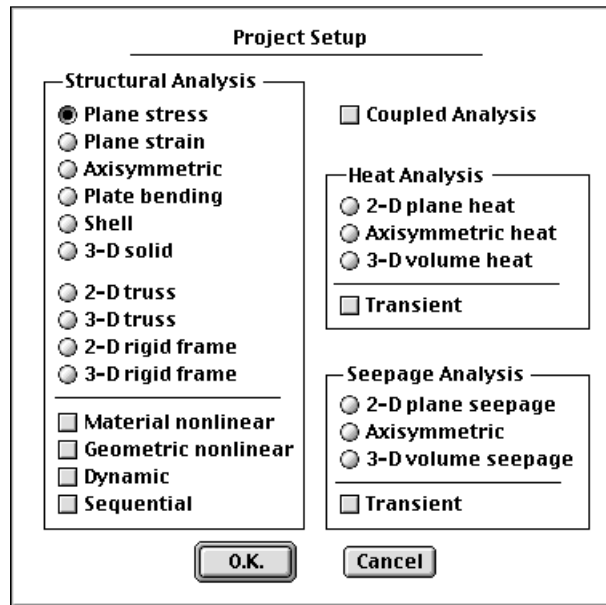
VisualFEA can be started by double clicking either VisualFEA icon or a project file icon. The data file name has extension .mtr for Windows version. In case a project file is double clicked, the file is opened with the launching of the software. This can also be achieved either by clicking one of those icons or by choosing “Open” command from Finder's **File** menu.



< Icons (Mac OS version) >

### ■ Starting a new project

After VisualFEA is launched, a new project can be started by choosing “New...” command from **File** menu. “Project Setup” dialog appears on the screen for defining the analysis type of the project.



< “Project Setup” dialog >

Select the analysis subject using the radio buttons, and set the analysis options using check boxes in the dialog, and click **O.K.** button. Then, the main window is opened and is filled with a grid at its initial state.

There are 3 categories of analysis subjects: structural analysis, heat conduction analysis and seepage analysis. Structural analysis can be coupled with either heat analysis or seepage analysis. Check the "Coupled Analysis" box to initiate the coupled analysis mode. In this case, one subject from structural analysis and another subject from the coupled category should be selected. In the case of uncoupled analysis, only one subject should be selected from all categories.

There are 5 options for structural analysis:

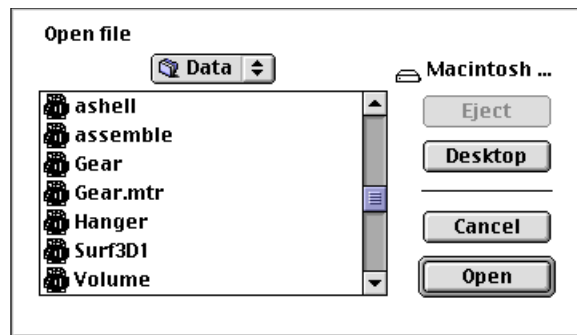
- linear static : Uncheck all the check boxes.
- material nonlinear: Check the "Material Nonlinear" box.
- geometric nonlinear: Check the "Geometric Nonlinear" box.
- dynamic: Check the "Dynamic" box.
- sequential: Check the "Sequential" box.

There are 2 options respectively for heat conduction and seepage analysis:

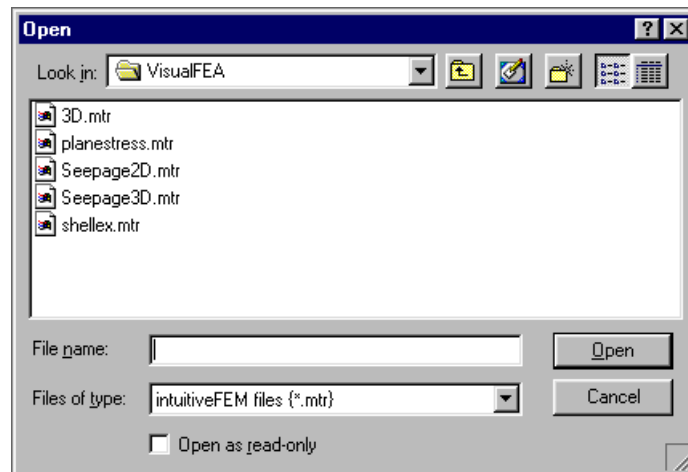
- steady state : Uncheck the check box.
- transient: Check the "Transient" box.

## ■ Opening a project file

A previously saved project file can be opened and worked for further modeling, analysis or visualization. To open a project file, choose “Open...” command from **File** menu. Then, “file opening” dialog appears on the screen. Using this dialog, browse and select the file to open, and click **O.K.** button. The file is opened and the finite element model in the file is restored on the screen just as it was when the file was saved.



< “Open file” dialog (Mac OS version) >

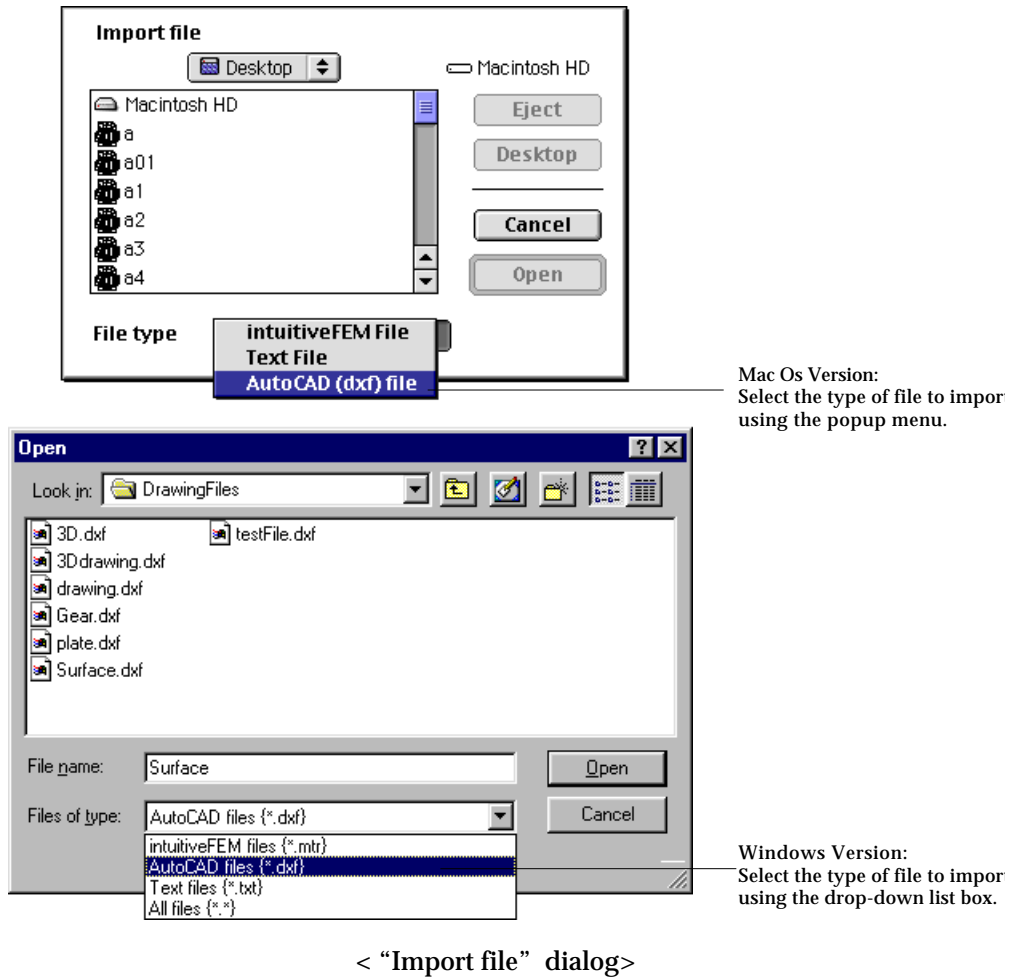


< “Open” dialog (Windows version)>

## ■ Importing an external file

Data can be imported from an external file.

- **VisualFEA file:** Another project file with extension ".mtr". Modeling data including geometry, attribute assignment, load conditions, etc. can be imported from an other project file. The analysis results cannot be imported. The analysis subject of the file being imported should be the same as that of the current project.
- **AutoCAD DXF file:** A file with extension ".dxf". Geometries can be imported from a DXF format file created by AutoCAD.
- **Text file:** A text file with extension ".txt". You may write geometric data in a text file, and import the data from an VisualFEA project.



### ■ Closing the project file

A project file need be closed to quit VisualFEA or to start another project. A project file can be closed in a few different ways:

- by selecting “Close” command in **File** menu.
- by clicking the close box on the top left of the main window.
- by quitting VisualFEA.
- by opening another project file.

Any one of the above actions request closing of the currently working project file. If the file has been modified or solved since last saving, you will be asked whether to save the file before closing.

### ■ Saving the project file

After working with a project, the project file is updated with the modified data by choosing “Save” command in **File** menu. The “Save” item in the menu is

enabled only when the data have been changed since last opening or saving the file. If the project is a new one with no existing file, it will work in the same way as “Save As...” command described below.

### ■ Saving the file with a new name

To save the currently working data in a file other than the current project file, choose “Save As...” command in **File** menu. Then, “file saving” dialog appears. The file name is initially given as “untitled” in the dialog. The name can be changed to the desired one. The data is saved in the file by clicking **O.K.** button.

### ■ Creating a text file with the list of modeling and analysis data

The data of a project can be viewed in text format by creating a text file with extension ".txt". Choose "Save As Text" item from **File** menu, and open the file using a text editing program.

```

TITLE:          untitled
ANALYSIS TYPE:  Structural Analysis (2-dimensional Frame)
SOLUTION TYPE:  Dynamic
PROBLEM SIZE:

  Number of nodes      = 4
  Number of elements   = 3
  Total degrees of freedom = 12
  Structure boundary cond. = 1
  Heat boundary cond.  = 0
  Number of property sets = 1
  Number of load cond.  = 1
  Number of dynamic motion = 0
  Number of temperatures = 0

NODAL COORDINATES AND BOUNDARY CONDITIONS:

  Node   *--- Coordinates ---*   * B.C. *
  No.    X       Y       Set No
  ----
  1      5.0000   9.0000   1
  2      5.0000  18.0000   -
  3     15.0000  18.0000   -
  4     15.0000   9.0000   1

FINITE ELEMENTS AND THEIR PROPERTIES:

  Elem.  Element  Prop. Temper  *--- Node Numbers ---
  No.    Type    Set    Set    1    2
  ----
  1  FRAME2D    1    -      1    2
  2  FRAME2D    1    -      2    3
  3  FRAME2D    1    -      3    4

BOUNDARY CONDITION SETS:

  Set No.    u       v       seta
  ----
  1  0.000e+00(X)  0.000e+00

```

< An example of text list >

### ■ Setting the project's analysis subject

The analysis subject is defined at the beginning by “New...” command, but can be altered in the middle of working with the project. Choose “Project Setup...” command from **File** menu. Then, “Project Setup” dialog appears as in the case of “New...” command. The analysis subject can be defined using the dialog.

### ■ Updating the file status

Some of the menu items are enabled or disabled depending on the status of the project file, especially whether the file contains the analysis data. Therefore, it is sometimes necessary to check whether the file has been updated with analysis data by an external solver since it was opened for the currently working project. This can be achieved by choosing “Update File Status” command.

### ■ Setting up page for printing

Page setup for printing is done with standard “Page Setup” dialog.

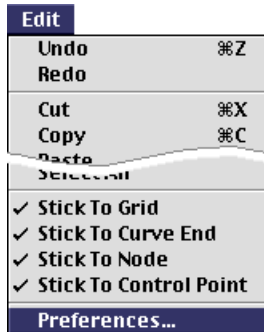
### ■ Printing

In order to print the screen image or text of the project file, choose “Print” command. If there are more than one windows on the screen, the image in the front most window will be printed.

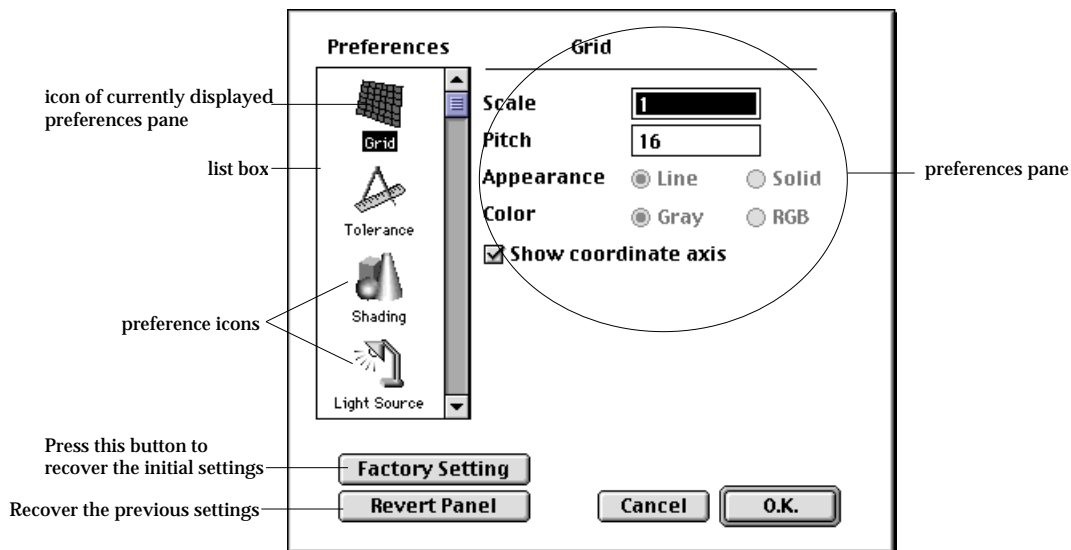
### ■ Quitting VisualFEA

To quit using VisualFEA, choose “Quit” command from **File** menu. All the windows are closed, and VisualFEA will be terminated. If the current project has been modified since opening or last saving, it will be asked to save the project.

## Setting preferences



There are a number of preferences for the environment of VisualFEA. They can be set using “Preference” dialog. To open the dialog, choose “Preferences...” command from **Edit** menu. The dialog consists of a list box and an area for preferences pane. The list box contains the icons representing the preferences items. One of the icons in the list box is highlighted, and the corresponding preferences pane is displayed to the right. The displayed preferences pane can be switched by clicking another icon in the list box. The icons in the list box can be scrolled using the scroll bar.



### ■ Grid settings

The initial state of the grid is determined by grid settings. To bring up “Grid” pane in the preferences dialog, click “Grid” icon in the list box. The pane has the following items:

- “Scale”: The scale of the grid represents the size of one grid cell in the world coordinates. That is the distance between two adjacent grid points. The desired scale can be inserted in the editable text box.
- “Pitch”: The pitch represents the number of pixels between two adjacent grid points in initial view state, i.e. prior to any view transformation. The desired pitch can be inserted in the editable text box.
- “Show coordinate axis”: If this box is checked, the coordinate axis is indicated with the grid.

## ■ Tolerance settings

There are tolerances applied for sticking to grid point, selection range and so on. The tolerance can be set on “Tolerance” pane of “Preference” dialog. The pane can be brought up by clicking “Tolerance” icon in the list box.

- “Input Range”: If “Stick to Grid” item in **Edit** menu is checked, the point of data input sticks to the grid point nearest to the mouse clicked point. But, the distance between the clicked point and the grid point should be within the tolerance range, which is set by the value inserted in “Input Range” text box. If the mouse clicked point is apart from any of the grid points beyond this tolerance, the point will not stick to any of the grid points. It works in the same way for “Stick to Node”, “Stick to Control Point”, “Stick to Curve End” and “Stick to Control Point” options in **Edit** menu.
- “Selection Range”: The nearest object among the front most ones is always selected by a mouse click, when the appropriate selection tool is activated. The selected object need not be exactly on the point of mouse click. There is a tolerance range from the clicked point, within which the nearest object is searched and selected.
- “Intersection Tolerance”: The intersection between curves or between surface primitives are obtained recursively or iteratively. The intersection tolerances set the precision of the recursion or iteration.
  - “Curve-curve” : tolerance of intersection between curves.
  - “Surface-surface” : tolerance of intersection between two surface primitives.
- “Number of iteration”: The precision of the intersection is set by the tolerance. On the other hand, there is also a limit in the number of iterations.
  - “Surface-surface” : The maximum number of iterations is applied only for surface-to surface intersection.
- “Precision of surface primitives”: A mesh can be generated on a surface primitive by automatic triangulation. The nodal points on the mesh are constrained on the primitive surface up to its precision.

Tolerance		
Input Range	<input type="text" value="4"/>	Pixels — range of inputting point sticking to grid point
Selection Range	<input type="text" value="5"/>	Pixels — range of seaching point from the mouse clicked point
Intersection Tolerance		
Curve-curve	<input type="text" value="0.001"/>	— curve-to-curve intersection tolerance
Surface-surface	<input type="text" value="0.01"/>	— surface-to-surface intersection tolerance
# of Iteration for		
Surface-surface	<input type="text" value="7"/>	— number of iteration for surface-to-surface intersection
Precision of Surface Primitive		
	<input type="text" value="0.0001"/>	— precision of surface primitive for mesh generation

## ■ Shading settings

The finite element models are frequently rendered by shading or by contouring with shading. The following options are applied in rendering the models.

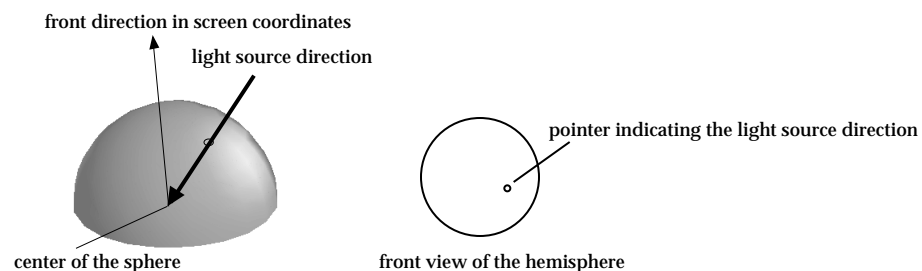
- “No. of Gray Levels”: Shading of an object is represented by gray levels. this option determines how many gray levels are used in shading. This option is also applied to contouring with shading.
- “Rendering Method”: Two methods of rendering are used for shading or contouring in VisualFEA. One is rendering by polygon fill. And the other is by painting pixel by pixel based on “area coherence algorithm.” The former is faster, but the latter produces better quality.
- “Smoothing”: Smooth surface shading is obtained by smoothing the surface gradients at the element or surface boundary edges. If this option is checked, the gradients are smoothed. Otherwise, they are not smoothed.
- “Optimum contrast”: If this option is checked, the contrast of the shading is determined so that the available gray levels are used as many as possible. Otherwise, the contrast are constant so that absolute gray levels are used for a given shading level.
- “Nonlinear intensity variation”: If this option is checked, nonlinear relationship between the angle of light source and the brightness is assumed in computing the shading level. Nonlinear variation sharpens the bright spot.
- “Transparency Model”: The transparency of the model is computed by the surface gradient, the volume thickness, and angle of light sources. But, the computation is based on a few factors provided in this option.
  - “Min.” : Minimum transparency set for the computation.
  - “Max.” : Maximum transparency set for the computation.
  - “Gradient Factor” : Factor of power raised to the surface gradient. The computed transparency is affected by this factor. If this factor is larger, surface gradient has smaller effect on the transparency.

<b>Shading</b>	
<b>No. of Gray Levels</b> _____	number of gray levels used in shading
<input type="radio"/> 16 <input type="radio"/> 32 <input checked="" type="radio"/> 64 <input type="radio"/> 128 <input type="radio"/> 256	
<b>Rendering Method</b> _____	method of rendering image
<input checked="" type="radio"/> By polygon <input type="radio"/> By pixel	
<input checked="" type="checkbox"/> <b>Smoothing</b> _____	Surface gradients are smoothed if checked.
<input type="checkbox"/> <b>Optimum contrast</b> _____	Colors of shading levels are determined so that all the available gray levels are used, if checked.
<input checked="" type="checkbox"/> <b>Nonlinear intensity variation</b> _____	Nonlinear relationship is assumed in computing brightness, if checked.
<hr/>	
<b>Transparency Model</b> _____	minimum transparency in percent
<b>Min.</b> <input type="text" value="80"/> <b>Max.</b> <input type="text" value="90"/>	maximum transparency in percent
<b>Gradient Factor</b> <input type="text" value="3"/>	power factor used in computing transparency

## ■ Light source settings

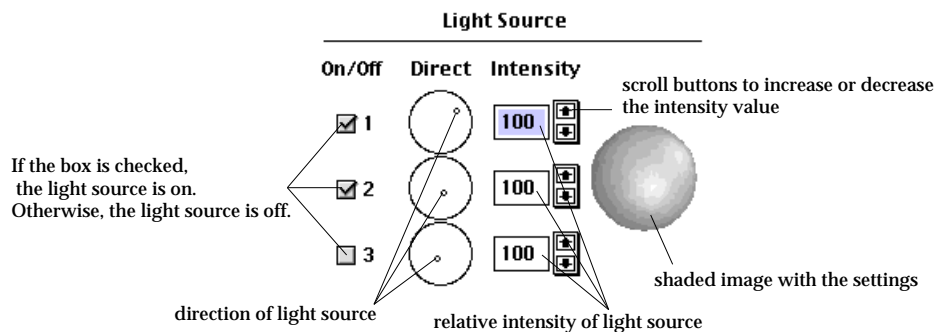
Parallel point light sources are used in shading. There are 3 light sources, with different directions and intensities, each of which can be turned on or off. The light source pane also shows an image of a sphere shaded with the settings.

- “On/off” check boxes: There are 3 check boxes for 3 light sources. If the box is checked, the corresponding light source is on. Otherwise, it is off.
- “Direction” circles: The direction of each light source is mapped on a circle, and indicated by a small pointer. The circle represents the top view of a hemisphere. The pointer can be dragged to any point within the circle. And the direction of the light source is set accordingly.



### < Mapping of a light source direction >

- “Intensity” editable text boxes: The relative intensity of each light source can be set in this text box. The values are in percentage. Each intensity value can be inserted by keyboard input or adjusted by using the scroll button on the right of the text box.
- “Immediate Display Mode” check boxes : If this box is checked, the shaded image is updated immediately after the settings are changed. Otherwise, click the shaded image to update the image.



## ■ View settings

View settings control the initial state of the window and the mode of updating the screen contents. The initial width and height of the window are determined by the default window resolution so that the window is sized to fit within the selected resolution. If the default resolution is set to "Actual", the window size is determined based on the actual screen resolution.

If "Remember the last window size" box is checked, the size of the window at the time of file saving is restored when the file is opened again.

There are several check boxes corresponding to various rendering modes. If the box corresponding to the current rendering mode is checked, the screen contents are redrawn with the rendering mode for every view transformation. Otherwise, the screen contents are updated by wireframe rendering, which produces fastest rendering.

- "Wireframe w/ hidden removal": Rendering of wireframe with hidden line removal is maintained for every view transformation.
- "Shaded image": Rendering of the model by shading is maintained for every view transformation.
- "Transparency": Rendering of the model by shading with transparency is maintained for every view transformation.
- "Contour image": Scalar data representation by contour image is maintained for every view transformation.
- "Iso-surface": Scalar data representation by iso-surface image is maintained for every view transformation.
- "Surface plot": Scalar data representation by surface plot is maintained for every view transformation.
- "Vector image": Vector data representation by arrow image is maintained for every view transformation.
- "Outline image": Rendering of the model by outline is maintained for every view transformation.

View		
<b>Default Window Resolution</b>		
<input type="radio"/> 640x480	<input type="radio"/> 800x600	
<input type="radio"/> 1024x768	<input type="radio"/> 1280x1024	Set default window resolution.
<input checked="" type="radio"/> Actual		
<input type="checkbox"/> Remember the last window size		The window size saved in the file is restored if this box is checked.
<b>Update Items After View Transform.</b>		
<input checked="" type="checkbox"/> Wireframe w/ hidden removal	<input type="checkbox"/> Transparency	
<input checked="" type="checkbox"/> Shaded image	<input type="checkbox"/> Iso-surface	Check the types of screen image to be updated after each view transformation
<input checked="" type="checkbox"/> Contour image	<input type="checkbox"/> Vector image	
<input type="checkbox"/> Surface plot		
<input type="checkbox"/> Outlined image		

## ■ Solver settings

Solver settings have several options applied during the solution process. These options are applied as defaults for subsequent finite element analysis. “Solver” pane can brought up by clicking “Solver” icon in the list box.

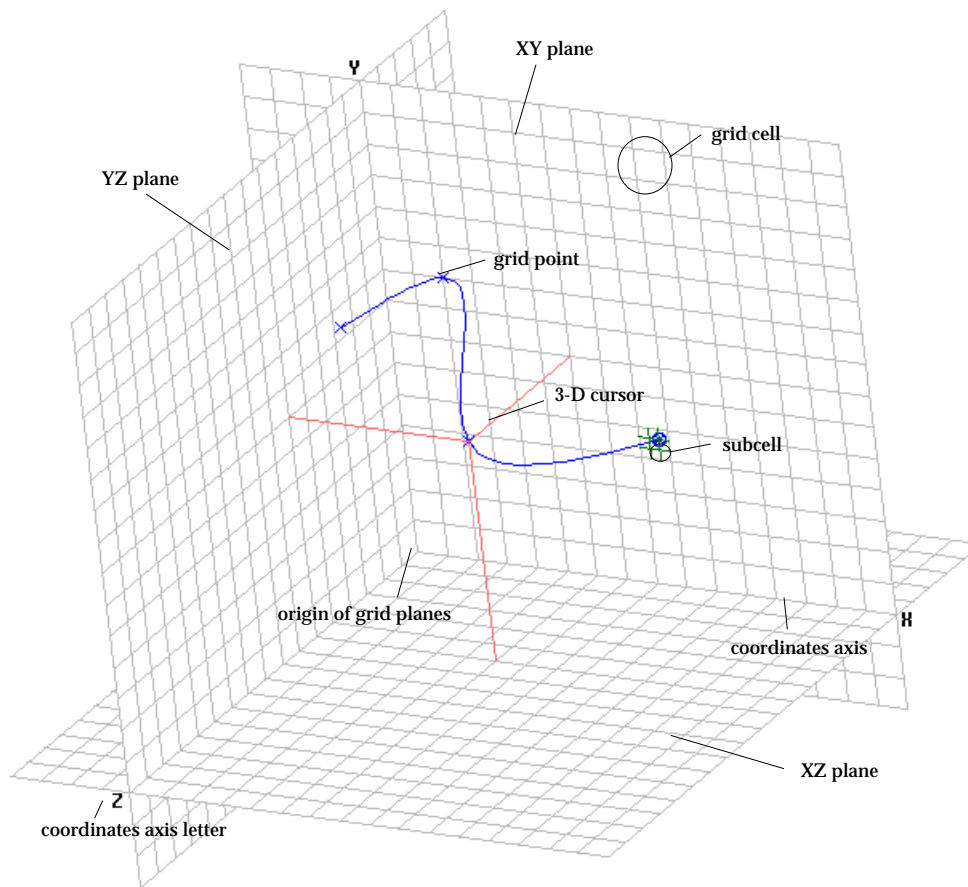
- “Gradient Recovery and Smoothing”: There are 4 radio buttons to select the method of computing secondary variables such as stresses, strains, heat fluxes and so on. For the first 3 methods, the nodal values are averaged between elements connected at the node so that the values are continuous across the element boundaries. But for the last method, the values are not averaged. Accordingly, there appears some discontinuity of the value across element boundaries.
  - “Conjugate smoothing” : Nodal values are computed from the values at integration points by conjugate smoothing algorithm.
  - “Planar extrapolation” : Nodal values are extrapolated from the values at integration points by least squares regression.
  - “Nodal recovery” : Nodal values are directly computed by kinematic relationship for the element.
  - “No smoothing” : The same as conjugate smoothing, but the nodal values are not averaged.
- “Number of Buffers for Frontal Solver”: If the frontal solution method is used in solving the system equations, array buffers are used to store the active equations. One or more array buffer(s) may be used. The number of array buffers are specified in this editable text box.
- “Show progress messages”: If this box is checked, the progress bar and messages are displayed to indicate progress while the solution process is going on.
- “Notice of completion”: If this box is checked, the completion of solution process is notified at the completion of the solution process.
- “Produce the time log file”: If this box is checked, the time elapsed for each stage of solution process is recorded in time log file .

There are other solver options determined using “Analysis Options” dialog.

<b>Solver</b>	
<hr/>	
<b>Gradient Recovery and Smoothing</b>	method of computing stresses, strains, or gradients
<input checked="" type="radio"/> Conjugate smoothing <input type="radio"/> Planar extrapolation <input type="radio"/> Nodal recovery <input type="radio"/> No smoothing	
<b>No. of Buffers for Frontal Solver</b>	number of array buffers used in frontal solution
<input checked="" type="checkbox"/> Progress messages	If this box is checked, progress bar and message is displayed during solution process.
<input checked="" type="checkbox"/> Notice of completion	If this box is checked, completion of solution process is notified on the screen.
<input checked="" type="checkbox"/> Time log file	If this box is checked, the elapsed time for solution is recorded in a time log file.

## Grid and 3-D Cursor

A curve or a primitive surface is defined by control points. Control points are, for example, the end points of a straight line, the center and two end points of a circular arc, or the center of a sphere. Thus, a curve or a primitive surface is constructed by inputting the coordinates of its control points. VisualFEA assumes that all the coordinates are in 3 dimensional Cartesian space. There are two methods of inputting the coordinates of control points, namely one using the mouse and the other using the keyboard.



< Grid system >

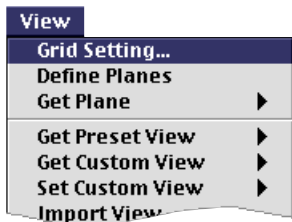
## Grid



Grid planes are used for convenience of inputting coordinates. Only one grid plane (xy plane) is used for 2-dimensional modeling, but three planes (xy, yz, and zx planes) are used for 3-dimensional modeling. Each of the grid planes can be independently moved forward and backward, resized, or turned on and off.

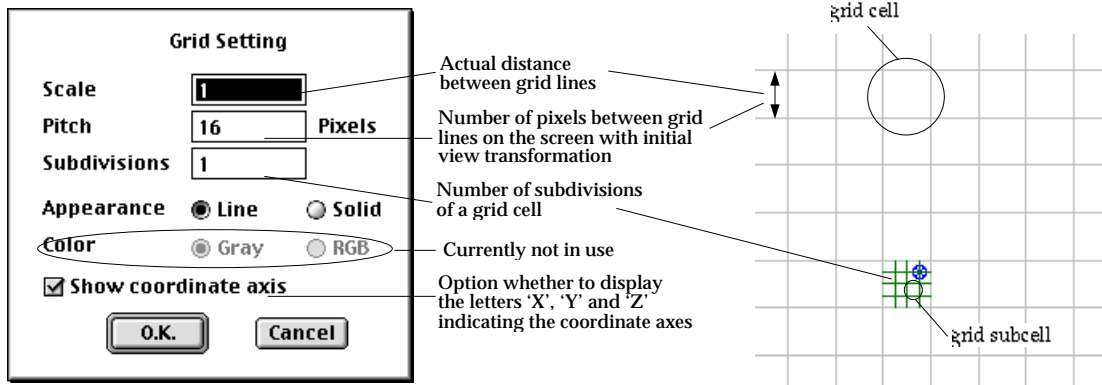
### ■ Setting grid

The properties and the appearance of the grid planes are set by using the "Grid Setting" dialog. The dialog has the following items:









- Scale : the actual distance between two adjacent grid lines
- Pitch : the number of screen pixels between two adjacent grid lines, which is assumed to be counted in the view state without any transformation.
- Subdivisions : the number of subdivisions between two adjacent grid lines. A grid cell is temporarily divided into subcells depending on this value.
- Show coordinate axis : Check this box to display the coordinate axis letters 'X', 'Y' and 'Z'.

This grid setting can also be done by using preference dialog. In this case, the setting is applied as the default setting for future sessions. The number of subdivisions can also be set simply by using F keys, which is explained in "Subdividing grid."



< Grid setting >





### ■ Turning grid planes on and off

All the grid planes are initially on, and the state of the grid planes are indicated by the shape of the tool buttons ,  and  respectively for the xy, yz and zx planes. Clicking these buttons toggle each of the planes on and off. The buttons are shaped respectively ,  and  in off state. Turning off a grid plane not only hides the plane, but also suppresses its function in future operations.

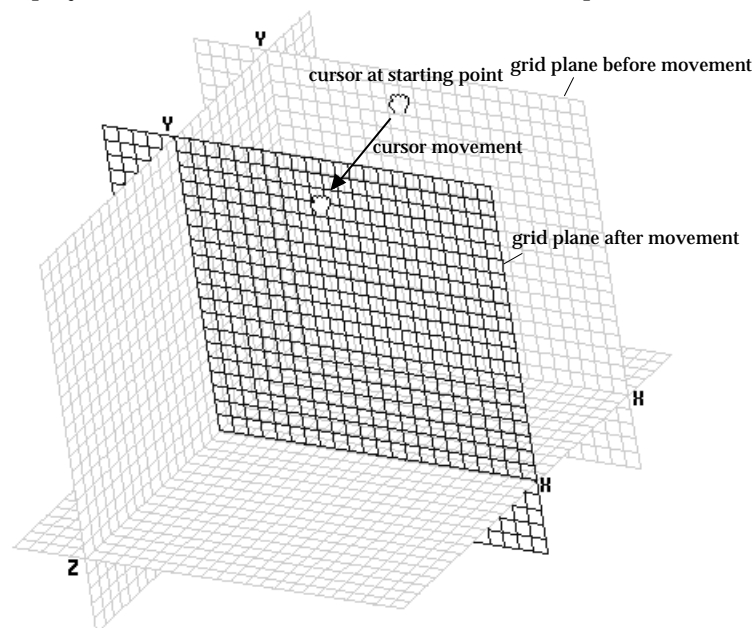
## ■ Moving grid planes



Each of the three grid planes (xy, yz, and zx planes) can be moved independently by using either the mouse or the keyboard. The grid planes can be moved by using the mouse by the following steps:

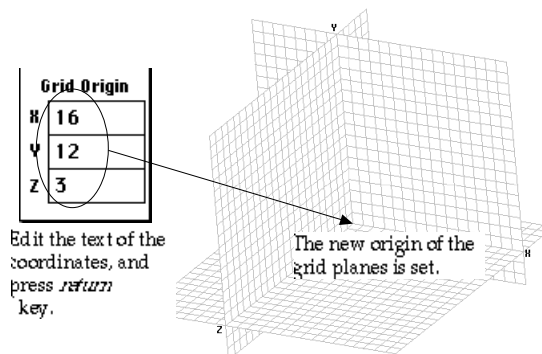
- 1) Start the grid movement tool by pressing the tool button , if it is not yet activated.
- 2) Do the view transformation if necessary.  
It is easy to move a plane facing obliquely to the front of the screen.
- 3) Place the screen cursor over the grid plane to move.  
The cursor shape changes into .
- 4) Press the mouse button.  
The cursor shape changes into .
- 5) Drag the cursor in the direction to move the grid plane.  
The grid plane moves along with the cursor movement.
- 6) Release the mouse button.  
The cursor shape returns to , and the grid plane is settled at the position.
- 7) Repeat 3) - 6) for each of the grid planes until all the grid planes are in the desired position.

While the grid planes are moving, the coordinates of the changing grid origin are displayed in the text box at the bottom of the tool palette.



< Moving grid planes by mouse >

On the other hands, the coordinates of the grid origin can be entered directly using the text box. Each of the grid planes is moved so that new grid origin is formed at the position of the entered coordinates.







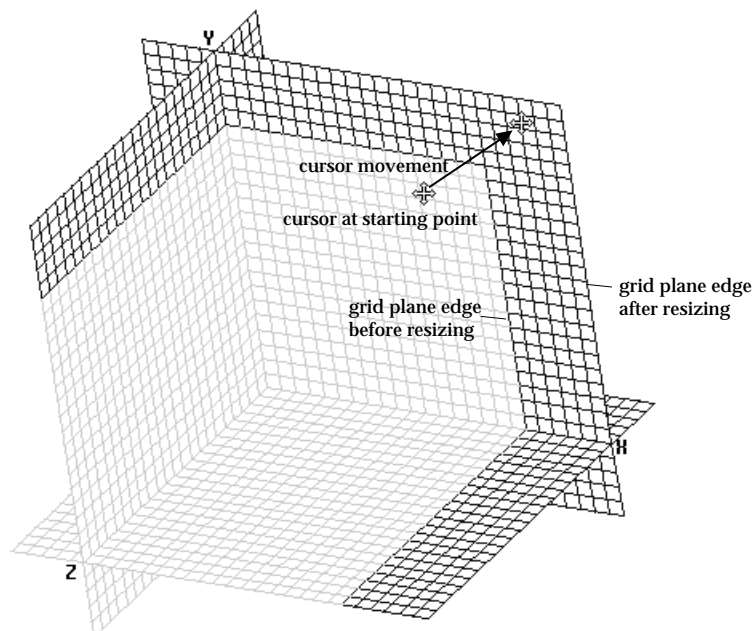
&lt; Moving grid planes by keyboard &gt;



### ■ Resizing grid planes

Each of the three grid planes can be expanded or shrunk to the appropriate size by the following steps:

- 1) Start the grid resizing tool by pressing the tool button .
- 2) Place the screen cursor over an edge of the grid plane to resize.  
The cursor shape changes into .
- 3) Press the mouse button.  
The cursor shape changes into .
- 4) Drag the cursor in the direction to expand or shrink the grid plane.  
The grid plane expands or shrinks along with the cursor movement.
- 5) Release the mouse button.  
The cursor shape returns to , and resizing of the grid plane ends.




&lt; Resizing grid planes &gt;


## ■ Subdividing grid

A grid can be subdivided as shown below. In order to get the subdivided grid, press F keys(F1 - F24). The F number corresponds to the number of divisions. For example, **F3** will set the division into 3, **F12** into 12, and **F1** into 1(no subdivision).

It is convenient to use subdivided grid points in entering a coordinates of a point which does not fall on a grid point, but on a subdivided point. This action is only effective under input or modification mode. Coordinate input using subdivided grid is achieved by the following steps:


- 1) Move the screen cursor near the point of input.
- 2) Press the mouse button.

The grid cell containing the screen cursor is subdivided at the moment the mouse button is pressed, and one of the subdivided grid points is marked by .

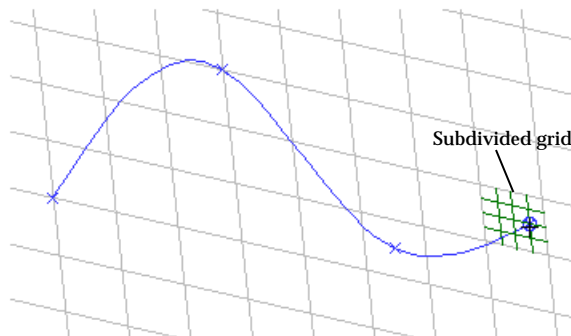
- 3) Drag the  mark to the subdivided grid point to be used as the input point.

The grid cell with subdivision moves along with the screen cursor, while moving the mouse with its button pressed.

- 4) Release the mouse button.

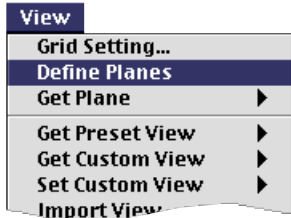
The coordinates of the point with  mark is entered as the new input point.

Grid subdivision may be set by using the "Grid Setting" dialog, or using preference dialog, which are described in "Setting Grid."



<Entering points using subdivided grid>

## User defined grid

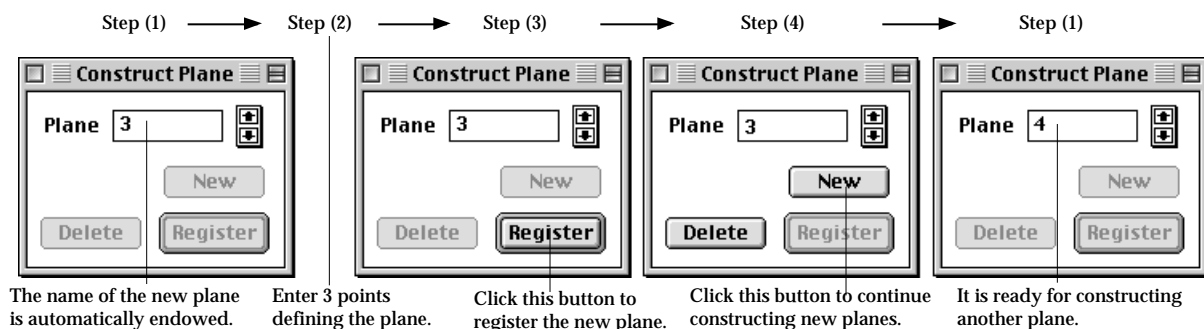


The system defined XYZ grid plane consists of three planes, i.e., XY, YZ and ZX planes which are respectively normal to Z, X and Y axes. User can define grid planes other than these ones. Their origin and their normal direction may be set arbitrarily as desired. It is very convenient, in some cases, to use these user defined grid planes for inputting curves or primitive surfaces. The user can register as many grid planes as necessary, and retrieve one of them for later use at any time.

### ■ Constructing user defined grid planes

A user defined grid plane can be constructed by the following steps:

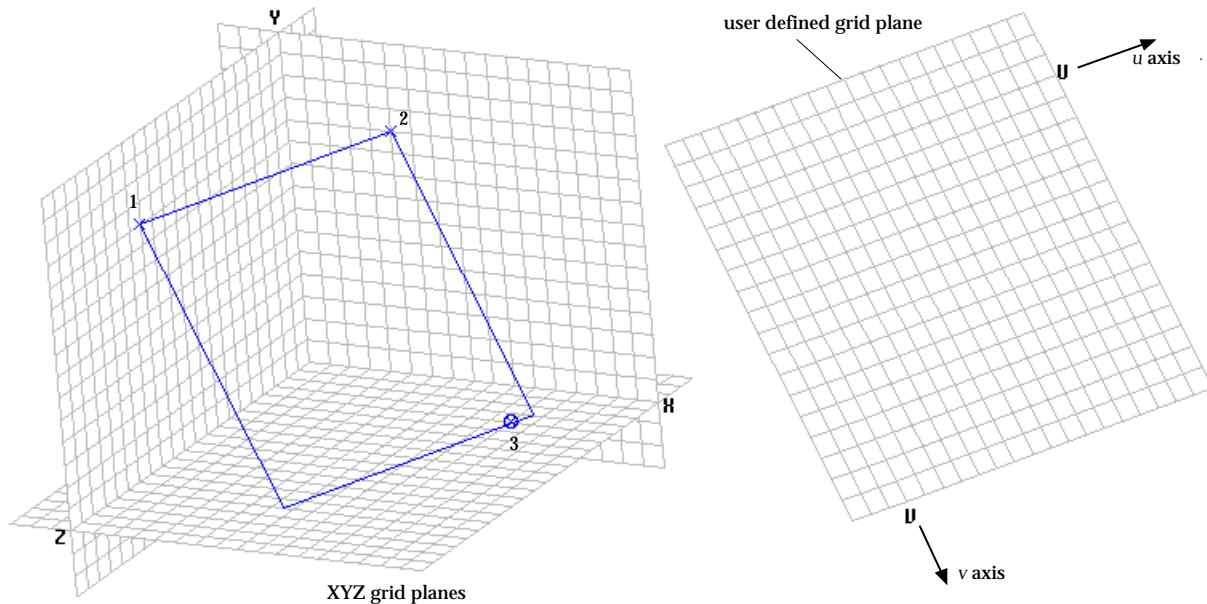
- 1) Select "Define Planes" item from **View** menu.  
 "Construct Plane" dialog appears on the screen, and "Plane" text item is filled with the name of the new plane to be constructed.  
*This name is automatically endowed by VisualFEA, and can be modified as explained in "Renaming grid planes."*
- 2) Enter 3 points which lie on the plane.  
 The first point is the origin of the plane, and the second one together with the first one forms the  $u$  axis of the plane. The plane is determined by the third point so that all the 3 points lie on the plane. Another axis of the plane, i.e.,  $v$  axis begins at the first point and is directed normal to the  $u$  axis. The 3 points can be entered by either mouse or keyboard. When all the 3 points are entered, **Register** button is enabled.
- 3) Click **Register** button of the dialog.  
 The new grid plane is created and displayed on the screen. **Register** button is disabled, and **New** and **Delete** buttons are enabled.



< "Construct Plane" dialog in each step of defining grid planes >

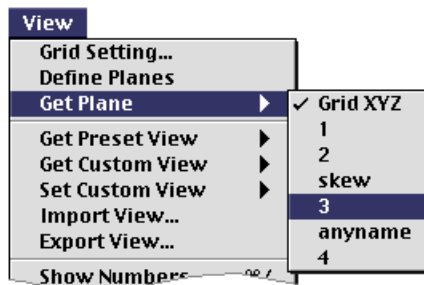
- 4) Close the "Construct Plane" dialog in order to end constructing grid planes and start entering coordinates using the grid planes. Or, click **New** button, and repeat step 2) and 3) to add more grid planes.

If you click **New** button, "Plane" text item is filled with a new plane number, and is ready for constructing a new grid plane again.



< Construction of user defined grid planes >

### ■ Retrieving user defined grid planes



The user defined grid planes as well as XYZ grid plane can be retrieved at any time for use in entering coordinates data. When a grid plane is constructed and registered, the name of the plane is added to the items of **Get Plane** submenu. The grid plane is retrieved by selecting the name from the submenu. The retrieved grid plane is displayed on the screen and effective for future entering of coordinates data. The name of the currently effective grid plane is marked in the submenu.


The user defined grid planes are saved together with the VisualFEA data file, and so can be used when the file is opened in later sessions.

### ■ Retrieving XYZ grid planes

The system defined XYZ grid plane can also be retrieved by selecting "Grid XYZ" item which is always on the top of the **Get Plane** submenu.


## ■ Deleting grid planes

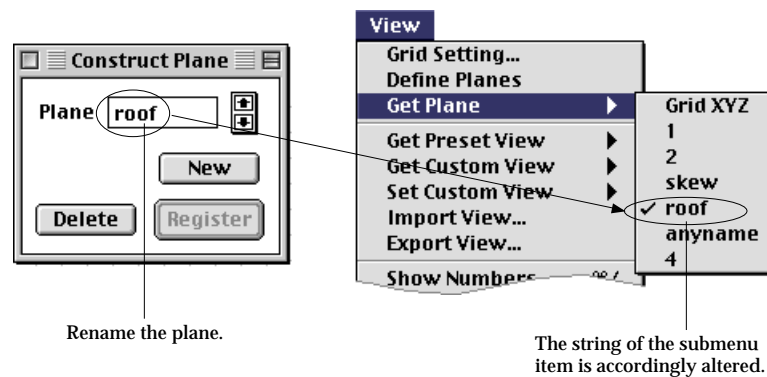
User defined grid planes can be deleted by the following steps.

- 1) Select "Define Planes" item from **View** menu.  
"Construct Plane" dialog appears on the screen, and "Plane" text item is filled with the name of the new plane to be constructed.
- 2) Scroll the effective plane using  button until the name of the desired plane appears in the "Plane" text box.  
The plane displayed on the screen is effective.
- 3) Click **Delete** button. Then, the effective plane is deleted.  
When the deleted plane disappears on the screen, and the effective plane scrolls forward to the next, or scrolls back if next one does not exist.

## ■ Renaming grid planes

The name of a user defined grid plane is automatically endowed by VisualFEA in the form of "1", "2", "3" and so on. This can be changed to other name, usually for better identification. Renaming is achieved simply by editing "Plane" text box of "Construct Plane" dialog, while the plane is being created. But, you may sometimes want to change the name later. In this case, follow the steps described below.

- 1) Select "Define Planes" item from **View** menu.  
"Construct Plane" dialog appears on the screen, and "Plane" text item is filled with the name of the new plane to be constructed.
- 2) Scroll the effective plane using  button until the name of the desired plane appears in the "Plane" text box.  
The plane displayed on the screen is effective.
- 3) Click "Plane" text box, or select the string of the name.
- 4) Change the string of the name.  
As you change the name, the string of the **Get Plane** submenu item is also altered accordingly.



< Renaming grid planes >




### 3-D cursor

VisualFEA has a special inputting aid called 3-D cursor. The 3-D cursor is useful in defining a position in 3-dimensional space in conjunction with grid or other points like nodes or control points.

The 3-D cursor consists of 3 lines and is always displayed in relation with grid planes. Each line ends on one of the grid planes. These ends are called 3-D cursor foot. The other ends of the lines meet at one point, which forms the 3-D cursor point. Coordinates in 3-dimensional space can be entered conveniently by clicking this 3-D cursor point.

#### ■ Turning 3-D cursor on and off



The 3-D cursor is initially off and not shown on the screen. The 3-D cursor button is shaped  in off state. Clicking the button  toggle 3-D cursor. The button is shaped  if the 3-D cursor is on. The 3-D cursor can be turned on only in input or modification mode. Therefore, the 3-D cursor works only when one of the input tool buttons, or modification tool buttons is pressed.

#### ■ Moving the 3-D cursor point

In order to enter coordinates using the 3-D cursor point, the point should first be moved to a desired position by the following steps.

- 1) Move the screen cursor over one of the 3-D cursor foot.
- 2) Press the *control* key .

The input mode is temporarily suppressed by pressing the *control* key.

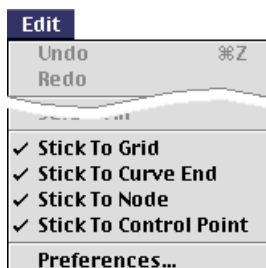
- 3) Press the mouse button.
- 4) Drag the 3-D cursor foot to the desired position by keeping the mouse button and *control* key pressed, while moving the screen cursor.

The 3-D cursor foot moves along with the movement of the screen cursor, and accordingly the 3-D cursor point moves.

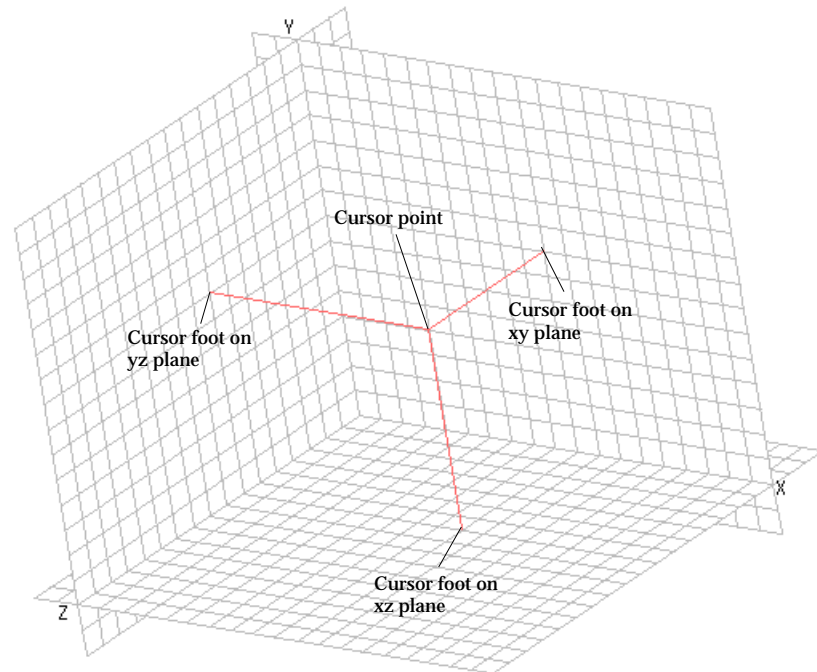
- 5) Release the mouse button.

Releasing the mouse button terminates the movement of 3-D cursor foot.

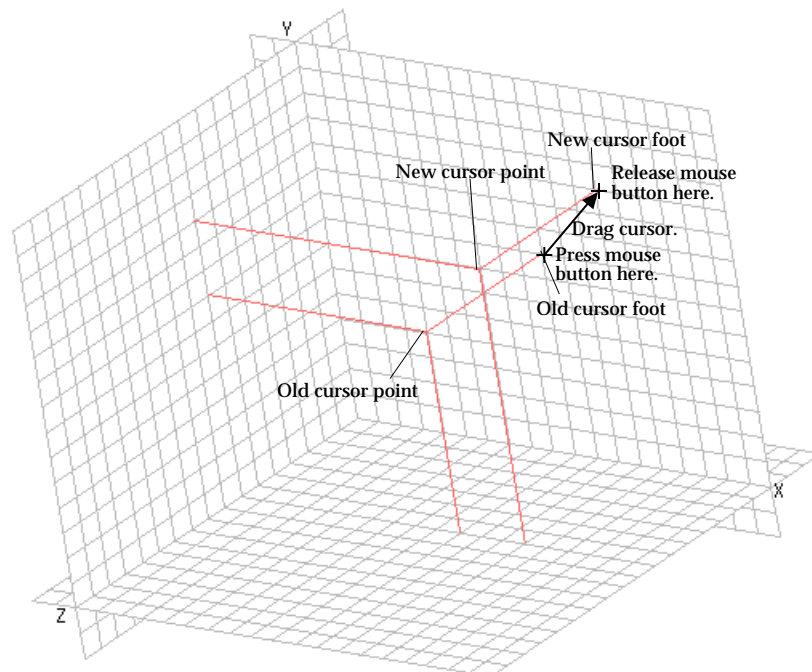
- 6) Move the screen cursor over to another 3-D cursor foot, and repeat 1)-5) until the 3-D cursor point settles at the desired position.



"Stick to Grid", "Stick to Curve End", "Stick to Node" and "Stick to Control Point" options in **Edit** menu are also applied to positioning the 3-D cursor. For example, if "Stick to Node" item is checked, the 3-D cursor foot can be laid on a node point.

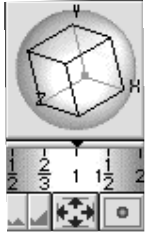


&lt; Composition of 3-D cursor &gt;



&lt; Moving 3-D cursor point &gt;

## Viewing Control



Finite element modeling objects and other related data are visualized on the screen while working with VisualFEA. You can control the graphical display of 2- or 3-dimensional data on the planar screen. It is called viewing control and consists of the following 3 actions.

- rotation: set the view direction
- zoom: set the scale of the view
- pan: translate the viewing part

Desired view can be obtained by applying these actions with proper combination. The methods of viewing control are relatively simple and intuitive as explained in the following.

### Rotating view

Rotating view can be interpreted in two different concepts:

- The user's view direction is fixed, and the 3-dimensional space containing the finite element model is rotating with respect to the view direction.
- The 3-dimensional space is fixed, and the user's view direction is changing.

In either case, the actual data are not affected, but only the display on the screen is changing. There are a few methods of rotating view in one of the two concepts.

*It is also possible to rotate the actual coordinates of the finite element models. But it is termed here as "model rotation" and has nothing to do with view rotation. Model rotation is explained in Chapter 4.*

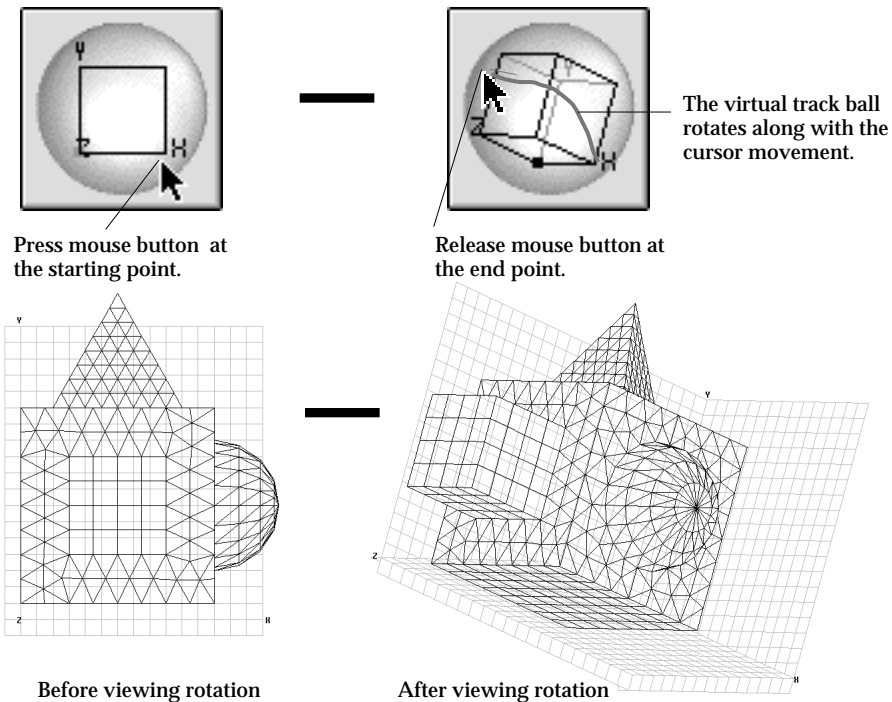
#### ■ Rotating view using virtual track ball

The virtual track ball is shown in the tool palette, and can be manipulated using mouse with similar feel of a real track ball in the following ways:

- 1) Place the screen cursor over the virtual track ball.  
Place the cursor at a point of the virtual track ball as if you were placing your finger tip on the actual track ball.
- 2) Press the mouse button.  
The current rotation angle about each coordinates axis is displayed at the bottom of the tool palette.
- 3) Move the mouse with button pressed.  
While you are moving the mouse, the virtual track ball is rotating in accordance with the cursor movement, in the same way as a real track ball rotates along with your finger tip. The rotation angles displayed at the bottom of the tool palette are constantly updated in accordance with the track ball rotation.
- 4) Release the mouse button.

Rotation of the virtual track ball ends, and the view rotation is conformed to the track ball rotation.

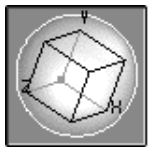
Any action in progress is not interrupted by operation of virtual track ball. For example, if you entered part of control points of a spline curve before rotating view, you can complete the curve by entering the remaining control points after the rotation.





<Rotating view using virtual track ball>

### ■ Rotating view using bounding box


The viewing rotation can also be manipulated directly on the main window using a so-called bounding box, which is a rectangular parallelepiped enclosing the modeling objects. This method is useful for more precise rotation, but interrupts other action in progress.





Locked virtual track ball

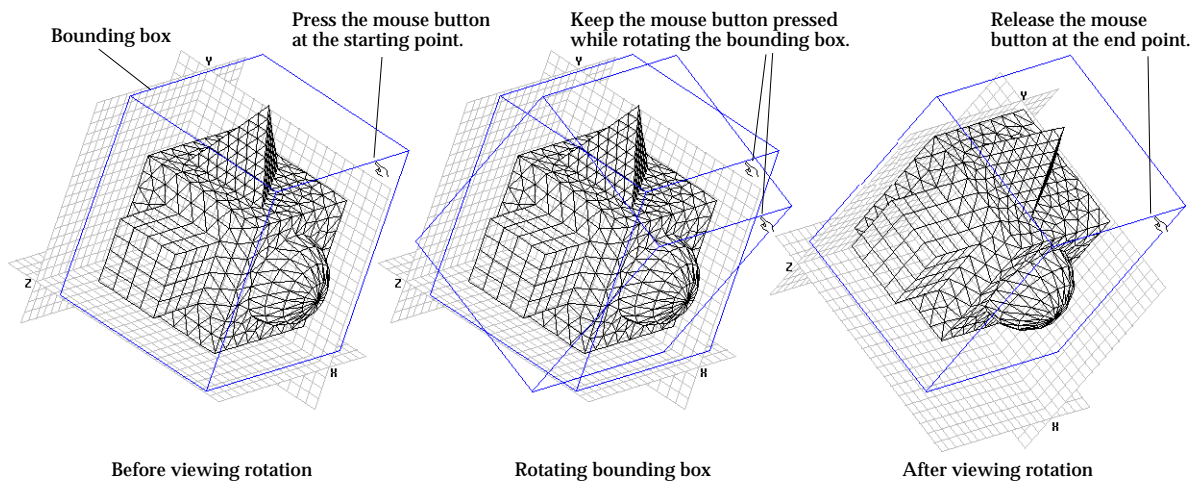
- 1) Lock the virtual track ball by clicking it with  key (Windows:  key) pressed, if it is not locked.

The locked virtual track ball is displayed as a pressed button as shown left. At the moment the virtual track ball is locked, the bounding box appears on the screen.

- 2) Move the screen cursor over an edge of the bounding box.  
The shape of the cursor changes into , when the cursor is placed within the main window.
- 3) Press the mouse button, and drag the edge of the bounding box.  
As you drag the edge, the bounding box rotates along with the edge.
- 4) Release the mouse button.

Rotation of the bounding box ends, and the view rotation is conformed to the track ball rotation. Accordingly, the string of rotation angles and the virtual track ball are updated.



The virtual track ball is still effective in locked state. In order to unlock the virtual track ball, click the track ball once again with  key (Windows :  key) pressed, or start any other tool.

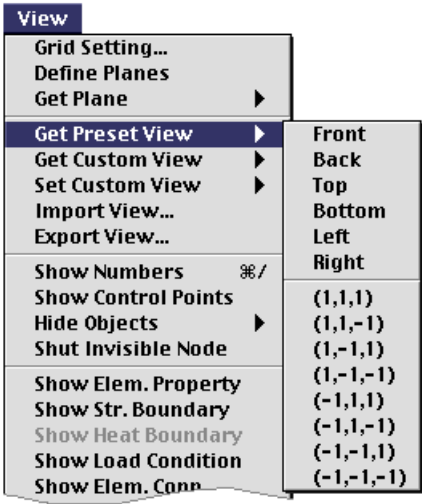


< Rotating view using bounding box >

■ Rotating view using key board

Rotation Angle	
θ <sub>x</sub>	38.9
θ <sub>y</sub>	340.1
θ <sub>z</sub>	307.8

While the view is being rotated, the angles of rotation about X, Y and Z axes are displayed at the bottom of the tool palette as shown left. Viewing rotation can be achieved by directly editing the text of these rotation angles. Click one of the text boxes, and edit the text. After all the angles are set to the desired values, press  key (Windows :  key). Then, the rotation angles are entered for new view. Accordingly, the virtual track ball as well as the view of the main window is rotated.



■ Getting the preset viewing rotations

A number of typical views, such as front view, top view, etc., are preset for quick retrieval. The desired viewing rotation is obtained simply by selecting one of the preset view items from **Get Preset View** submenu. The virtual track ball as well as the rotation angle texts are also updated in accordance with the viewing rotation.

The rotation by each of the preset view is indicated on the following table. The submenu items include 8 oblique rotations in the form of (l,m,n), in which l, m and n implies the view directions respectively in X, Y and Z axis.

## &lt; Preset view rotation items &gt;

Menu Item	View Direction	Rotated View	Menu Item	View Direction	Rotated View
Front			Back		
Top			Bottom		
Left			Right		
$(-1,-1,-1)$			$(1,1,-1)$		
$(1,-1,1)$			$(1,-1,-1)$		
$(-1,1,1)$			$(-1,1,-1)$		
$(-1,-1,1)$			$(-1,-1,-1)$		

## ■ Setting view direction

### Rotation Angle

$\theta_x$	38.9
$\theta_y$	340.1
$\theta_z$	307.8





Rotation of the view can be specified by the view direction, which is directed from user's eye to the screen. You can set the view line, and get the viewing rotation by forcing the line to be the view direction.

- 1) Lock the virtual track ball by clicking it with *control* key pressed, if it is not locked.

If you click the virtual track ball with *control* key pressed, the shape of the track ball becomes a pressed button with an arrow as shown to the left. This indicates that you are in the "view direction" mode. At the moment the virtual track ball is locked, modeling objects appears to be surrounded by a bounding box with cross hairs representing the current view point and direction.

- 2) Specify the new view point by clicking a point on one of the faces of the bounding box .

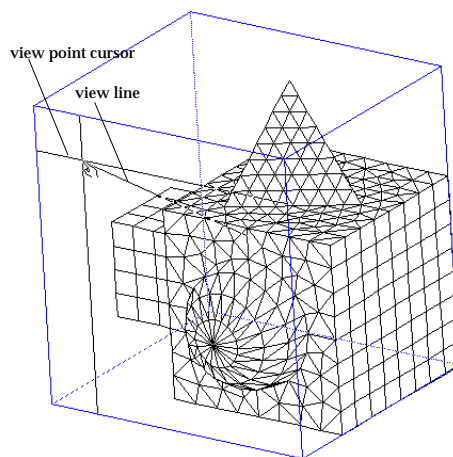
The new view point is marked by cross hairs, and the view line is represented by a straight line connecting the point and the center of the bounding box.

If you want to specify the new view point on one of the back faces, click with  key (Windows :  key ) pressed.

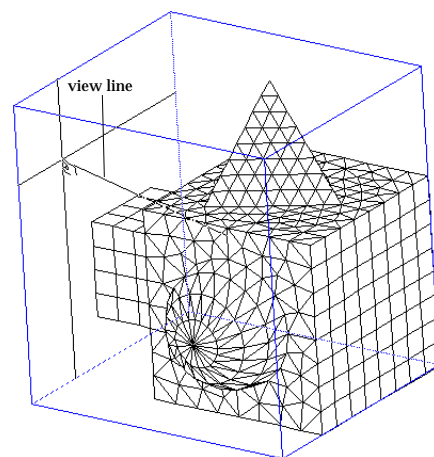
- 3) Double click the mouse button.

Double clicking rotate the view so that the specified view line is directed normal to the screen.

In order to unlock the virtual track ball, click the track ball once again with *control* key pressed, or start any other tool.



Set the view point on the front face



Set the view point on the back face


< Setting the view direction >

## Zooming in and out

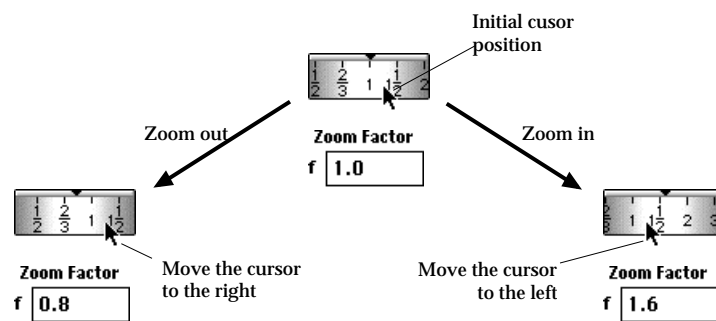


The modeling objects can be drawn in different scales depending on the required detail. Enlarging and reducing the display scale are termed here as "zooming in" and "zooming out" respectively. This zooming operation can be achieved in many different ways as explained below.

### ■ Zooming in and out using zoom dial

Zooming in or out can be achieved by turning the zoom dial which is on the tool palette. The zoom dial has scale notchmarks indicating the current display scale by  mark at the center. The zoom dial can be manipulated with the feel of a real dial as described in the following.

- 1) Place the screen cursor over the zoom dial.  
Place the cursor at a point of the zoom dial as if you were placing your finger tip on an actual dial.
- 2) Press the mouse button.  
The current zoom scale is displayed at the bottom of the tool palette.
- 3) Drag the zoom dial with the mouse button pressed.  
While you are moving the mouse, the scale notchmarks is moving to the right or to the left in accordance with the cursor movement. The contents in the main window as well as the text of the zoom scale at the bottom of the tool palette are instantly updated in accordance with the zoom dial movement.
- 4) Release the mouse button when the modeling objects on the main window are displayed with the desired scale.  
The display in the main window is rescaled with the zoom factor at the moment the mouse button is released.



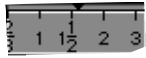
< Zooming in and out using the zoom dial >

The display scale, which is represented by the zoom factor, is initially set to 1.0 before any viewing control. This value is based on the current grid setting described in "Grid" section of this chapter. The minimum and the maximum scale on the zoom dial are respectively 1/32 and 32. It is not desirable to zoom in or out with excessive factors. So, if you need to change the display scale drastically, change the grid scale or the grid pixel value in the grid

setting dialog, instead of trying excessive zooming in or out.



Any action in progress is not interrupted while working with the zoom dial.

### ■ Zooming in and out using rubber-band rectangle




Locked zoom dial

The display scale can be altered either by expanding part of the display to fill the window, or by shrinking the whole display to a part of the window. In fact, zooming and panning are executed at the same time. You can specify the part of the display or the part of the window using rubber-band rectangle as described in the following:

- 1) Lock the zoom dial by clicking it with  key (Windows:  key) pressed, if it is not locked.

The locked zoom dial is displayed as a pressed button as shown left.

- 2) Position the screen cursor at a point in the main window.



The shape of the cursor changes into , when the cursor is placed within the main window.

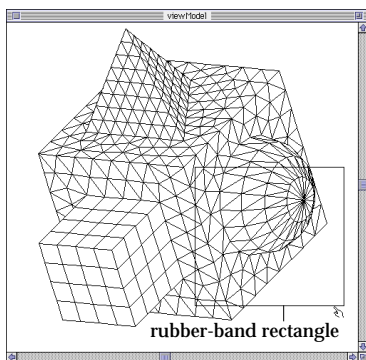
- 3) Press the mouse button, and draw a rubber-band rectangle by moving the mouse diagonally across the screen.

A rubber-band rectangle is drawn with one corner at the initial cursor position. As you move the mouse, the rectangle grows along with the cursor movement. The rubber-band rectangle keeps to be a similar figure of the main window rectangle.

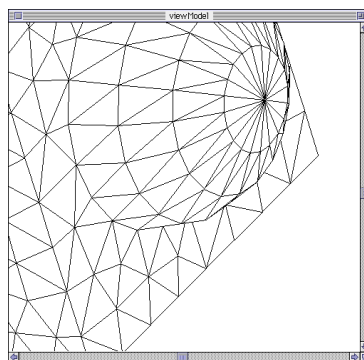
- 4) Release the mouse button.

The view is changed on the basis of the rubber-band rectangle drawn at the moment you release the mouse button. The view is zoomed out, if *control* key is pressed at the moment the mouse button is released. Otherwise, the view is zoomed in.

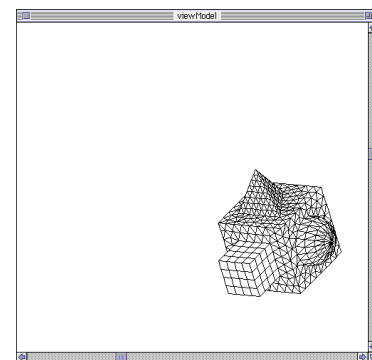
The zoom dial is still effective in locked state. and may be used for zooming. In order to unlock the zoom dial, click it once again with  key (Windows :  key) pressed, or start any other tool..



Before zooming




After zooming in  
(without pressing control key)

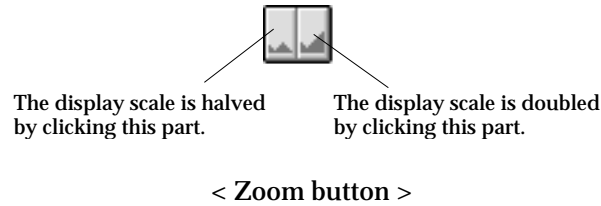


After zooming out  
(with pressing control key)


< Zooming by rubber-band rectangle >

### ■ Instant zooming by zoom button

Pressing the zoom button  enlarges the display scale by double, or reduced it by half. It is useful for instant zooming in or out. The zoom button consists of two part as show below. Clicking the left part of the button reduce the display scale by half, and clicking the right part increases the scale by double.



### ■ Fitting the display to the window

Pressing the fit-window button  makes the entire display of the modeling objects fit into the main window. If there are no modeling objects, the display of the grid planes is fitted to the window.

### ■ Entering the zoom factor by key board

**Zoom Factor**  
f

While a zooming operation is going on using any one of the above methods, the zoom factor is displayed in the text box at the bottom of the tool palette. The scale of the display on the main window can also be set by directly editing the text of the zoom factor.



## Panning

You may want to see the part which is out of the display range in the window. You can bring the part into the window by panning. Panning is an operation moving the display of the modeling objects on a window. This can be achieved by a few different methods as explained below.


### ■ Panning by scroll bar

The main window has a horizontal and a vertical scroll bar. You may pan the display in the window horizontally or vertically using the scroll bar.

### ■ Panning by *option-drag*

The shape of the cursor turns into , if you position the cursor on the main window, and press the mouse button with *option* key pressed (Windows: press the right mouse button). As you move the mouse in this state, the screen display moves along with the cursor. The movement ends and the shape of the cursor returns to , when the mouse button is released. The display of the window is updated upon the completion of this panning operation. The update may take a short or long period of time depending on the setting of "view" preference.

### ■ Centering the display

You may bring the display of the modeling objects instantly to the center of the window. Click the centering button . Then, the entire part of the display moves so that the center of the modeling objects coincides with the center of the main window. If there are no modeling objects, the display of the grid planes is centered on the main window.

## Setting and getting custom views

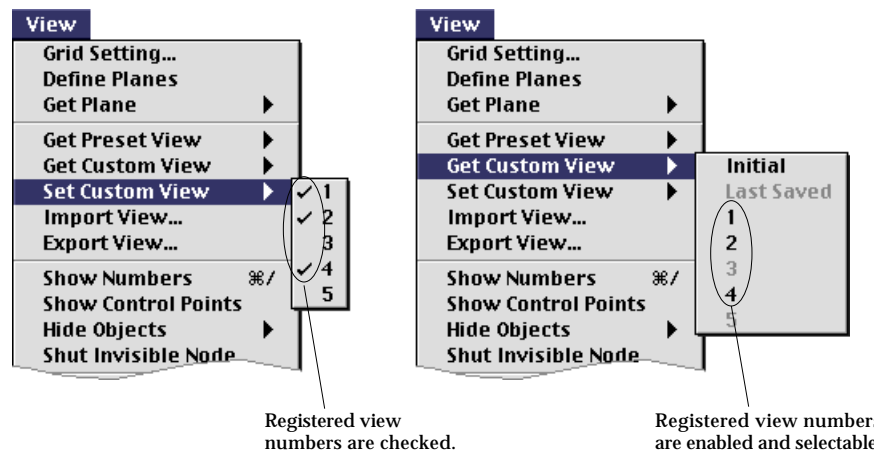
You may sometimes want to get back to the current view state after going through many view transformations. For this reason, there are functions for setting a few custom views and getting one of them later if necessary. VisualFEA allows setting up to 5 custom views.

### ■ Setting custom views

Select one of the numbers '1' through '5' from **Set Custom View** submenu. Then, the current view of the main window is registered as a custom view with that number, and the menu item of the number is marked in front. Each marked number is associated with a custom view. If you select an already marked number, the current view will replace the previous custom view with that number.

### ■ Getting custom views

Each menu item in **Get Custom View** submenu has conjunction with the item with the same number in **Set Custom View** submenu. Only the marked numbers in **Set Custom View** submenu are enabled and selectable from **Get Custom View** submenu. Selecting the number will retrieve the associated custom view, and subsequently, the display of the main window will be transformed to the custom view.



< Setting and getting custom views >

### ■ Removing custom views

In order to remove a custom view, first pull-down the **View** menu with *option* key pressed. You will find that the **Set Custom View** submenu is substituted by **Delete Custom View** submenu. Now, selecting a marked number in the submenu will remove the custom view registered by that number.

## Getting the initial view and the last saved view

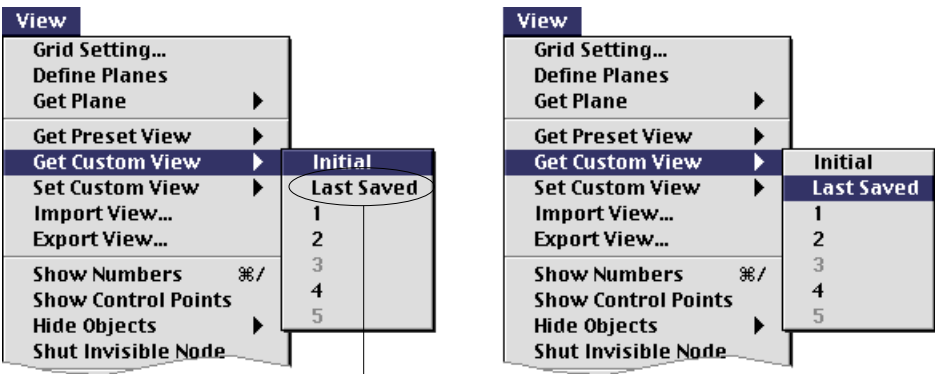
The initial view is the one set when the project begins, and the last saved view is the one set when the project file is opened or saved. It is useful to retrieve one of these view states, especially when you get lost after improper viewing transformations.

### ■ Getting the initial view of the project

In order to get the initial view, select "Initial" item from **Get Custom View** submenu. The initial view is the one defined at the beginning of the project and is not altered by any viewing transformation including rotation, zooming and panning. The initial view is affected only by grid settings. For initial view, the grid factor is 1.0, and the rotation angles about X, Y and Z axes are 0.0.

### ■ Getting the last saved view of the file

The data related to the viewing transformation and grid settings are saved together with the project file. The view data saved in the project file are retrieved when the file is opened. You may obtain the saved view at any moment while working with a project file. If you save a file, the current view is also saved, and thus becomes the last saved view. The "Last Saved" item in **Get Custom View** submenu is enabled only after the project file has been saved at least once. The last saved view is obtained by selecting the menu item.



This item is enabled only when the current project has been saved at least once.

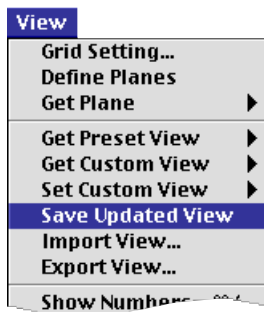
< Menu items for the initial view and the last saved view >

## Saving, importing and exporting views

The view data stored in the file of current project can be updated to the view displayed on the screen, without saving the whole content of the file. The view data can also be exported to a separate file, i.e., a view file. And, it is also possible to import view data of view files or other project files into the current project. The view data for importing and exporting include following items:

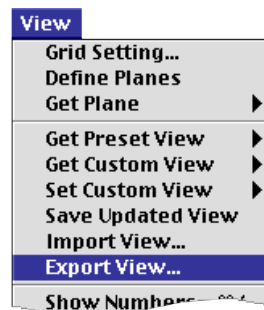
- Current viewing transformation : rotation, zoom, pan
- Custom views
- Grid settings : grid scale, grid pixel and grid origin
- User-defined grid planes

When view data is imported, the current view data are replaced by the imported data, and accordingly, the display on the window is transformed.



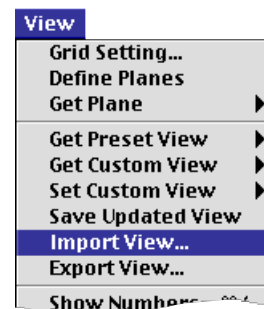
### ■ Updating view data in the working file

When a project file is opened, the view data are read, and applied in displaying the model. After one or more view transformations including zooming, panning and view rotation, the screen view is no longer the same as the view stored in the file. In order to make the view data in the file identical to the screen view, select "Save Updated View" item from **View** menu. Only the view data is updated in the project file, and other data remain intact.



### ■ Exporting views

In order to export the view data, select "Export view" item from **View** menu. A standard "Save As" dialog appears and allows you to provide a name for the view file and to choose where it will be saved. The name is initially designated as "untitled.view". Edit the text of the file name if necessary, and click "Save" button. Then, a new view file is created, and view data are stored in the file.

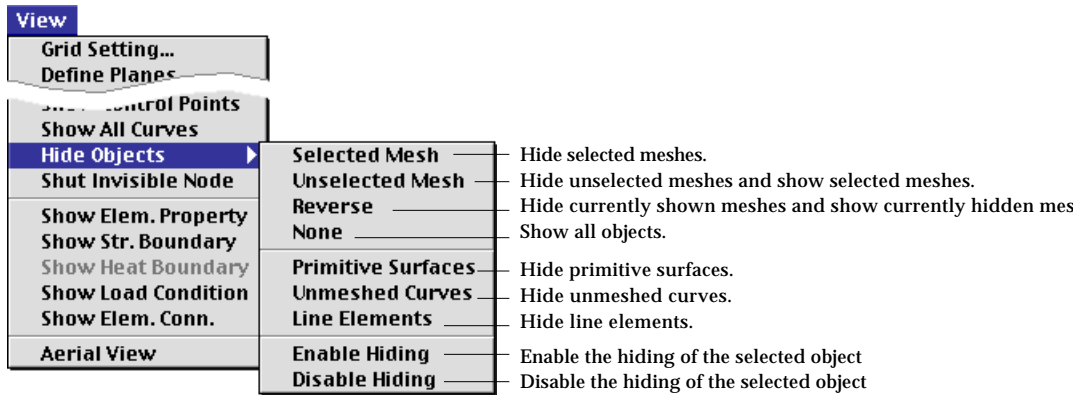


### ■ Importing views



In order to import the view data, select "Import view" item from **View** menu. A standard file dialog appears and allows you to browse through the file system and to select the file with the view data you want to import. You may select either a view file or a project file. In case of a project file, only the view data are read into the current project. The screen view is updated immediately, but the project file will not be affected until the file is saved using "Save" command in **File** menu, or the view data is saved using "Save Updated View" command of **View** menu.

### Hiding objects

It is sometime necessary to hide some parts of the model. Especially for complicated model, it is more convenient to make unwanted parts invisible and work only with remaining parts. Hiding can be applied either to selected meshes or to selected types of objects as indicated by the items of **Hide Objects** submenu shown below. Hiding is just for screen display including pre- and postprocessing, but does not affect actual modeling data.



#### ■ Hiding selected meshes

In order to hide unwanted meshes, first select the meshes, and choose "Selected Mesh" item from **Hide Objects** submenu. Consecutive hiding operations on "Selected Mesh" add the selected meshes to the list of hidden meshes. In order to hide a surface mesh, the surface mesh selection tool  should be used to hide surface meshes, and the volume mesh selection tool  to hide volume meshes. *Surface meshes surrounding volume meshes cannot be hidden individually, because volume meshes are rendered by surrounding surface meshes.*

#### ■ Hiding unselected meshes

If you choose "Unselected Mesh" item from **Hide Objects** submenu, after selecting meshes, the selected meshes remain visible, and the other unselected meshes become invisible. Consecutive hiding operations on "Unselected Mesh" add the unselected meshes to the list of hidden meshes.

#### ■ Reversing visibility

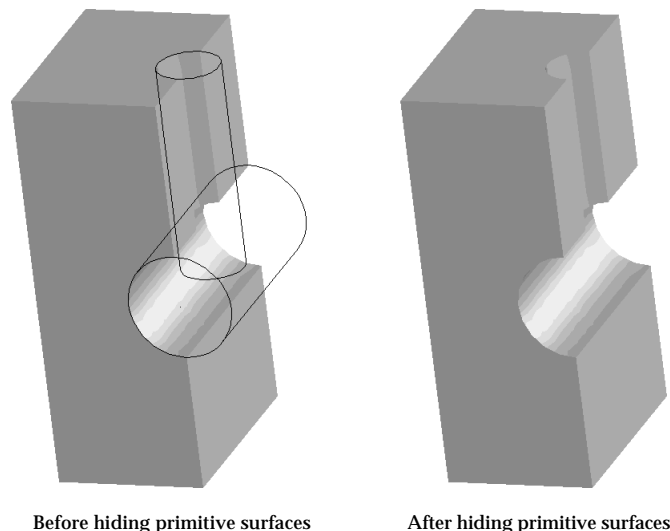
If you choose "Reverse" item from **Hide Objects** submenu, currently invisible meshes become visible, and visible meshes become invisible.

### ■ Recovering visibility of all objects

In order to release mesh hiding and make all the meshes visible, choose "None" item from **Hide Objects** submenu. However, this command does not affect the visibility of primitive surfaces, unmeshed curves, and line elements which can be hidden by the commands described below.

### ■ Hiding primitive surfaces

Primitive surfaces, such as cylinders, cones or spheres are useful as auxiliary objects in mesh generation, but are not directly involved in the analysis model. Therefore, their appearance may be cumbersome at various stages other than preprocessing. It is sometime desirable to make these primitive surfaces invisible as exemplified in the figure below. To hide primitive surfaces, choose "Primitive Surfaces" item from **Hide Objects** submenu. The item is checked while primitive surfaces are hidden. To make primitive surfaces visible, uncheck the menu item by selecting it again.



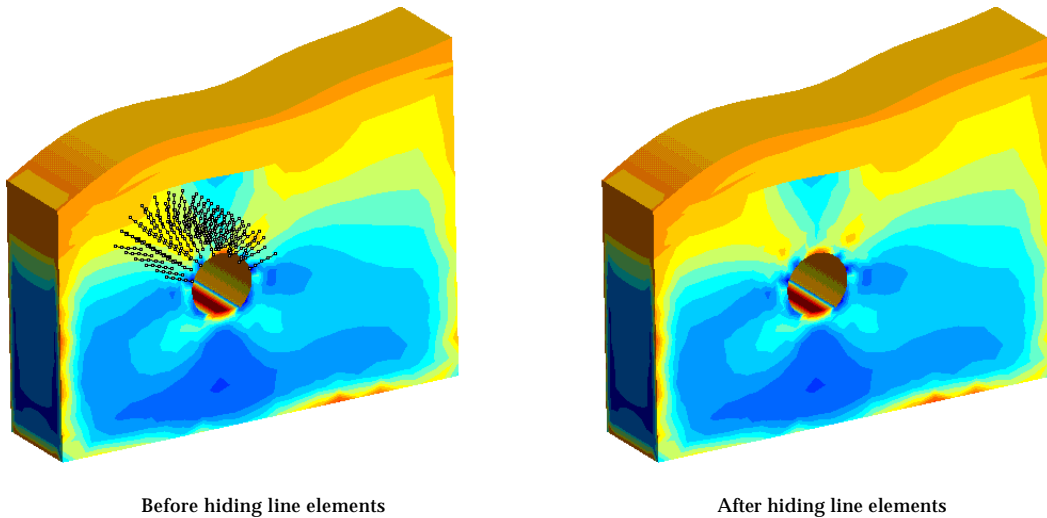
<Example of hiding primitive surfaces>

### ■ Hiding unmeshed curves

VisualFEA uses lines and curves for mesh generation, and accordingly most of them are listed as components of meshes. Such curves (or lines) are termed here as "meshed curve." Meshed curves are not explicitly rendered. However, there may still remain unmeshed curves even after completion of mesh generation. These curves may appear redundant in rendered model image or in contoured image. If it is the case, it is desirable to hide unmeshed curves. Hiding unmeshed curves can be achieved by selecting "Unmeshed Curves" item from **Hide Objects** submenu. The item is checked while unmeshed curves are hidden. To make unmeshed curves visible, uncheck the menu item by selecting it again.

### ■ Hiding line elements

You may include line elements such as truss or frame elements in continuum model. In order to reduce the complexity of model rendering, it is desirable in some cases to hide such line elements from contoured or rendered model image. Hiding line elements can be achieved by selecting "Line Elements" item from **Hide Objects** submenu. The item is checked while line elements are hidden. To make line elements visible, uncheck the menu item by selecting it again.



<Example of hiding line elements>

### ■ Disabling or Enabling the hiding of the selected objects

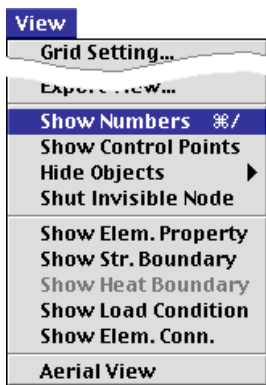
Surface meshes as components of a volume mesh become visible or invisible depending on the visibility of the volume mesh. Likewise, the visibility of the curves which belong to a surface mesh is determined by the visibility of the surface mesh. However, some curves or surfaces need be made visible regardless of the visibility of their master objects. You may keep some objects from being hidden along with their master object. Choose "Disable Hiding" submenu item after selecting the desired objects. Then, the objects are marked as hiding disabled. Use "Enable Hiding" item to release the disabled hiding status.

## Object numbers

Various objects are numbered among those of the same kind. The numbers assigned to objects can be displayed or changed. Automatic renumbering of nodes and elements are also possible.

### ■ Displaying numbers

Object numbers can be displayed one by one or in group by the following steps.



- 1) Press an object selection tool.

Click to show node number, to show element number, to show curve number, to show surface primitive number, to show surface mesh number, and to show volume mesh number.

*Object selection is described in detail in the section "Object Selection" of this chapter.*

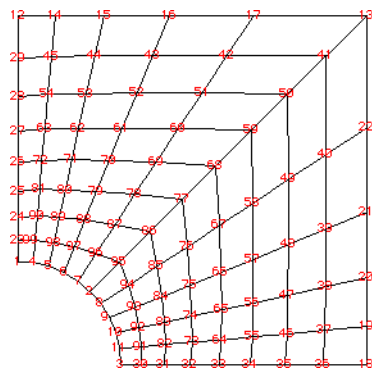
- 2) Select objects whose numbers are to be shown.

Select one object to display the number of a single object, or multiple objects to display multiple numbers at once. All the numbers are displayed when no objects are selected.

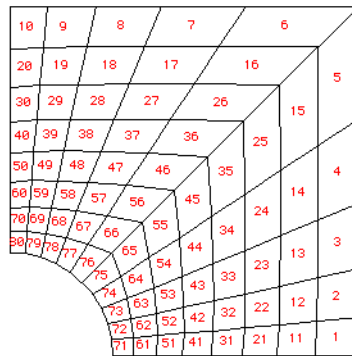
- 3) Choose "Show Numbers" item from **View** menu.

The object numbers are displayed over the selected objects. If no objects are selected, all numbers are displayed.

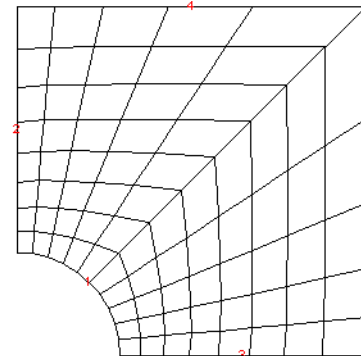
*Only element assigned with element property are numbered. Therefore, numbers will not be shown for elements without property assignment.*



Node number



Element number



Curve number

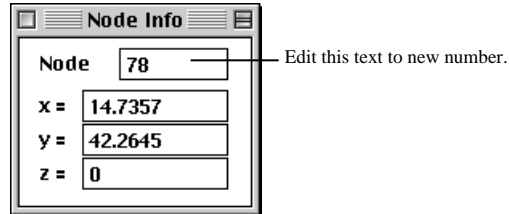
<Examples of displayed numbers>

### ■ Changing numbers

The object numbers can be altered individually by the following steps.

- 1) Press a corresponding object selection tool.
- 2) Double click the object, the number of which is to be altered.

Depending on the object selection tool in action, dialog of "Node Info", "Element Info", and so on pops up as shown below.

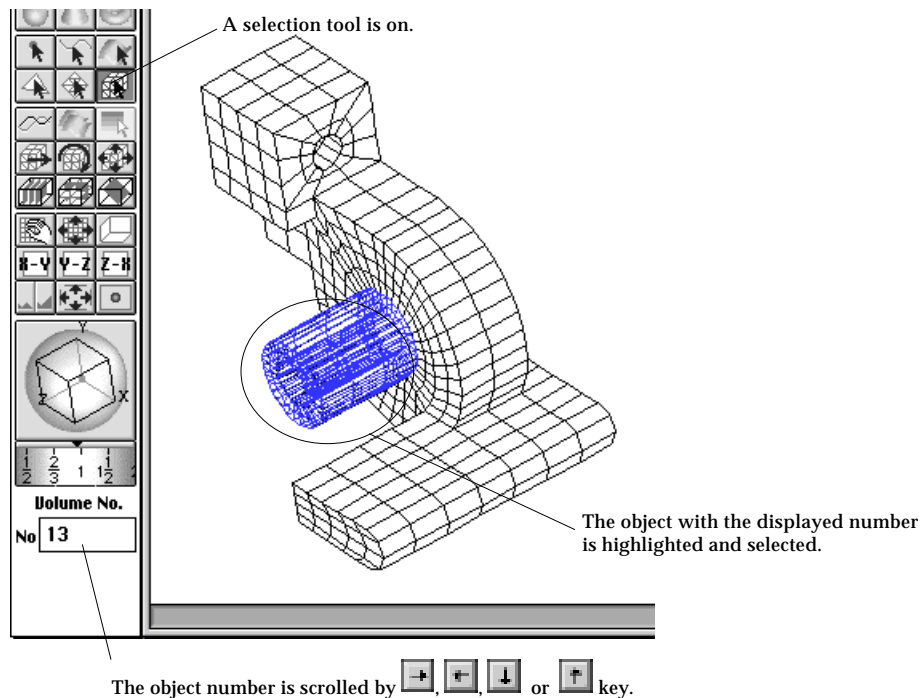


### 3) Change the number text.

Edit the editable text and press **return** key (Windows : **Enter** key). The new number should be greater than or equal to 1 and less than or equal to the total number of objects. The new number is assigned to the object, and the other objects are renumbered accordingly.

## ■ Selecting an object by its number

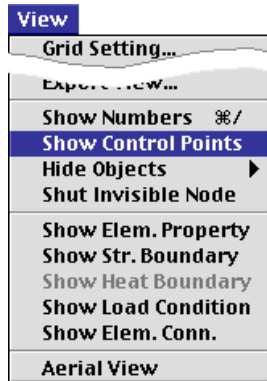
An object can be selected by its number. When a selection tool is on, scrolling the object number using **↑**, **↓**, **←** or **→** key highlight the object with currently displayed number in the editable text item at the bottom of the tool palette. The highlighted object is included in the selection list.



<Scrolling the object number>

## Other functions related to view control

There are a number of miscellaneous functions related with view control. Among them, only the functions included in **View** menu are described here.

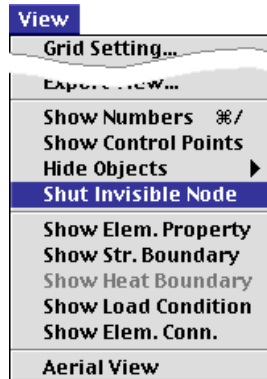


### ■ Displaying control points of curves

While inputting or reshaping a curve, its control points are displayed. But, they are hidden under other states of modeling. In order to make control points visible all the time, choose "Show Control Points" item from **View** menu. Then, the item changes into "Hide Control Points." In order to hide control points again, choose the item again.

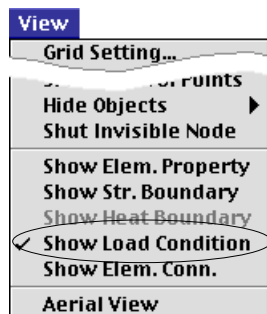
### ■ Making invisible nodes unselectable

The nodes on the backside of the model are hidden in the model image rendered by shading or wireframe mesh with hidden line removal. However, the hidden nodes on the backside of the scene are still selectable. It is sometimes convenient to make such nodes unselectable. Choose "Shut Invisible Node" item from **View** menu in order to make such nodes unselectable. While this option is on, the menu item is checked. In order to release this option, uncheck the menu item by choosing it again. While this option is on, the speed of model rendering is slow.



### ■ Making attribute assignments displayed

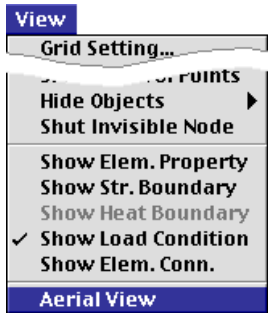
The assignments of various attributes, such as load conditions, boundary conditions and so on are displayed only when the assignment is going on and thus the corresponding dialog is on. It is sometimes necessary to make those assignments constantly displayed regardless of the modeling state. This can be achieved by selecting the corresponding menu item from **View** menu.



While this item is checked, the attribute remains displayed

### ■ Controlling the view using aerial view

While working with a complicated model, you may need to zoom in only a small part of the model to fill the screen, and want to control the view of the model as a whole. In such a case, the aerial view can be used as a convenient tool of controlling the view by the following steps:



- 1) Select "Aerial View" item from **View** menu.

Then, "Aerial View" window appears on the screen, and the overview of the whole model is displayed on the "Aerial View" window.

*The position and size of the window can be adjusted in the same way as the other window. The overall view of the model is always fit to the window regardless of the size and position of the window.*

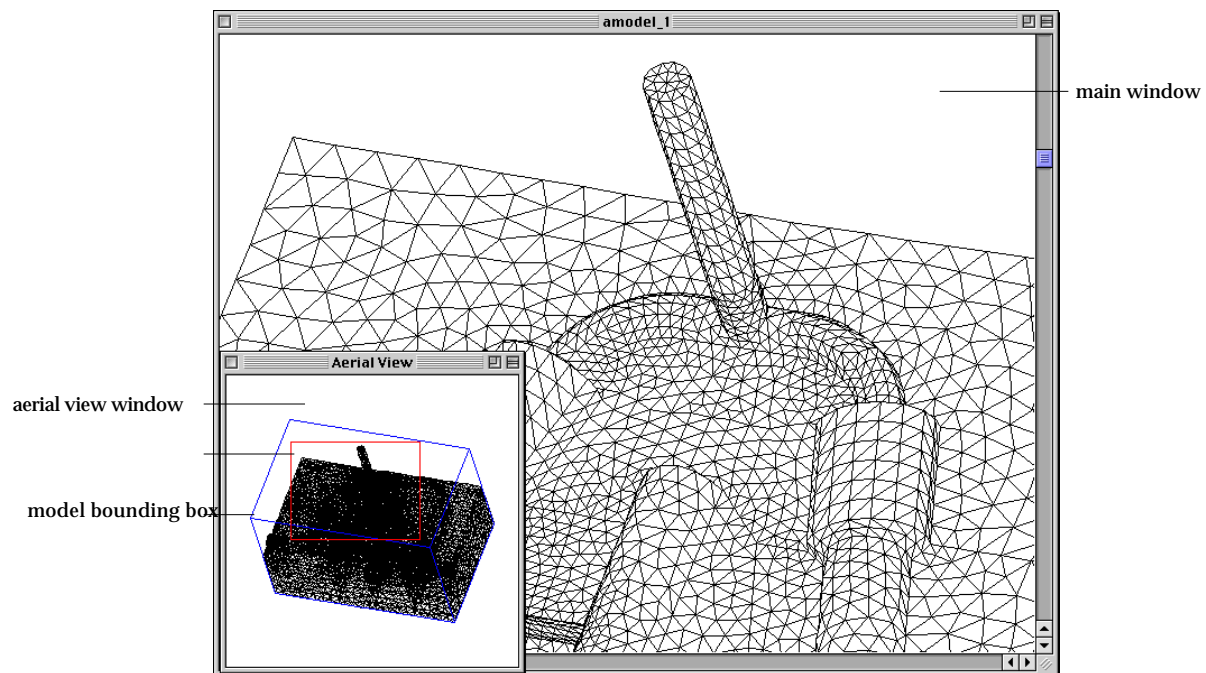
- 2) Drag the window bounding rectangle to pan the display of the model.

The window bounding rectangle represents the view range of the display on the main window. While the rectangle is being dragged, the view of the model pans continuously.

- 3) Choose "Show Numbers" item from **View** menu.

The object numbers are displayed over the selected objects. If no objects are selected, all numbers are displayed.

*Only element assigned with element property are numbered. Therefore, numbers will not be shown for elements without property assignment.*



<The aerial view window and the image display in the main window >

## Model Rendering

The finite element model can be represented graphically in various forms. You may set the rendering style and the projection mode by choosing menu items as shown below. When you change the rendering style or the projection mode, they are applied to the immediate redrawing of the model and to the future updates of the image.

Render	
<input checked="" type="checkbox"/> Wireframe	Render by wireframe mesh
<input type="checkbox"/> Hidden Removed	Render by wireframe mesh with hidden lines removed
<input type="checkbox"/> Outline	Render by outlines
<input type="checkbox"/> Shaded Image	Render by shading image
<input type="checkbox"/> Transparency	Render by transparency shading
<input type="checkbox"/> Broken Mesh	Render by broken mesh
<input type="checkbox"/> Perspective	Apply perspective projection
<input type="checkbox"/> Stereo	Apply stereo projection
<input type="checkbox"/> Depth Cued	Render by broken mesh
<input type="checkbox"/> Background Title	Display the title of the program in the screen background

### Setting rendering style

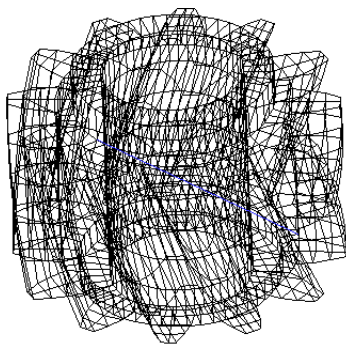
For graphical representation of model, VisualFEA provides different styles of rendering as follows.

- wireframe mesh without hidden line removal
- wireframe mesh with hidden line removal
- outline
- shading
- transparency shading
- broken mesh

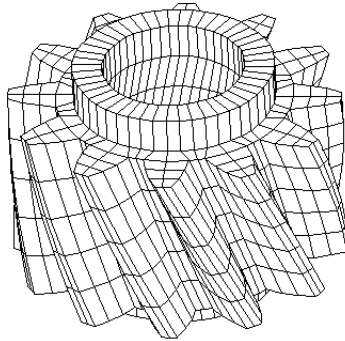
The default style is wireframe mesh. Choose corresponding item from **Render** menu to change the rendering style. The screen image of the model is immediately updated using the new style. The style remains effective for future model rendering until you change the style again.

#### ■ Rendering by wireframe with or without hidden line removal

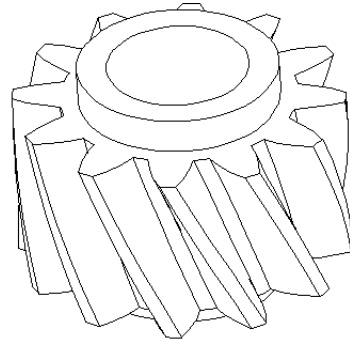
Wireframe rendering is the one represented by the mesh lines of the model. For solid models, the mesh lines inside the volume are not shown, and only lines on outer surface meshes are drawn. It is efficient and convenient in many cases to apply wireframe rendering owing to its speed. However, the shape of shell or 3-D solid may sometimes look obscured when rendered by wireframe mesh without hidden line removal. The visual clarity is much improved by removing the hidden lines. But, rendering with hidden line removal usually takes much longer time than the one without.



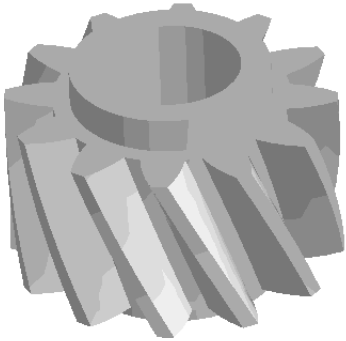
Wireframe



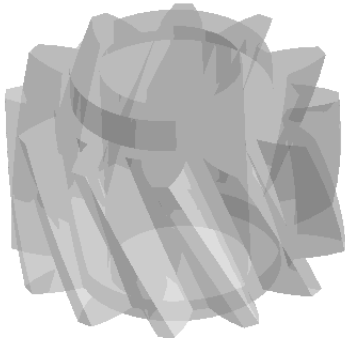
Wireframe with hidden lines removed



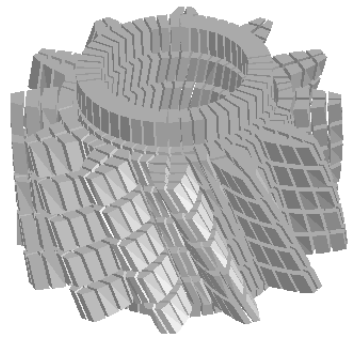
Outline



Shading



Transparency shading



Broken mesh

<Styles of model rendering >

### ■ Rendering by outline

Rendering by outline further simplifies the model image. Only the outlines necessary for proper representation of the model are extracted and used in rendering. Its rendering speed is almost equivalent to that of wireframe mesh with hidden line removal. This style of rendering is also useful for rendering of the outer surface with contours inside the volume.

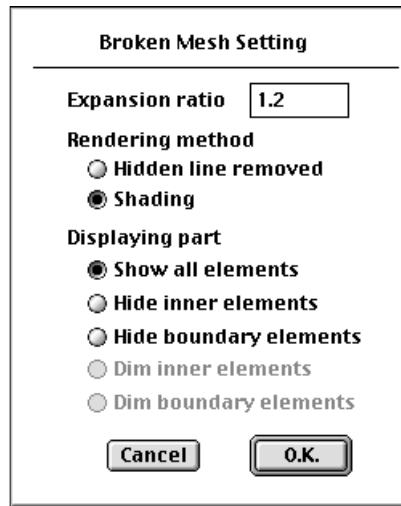
### ■ Rendering by shading or transparency shading

Rendering by shading produces more realistic view of the model. The model is represented by solid surfaces with brightness due to assumed light sources. The directions and intensity of light sources can be adjusted by using "Preference" dialog.

Transparency shading represents the model as a transparent object. This rendering style is also useful for representing the data distribution using iso-surfaces, parallel planes and so on.

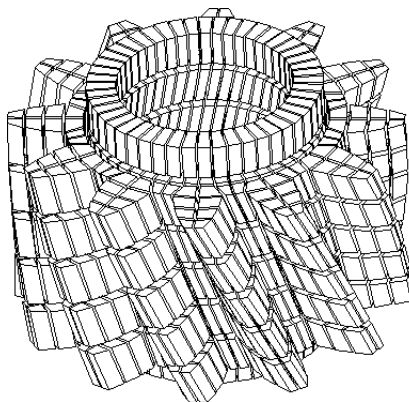
### ■ Rendering by broken mesh

Rendering by broken mesh produces image of the model with all the elements torn apart. In order to get broken mesh image, select "Broken Mesh" item from **Render** menu. Then, the following "Broken Mesh Setting" dialog pops up on the screen.

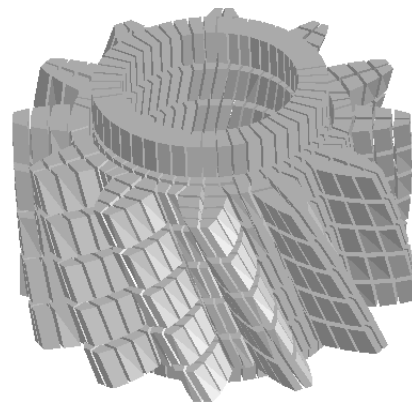


The dialog has following items:

- expansion ratio : Set the value in the editable text box. The expansion ratio determines how the broken mesh expands or shrinks. If this ratio is greater than 1, the broken mesh is obtained by inserting gaps between elements. If it is less than 1, the broken mesh is obtained by shrinking the individual elements in their original places and thus creating gaps between elements.
- rendering method : Broken meshes can be rendered either by shading or by wireframe mesh with hidden line removed. Click appropriate dialog button to select the option.



Wireframe with hidden lines removed



Shading

<Rendering styles of broken mesh>

- rendering part : This item determines which part of the model is to be shown using broken mesh.
  - "Show all elements" : All elements are included in the broken mesh.
  - "Hide inner elements" : Only the elements on the outer surface will be drawn.
  - "Hide boundary elements" : Only the inner elements will be drawn.

After setting all the above items in the dialog, and click "O.K" button. Then, the broken mesh image is drawn.

### Setting projection mode

The projection mode is the method of projecting the model to the screen, and concerned with how the model is to be rendered. The following projection modes are available in VisualFEA.

- perspective mode
- stereo mode
- depth cued mode

Choose corresponding item from **Render** menu to change the projection mode. The screen image of the model is immediately updated using the new mode. The mode remains effective for future model projection until you change the style again.

#### ■ Perspective mode

VisualFEA supports parallel and perspective modes. The default setting is parallel mode. Select "Perspective" item from **Render** menu to switch the projection mode. The grid as well as the model is represented using the current projection mode. The projection mode is applied also to the data input and object selection.

#### ■ Stereo mode (not available in Windows version)

Stereo mode is used for stereoscopic view of the model. The two separate images, one in blue and the other in red, are overlaid on the screen, and make a stereoscopic image when viewed using red and blue glasses.

#### ■ Depth cued mode

This mode is used to enhance the 3-dimensional visibility. The objects near in front side are displayed in darker color, and those far in back are displayed in lighter color.

## Inputting Coordinates of Points

A curve or a primitive surface is defined by control points. Control points are, for example, the end points of a straight line, the center and two end points of a circular arc, or the center of a sphere. Thus, a curve or a primitive surface is constructed by inputting the coordinates of its control points. VisualFEA assumes all the coordinates are in 3 dimensional Cartesian space. There are two methods of inputting the coordinates of control points, namely the one using mouse and the other using keyboard.

### Inputting coordinates using mouse

The coordinates of a control point may be entered simply by a mouse click. The position of the screen cursor at the moment of clicking is related with a point in 3-dimensional space, which is one of the following five kinds of points:

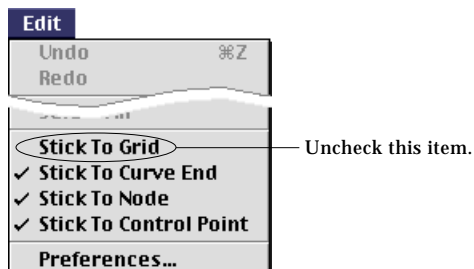
- a point constrained on grid planes
- a grid point
- a control point of an existing curve or a surface primitive
- an existing node point
- 3-dimensional cursor point

Thus, the 3-dimensional coordinates of the point are entered for a new control point.

#### ■ Entering coordinates using grid planes

The coordinates of a point can be entered using a grid plane (xy, yz or zx plane) by the following steps:

- 1) Turn on the grid plane, if it is off.
- 2) Uncheck "Stick to Grid" item of **Edit** menu, if it is checked.



- 3) Move the cursor over the grid plane image.  
The screen cursor changes into + shape.

- 4) Press the mouse button.

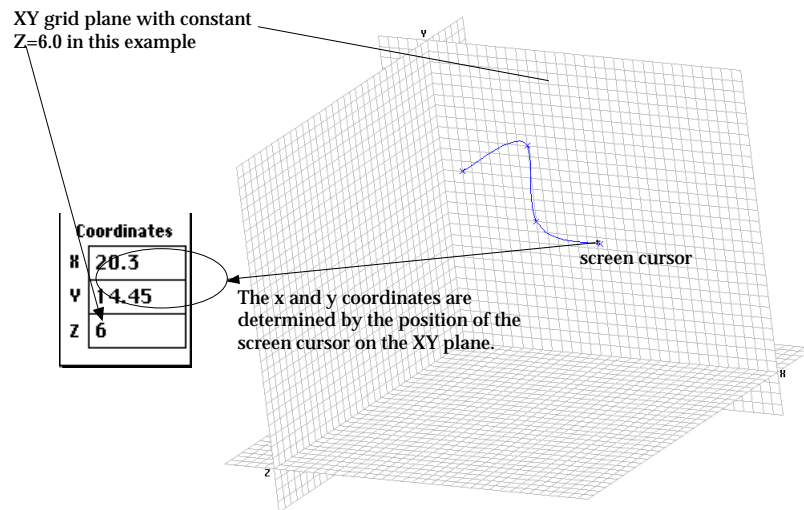
While the mouse is pressed, the coordinates at the cursor point are echoed in the text boxes at the bottom of the tool palette. If you move the mouse

keeping the mouse button pressed, the text of the coordinates constantly changes in accordance with the cursor position.

5) Release the mouse button.

The coordinates of the point are entered by the coordinates echoed in the text box at the moment the mouse button is released.

The coordinates entered as above is constrained by the grid plane. If the screen cursor is contained within the image of any grid plane,  $xy$ ,  $yz$ , or  $zx$ , the position of the cursor is mapped into 3-dimensional space by the plane. Accordingly, the cursor point is related to a point on the plane. In other words, while the cursor moves on the screen, the movement of the corresponding point in 3-dimensional space is constrained over the surface of the plane.

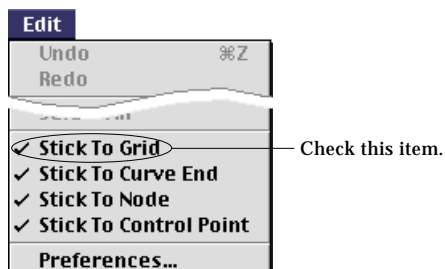


< Example of entering coordinates using the grid planes >



## ■ Entering coordinates using grid points

A grid point is a point at which two orthogonal grid lines cross on a grid plane. Three-dimensional coordinates can be entered using these grid points by the following steps.

- 1) Turn on the proper grid plane, if it is off.
- 2) Check "Stick to Grid" item of **Edit** menu, if it is unchecked.

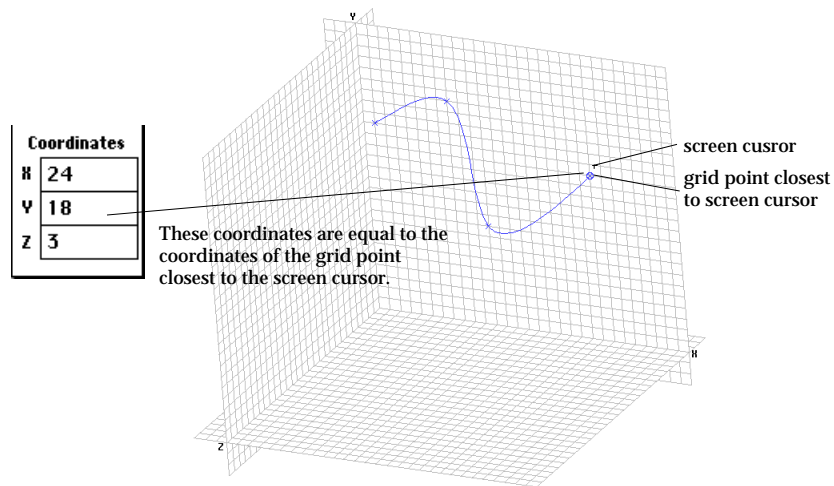


- 3) Move the cursor close to a grid point.
- 4) Press the mouse button.

At the moment the mouse button is pressed, the grid point closest to the screen cursor is marked by  and the coordinates of the grid point are echoed in the text boxes at the bottom of the tool palette. As you move the mouse keeping the button pressed,  mark moves to another grid point closest to the new cursor position.

- 5) Release the mouse button.

The coordinates of the point are entered by the coordinates of the grid point echoed in the text box at the moment the mouse button is released.




< Example of entering coordinates using grid points >

## ■ Entering coordinates using control points

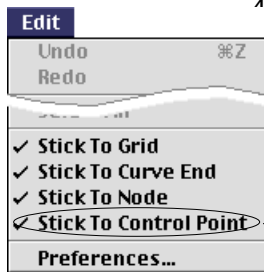
The coordinates can be entered using control points of previously created curves or surface primitives by the following steps.

- 1) Check "Stick to Control Point" item of **Edit** menu, if it is unchecked.
- 2) Move the cursor close to a control point of a curve or a surface primitive.
- 3) Press the mouse button.

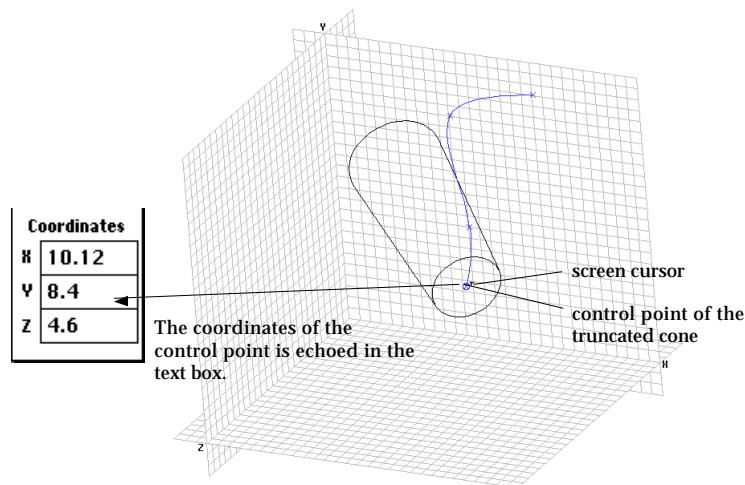
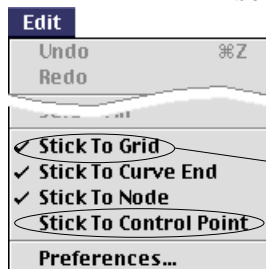
At the moment the mouse button is pressed, the control point closest to the screen cursor is marked by  and the coordinates of the control point are echoed in the text boxes at the bottom of the tool palette.

- 4) Release the mouse button.

The coordinates of the point are entered by the coordinates of the grid point echoed in the text box at the moment the mouse button is released.



The coordinates can be entered using only the end points, instead of control points, of previously created curves or surface primitives. For example, the end points of a spline curve, but not the control points in the middle are used for entering coordinates. To use this option, "Stick to Control Point" item should not be checked, while "Stick to Curve End" item is checked.

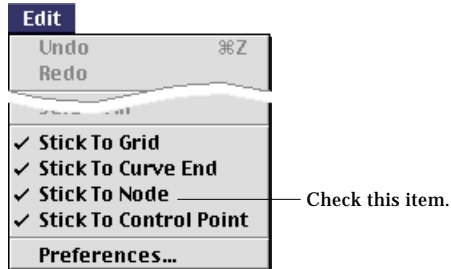


< Example of entering coordinates using a control point of a primitive surface >


### ■ Entering coordinates using nodes

The coordinates can be entered using nodes by the following steps.

- 1) Check "Stick to Node" item of **Edit** menu, if it is unchecked.

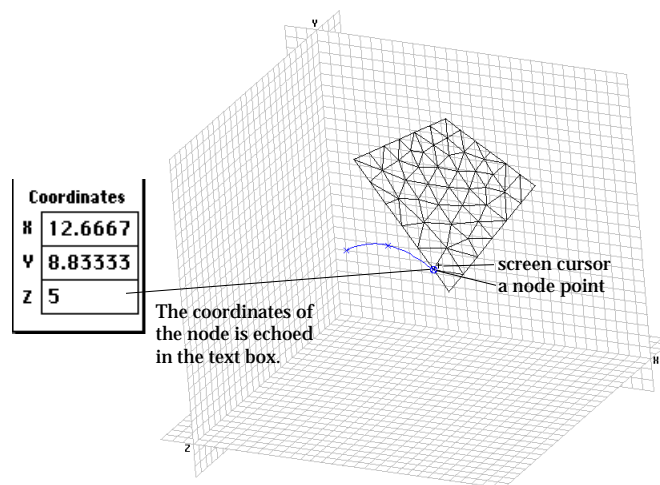


- 2) Move the cursor close to a node.
- 3) Press the mouse button.

At the moment the mouse button is pressed, the node closest to the screen cursor is marked by  and the coordinates of the node are echoed in the text boxes at the bottom of the tool palette.

- 4) Release the mouse button.

The coordinates of the point are entered by the coordinates of the node echoed in the text box at the moment the mouse button is released.






< Example of entering coordinates using a node >

## ■ Entering coordinates using 3-D cursor

VisualFEA has a special inputting aid called 3-D cursor. The 3-D cursor is useful in defining a position in 3-dimensional space in conjunction with grid or other points like nodes or control points. The coordinates can be entered using this 3-D cursor by the following steps.



- 1) Turn on the 3-D cursor.

Clicking the 3-D cursor button  toggle 3-D cursor. The button is shaped  if the 3-D cursor is on, and  if it is off. The 3-D cursor appears on the screen only in input or modification mode.

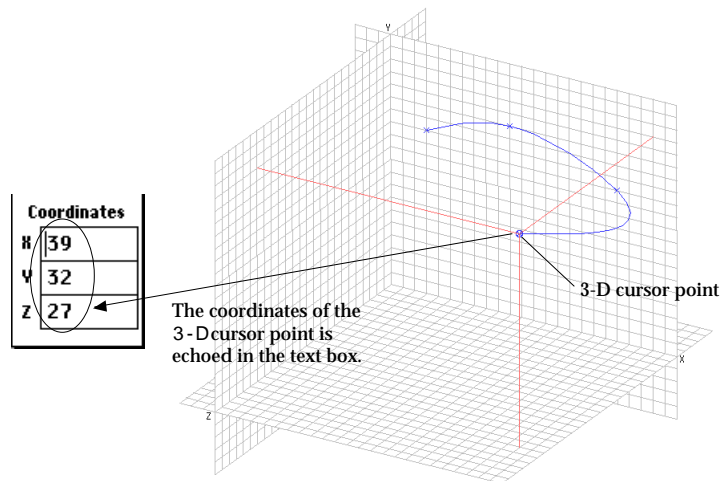
- 2) Move the 3-D cursor point to the desired position.

The method of moving the 3-D cursor is explained in "3-D Cursor" section of this chapter.

- 3) Move the screen cursor over the 3-D cursor point.

- 4) Click the mouse button.



Then, the 3-D cursor point is marked by , and the coordinates of the 3-D cursor point are echoed in the text boxes at the bottom of the tool palette.



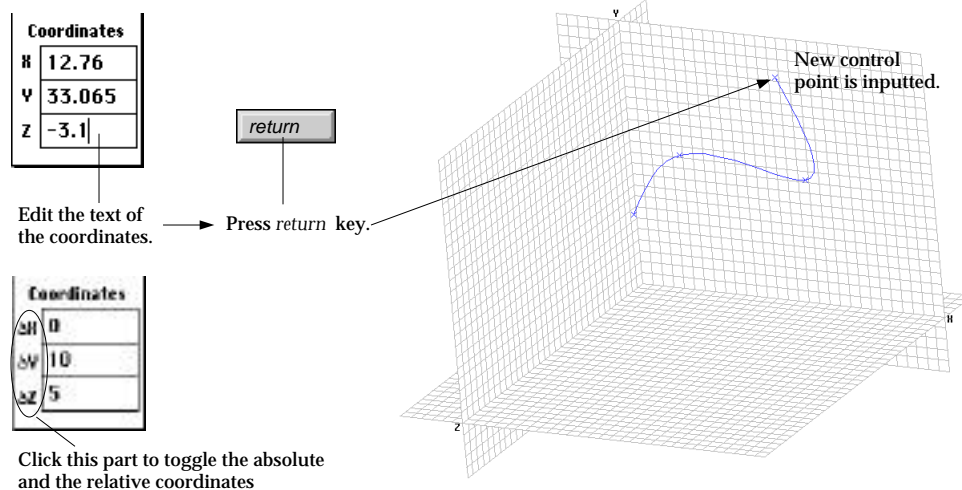
< Example of entering coordinates using 3-D cursor >

## Inputting coordinates using keyboard

While one of input tools or modification tools is in action, editable text boxes are shown at the lower part of the tool palette. The x, y and z coordinates of the control point can be entered or edited within these editable text boxes by the following steps.

- 1) Click one of the text boxes containing x, y and z coordinates.  
A caret start blinking in the text box, which implies the text is now in editable state.
- 2) Edit figures in the text box to represent the desired coordinate value.
- 3) Click other box and repeat steps 1) and 2), as needed.  
Repeat the editing for each text box for the x, y and z coordinates.
- 4) Press  key (Windows :  key).

The values in the text boxes will be entered as the coordinates of a control point.



< Example of entering coordinates using keyboard >







The keyboard input is useful when the control point does not agree with any grid point or existing control points, but accurate coordinate input is required.

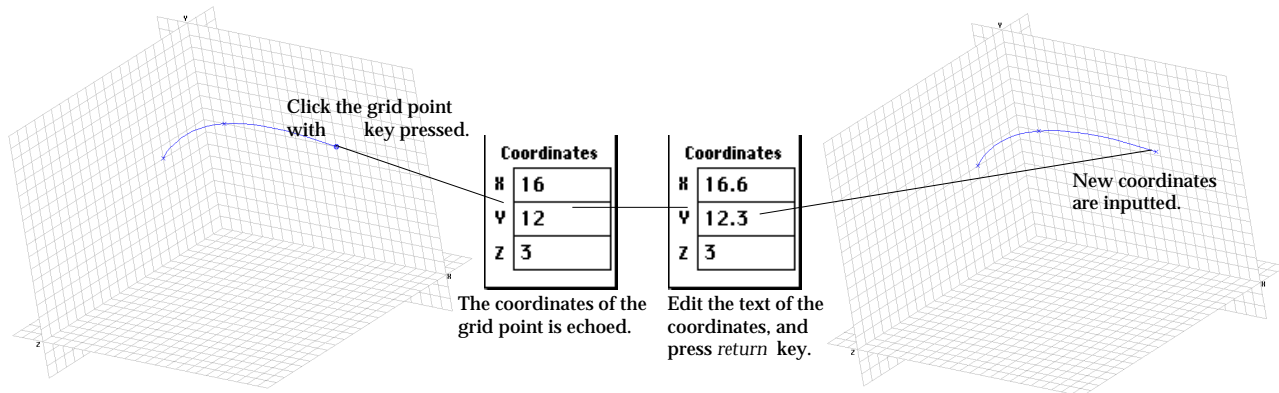
### ■ Inputting the coordinates by offset distance

The coordinates may be inputted by relative values, or offset distance. Click the part circled in the above figure. Then, the heading turns into " X", " Y" and " Z", which indicate that the inputted values represent the relative coordinates.

### ■ Keyboard input with combined use of the grid points

The coordinate input may be further facilitated by combining the grid input and the keyboard input in the following way.

- 1) Move the cursor over to the grid point close to the desired input point.
- 2) Press  key (Windows :  key), and click the grid point.  
At the moment the mouse button is pressed, the grid point is marked by , and its coordinates are echoed in the text boxes. But, the  mark disappears as soon as the mouse button is released. This indicates that the coordinates are not actually inputted.
- 3) Edit a part of the text to represent the desired coordinate value.  
Repeat the editing for each of text box for the x, y and z coordinates.
- 4) Press  key (Windows :  key).  
The values in the text boxes will be entered as the coordinates of a control point.

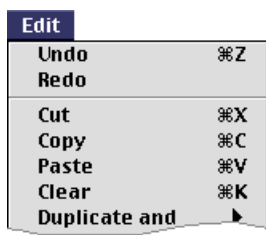



< Example of keyboard input with combined use of the grid points >

## ■ Repeating the last input

It is sometimes necessary to overlay the new point on the last input point. It is achieved simply by pressing the "space bar" of the keyboard. It works only immediately after the last point is entered. Once any other action is taken, this function of repeating the last input will not work until a new point is inputted again.

## Undoing coordinates input



In order to cancel the last input, select "Undo" item of **Edit** menu. Then, the last entered control point disappears, and the menu item "Redo" is enabled. If you select "Redo" item, the disappeared control point will be recovered. Another way of undoing is to press  key.

For coordinates input, "Undo" and "Redo" can be repeated as many times as it is effective. For example, if you are creating a spline curve and have entered 5 control points so far, you can consecutively undo these 5 points by selecting repeatedly "Undo" item. At the moment all the points are undone, there is a beep sound indicating that there are no more points to undo.

## Selection



Press one of the selection tool buttons in order to select objects including curves, primitive surfaces, nodes, elements, or meshes. Selected objects are always highlighted in blue color. Selected objects are usually applied for next processing. Accordingly, some of the processings are activated depending on the state of the selected objects. They are summarized in the following table.

< Object selection and related processings >

Selected object	Related processings
Node	get node information, show node number, drag node, assign boundary conditions, assign load(nodal)
Element	get element information, show element number, assign element properties, assign load(uniform, mid point, trapeziform, body)
Curve	get curve information, show curve number, reshape, delete, copy, project, intersect, link, separate, fillet, duplicate, assign boundary conditions, assign load(uniform, mid point, trapeziform), divide, generate mesh (auto mesh, auto mesh on primitive, 2 edges, 3 edges, 4 edges, 12 edges, extrude to curve, sweep, revolve, twist)
Surface mesh	get surface primitive information, show surface primitive number, delete, copy, intersect, generate mesh (auto mesh on primitive)
Surface primitive	get surface mesh information, show surface mesh number, delete, copy, project, duplicate, hide, move, assign boundary conditions, assign element properties, assign load (uniform, hydrostatic, body), contour on selected object, generate mesh (extrude to surface primitive, extrude to mesh, sweep, revolve, twist)
Volume mesh	get volume mesh information, show volume mesh number, delete, copy, duplicate, assign element properties, assign load(body), hide, expand, move, rotate






## Tools for object selection

In VisualFEA, curves, primitive surfaces, nodes, elements, and meshes are the kinds of objects that can be selected. There are a few different tools of selection specific to each kind of the objects.

### ■ Node selection tool



Node No.  
No 235


The node selection tool  is used to select node(s). When the tool button is pressed, a text box for inputting or displaying a node number appears at the bottom of the tool palette. The selected node number is shown in this text box only when a single node is selected. If a number is entered in this text box, the node with corresponding number is selected. The text box is initially blank as shown below, when the tool is started. It also becomes blank when more than one node is selected. The number can be increased or decreased by using  or  keys respectively. The first and the last number can be selected instantly by pressing  or  keys respectively.

Although the nodes on the backside of the model are hidden in shading or wireframe image with hidden line removal, they are still selectable. You may make these nodes unselectable by choosing "Shut Invisible Node" item from **View** menu. See also "Making invisible nodes unselectable" in the "Viewing Control" section of this chapter.



Curve No.  
No 4


### ■ Curve selection tool

The curve selection tool  is used to select curve(s). When the tool button is pressed, a text box for inputting or displaying a curve number appears at the bottom of the tool palette. The selected curve number is shown in this text box only when a single curve is selected. If a number is entered in this text box, the curve with corresponding number is selected. The text box is blank, when the tool is started, or when more than one curve is selected.



Primitive No.  
No 8


### ■ Surface primitive selection tool

The surface primitive selection tool  is used to select surface primitive(s). When the tool button is pressed, a text box for inputting or displaying a surface primitive number appears at the bottom of the tool palette. The selected surface primitive number is shown in this text box only when a single surface primitive is selected. If a number is entered in this text box, the surface primitive with corresponding number is selected. The text box is blank, when the tool is started, or when more than one surface primitive is selected.



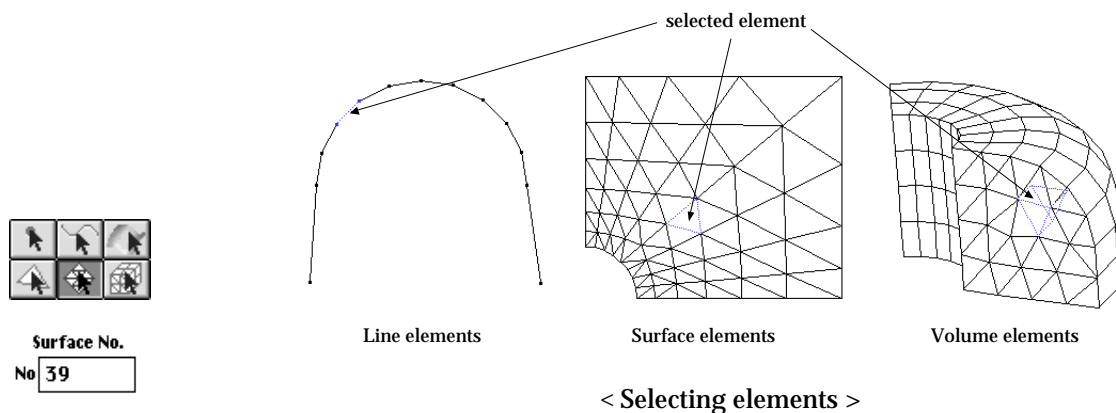
Element No.  
No 141

### ■ Element selection tool


The element selection tool  is used to select element(s). When the tool button is pressed, a text box for inputting or displaying a element number appears at the bottom of the tool palette. The selected element number is shown in this text box only when a single element is selected. If a number is entered in this text box, the element with corresponding number is selected. The text box is blank, when the tool is started, or when more than one element is selected.



When the element selection tool is on, there appears **Select** menu to the right of the menu bar. The menu has items designating what type of element is to be selected: line, surface or volume element. The current type of element for selection is indicated by the check mark in front of a menu item.




### ■ Surface mesh selection tool

The surface mesh selection tool  is used to select surface mesh(s). When the tool button is pressed, a text box for inputting or displaying a mesh number appears at the bottom of the tool palette. The selected surface mesh number is shown in this text box only when a single surface mesh is selected. If a number is entered in this text box, the surface mesh with corresponding number is selected. The text box is blank, when the tool is started, or when more than one surface mesh is selected.



### ■ Volume mesh selection tool

The volume mesh selection tool  is used to select volume mesh(s). When the tool button is pressed, a text box for inputting or displaying a mesh number appears at the bottom of the tool palette. The selected volume mesh number is shown in this text box only when a single volume mesh is selected. If a number is entered in this text box, the volume mesh with corresponding number is selected. The text box is blank, when the tool is started, or when more than one volume mesh is selected.

## Methods of selection

In order to select object(s), one of the object selection tools should be in pressed state. Only the kind of objects corresponding to the currently pressed tool can be selected. The selection is aware of 3 dimensional space. That is, the selection is sensitive to front and back relationship of the objects in screen coordinates, depending on the selection method and the kind of objects. Objects may be selected either by single or by multiple. There are a number of ways to select objects as described below.

### ■ Selecting a single object by a mouse click

An object can be selected by a mouse click in the following 2 steps.

- 1) Position the screen cursor over the object to select.

At this stage, an arrow cursor  should appear on the screen.





- 2) Click the mouse button.

Selection is determined by the location of the cursor at the moment of mouse click, and the selected object is highlighted in blue color.

The rules of determining the selection are different depending on the kind of objects, as described in the table "Rules of object selection" of next page.



*The tolerance range in the table implies the range of searching the selected object, and is initially set as 5 pixels, but can be altered by preference setting. Refer to "Tolerance settings" of "Setting Preferences" section of this chapter.*

### ■ Selecting objects using keyboard


If you insert a number in the text box at the bottom of the tool palette, the object with the number is selected. Instead, you may scroll the number using , ,  or  key. Selection changes in conjunction with the scrolling number.

*See also "Selecting an object by its number" in "Viewing Control" section of this chapter.*

### ■ Selecting an object in the rear side by () key click

In the case of selecting an element or a mesh by mouse click. as described above, the one in the front side is always selected. However, you may sometimes need to select the one in the rear. This can be achieved by clicking the object while pressing  key (Windows :  key).

### ■ Adding selected objects using shift click

Any previous selection is automatically cleared by the new selection. That is, if you select some objects, the list of the previously selected objects is cleared, then the newly selected objects are registered in the list. If you want to add more objects to the list of the selected objects without clearing previous selection, select the objects by the one of the selection method while keeping  key pressed.

## &lt; Rules of object selection &gt;

Kind of objects	Rules of selection
Node	<ol style="list-style-type: none"> <li>1. The node is closest to the cursor.</li> <li>2. The node is within the tolerance range from the cursor.</li> </ol>
Element (line)	<ol style="list-style-type: none"> <li>1. The shortest distance from the line element to the cursor is the smallest.</li> <li>2. The shortest distance is within the tolerance range.</li> </ol>
Element (surface) Element (volume)	<ol style="list-style-type: none"> <li>1. The screen image of the element contains the cursor point.</li> <li>2. The element is most in front among the elements containing the cursor.</li> </ol>
Curve	<ol style="list-style-type: none"> <li>1. The shortest distance from the curve to the cursor is the smallest.</li> <li>2. The shortest distance is within the tolerance range.</li> </ol>
Primitive surface	<ol style="list-style-type: none"> <li>1. The shortest distance from the outlines of the primitive to the cursor is the smallest.</li> <li>2. The shortest distance is within the tolerance range.</li> </ol>
Surface mesh Volume mesh	<ol style="list-style-type: none"> <li>1. The screen image of the mesh contains the cursor point.</li> <li>2. The mesh is most in front among the meshes containing the cursor.</li> </ol>

### ■ Selecting multiple objects by rubber banding

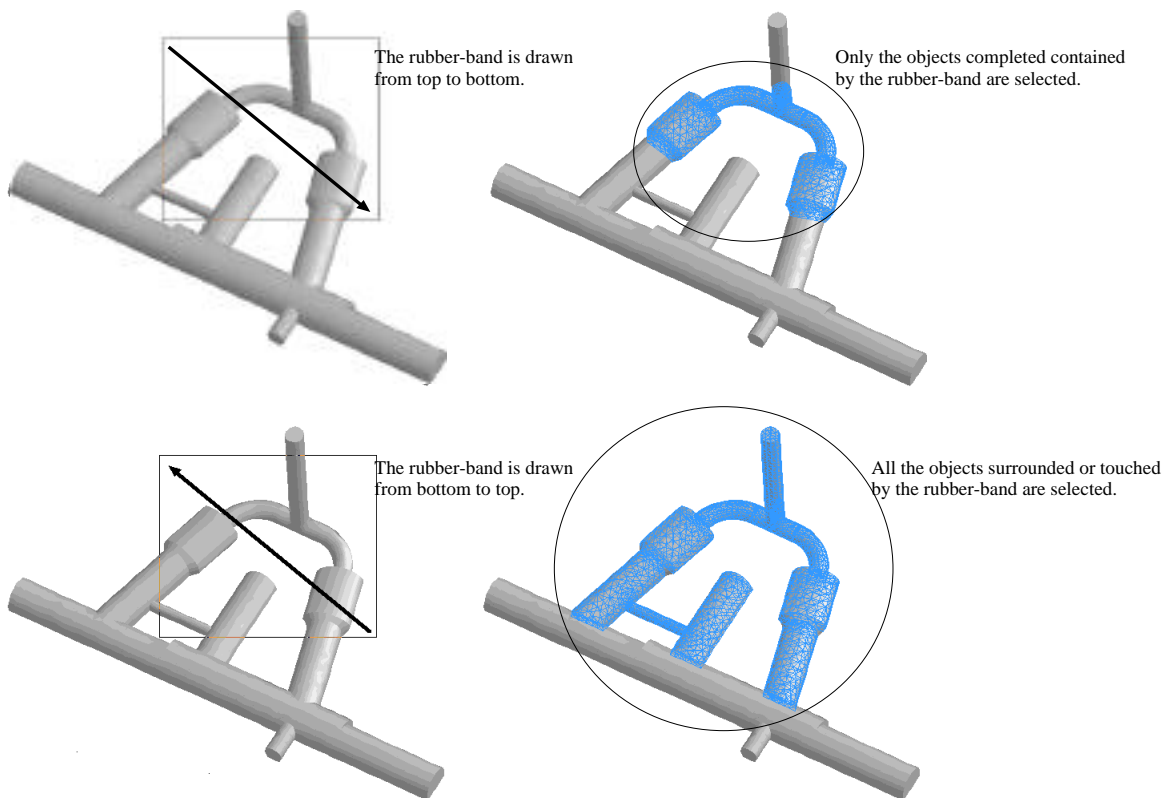
You can select more than one object at once by rubber-banding, which implies drawing a rectangle as follows:

- 1) Position the screen cursor at one point.  
Imagine 4 corners of a rectangle on the screen. Place the cursor on one of the corners.
- 2) Press the mouse button and drag the cursor diagonally.  
You are now moving the cursor from one corner to the opposite corner. A rectangle is drawn. The rectangle constantly changes along the movement of the cursor. This is called rubber-band rectangle.
- 3) Release the mouse button.

Then the rubber-band rectangle disappears, and the objects within or across the rubber-band rectangle are selected.

The range of selection is determined by how the rubber-band rectangle is drawn as described below.

- Rubber-band rectangle is drawn from top to bottom direction: Objects completely contained by the rubber-band rectangle are selected.
- Rubber-band rectangle is drawn from bottom to top direction: Objects completely contained or touched by the rubber-band rectangle are selected.



< Selecting objects by rubber-banding >

### ■ Selecting all

Choose "Select All" item from **Edit** menu. All objects of the kind corresponding to the current selection tool will be selected.

### ■ Unselecting objects

Click the window so that no selection is made. Then, the previous selection is cleared. If you want to unselect only part of the previous selection, click the objects to unselect, while keeping *shift* key pressed.

