

An Integrated  
Finite Element Analysis Software  
for MS Windows and Mac OS

# **VisualFEA**

User's Manual

© 2001 Intuition Software  
All Rights Reserved




# Contents

<b>Chapter 1</b>	<b>Overview</b>	<b>1-1</b>
	<b>Finite Element Analysis and VisualFEA</b>	<b>1-2</b>
	<b>Procedure of finite element analysis using VisualFEA</b>	<b>1-2</b>
	<i>Stages of finite element analysis / Finite element analysis using VisualFEA / Mixed use of VisualFEA and external solvers</i>	
	<b>Creating a finite element model using VisualFEA</b>	<b>1-4</b>
	<i>Node and element data / Other data assignment</i>	
	<b>Visualizing analysis results using VisualFEA</b>	<b>1-5</b>
	<i>Visualization of scalar data / Visualization of vector data / Visualization of truss and frame data</i>	
	<b>Framework of VisualFEA</b>	<b>1-7</b>
	<b>Basic User Interface</b>	<b>1-7</b>
	<i>Project and file / View control / 3-D geometric data input system / Model selection and control / Model rendering</i>	
	<b>Preprocessing</b>	<b>1-8</b>
	<i>Curve and surface primitive modeling / Mesh generation / Data Assignment / Others</i>	
	<b>Processing</b>	<b>1-11</b>
	<i>Structural analysis / Heat conduction / Seepage</i>	
	<b>Postprocessing</b>	<b>1-12</b>
	<i>Data visualization / Visualization aids / Image handling / Printing / Others</i>	
	<b>Educational Aids</b>	<b>1-14</b>
	<i>Element stiffness computation / Assembly and solution process / Shape function and interpolation / Eigen mode / Stress recovery and smoothing / Adaptive analysis process / Structural behavior</i>	
<b>Chapter 2</b>	<b>Basic User Interface</b>	<b>2-1</b>
	<b>Basic Elements of User Interface</b>	<b>2-2</b>
	<b>Mac OS and Windows versions</b>	<b>2-3</b>
	<b>Menu and tool button</b>	<b>2-4</b>
	<i>Menu / Tool button</i>	
	<b>Window and dialog</b>	<b>2-6</b>
	<i>Window / Dialog</i>	
	<b>Project and File</b>	<b>2-9</b>
	<b>Working with a project</b>	<b>2-9</b>
	<i>Launching VisualFEA / Starting a new project / Opening a project file / Importing an external file / Closing the project file / Saving the project file / Saving the file with a new name / Creating a text file with</i>	

the list of modeling and analysis data / Setting the project's analysis subject / Updating the file status / Setting up page for printing / Printing / Quitting VisualFEA	
<b>Setting preferences</b>	<b>2-15</b>
Grid settings / Tolerance settings / Shading settings / Light source settings / View settings / Solver settings	
<b>Grid and 3-D Cursor</b>	<b>2-21</b>
<b>Grid</b>	<b>2-22</b>
Setting grid / Turning grid planes on and off / Moving grid planes / Resizing grid planes / Subdividing grid	
<b>User Defined Grid</b>	<b>2-26</b>
Constructing user defined grid planes / Retrieving user defined grid planes / Retrieving XYZ grid planes / Deleting grid planes / Renaming grid planes	
<b>3-D Cursor</b>	<b>2-29</b>
Turning 3-D cursor on and off / Moving the 3-D cursor point	
<b>Viewing Control</b>	<b>2-31</b>
<b>Rotating view</b>	<b>2-31</b>
Rotating view using virtual track ball / Rotating view using bounding box / Rotating view using key board / Getting the preset viewing rotations / Setting view direction	
<b>Zooming in and out</b>	<b>2-36</b>
Zooming in and out using zoom dial / Zooming in and out using rubber-band rectangle / Instant zooming by zoom button / Fitting the display to the window / Entering the zoom factor by key board	
<b>Panning</b>	<b>2-39</b>
Panning by scroll bar / Panning by option-drag / Centering the display	
<b>Setting and getting custom views</b>	<b>2-40</b>
Setting custom views / Getting custom views / Removing custom views	
<b>Getting the initial view and the last saved view</b>	<b>2-41</b>
Getting the initial view of the project / Getting the last saved view of the file	
<b>Saving ,importing and exporting views</b>	<b>2-42</b>
Updating view data in the working file / Exporting views / Importing views	
<b>Hiding objects</b>	<b>2-43</b>
Hiding selected meshes / Hiding unselected meshes / Reversing visibility / Reversing visibility of all objects / Hiding primitive surfaces / Hiding unselected curves / Hiding line elements / Disabling or Enabling the hiding of the selected objects	
<b>Object numbers</b>	<b>2-46</b>
Displaying numbers / Changing numbers / selecting an object by its number	
<b>Other functions related with view control</b>	<b>2-48</b>
Displaying control points of curves / Making invisible nodes unselectable / Making attribute assignment displayed / Controlling the view using aerial view	



<b>Model Rendering</b>	<b>2-50</b>
<b>Setting rendering style</b>	<b>2-50</b>
<i>Rendering by wireframe with or without hidden line removal / Rendering by outline / Rendering by shading and transparency shading / Rendering by broken mesh</i>	
<b>Setting projection mode</b>	<b>2-53</b>
<i>Perspective mode / Stereo mode / Depth cued mode</i>	
<b>Inputting Coordinates of Points</b>	<b>2-54</b>
<b>Inputting coordinates using mouse</b>	<b>2-54</b>
<i>Entering coordinates using grid planes / Entering coordinates using grid points / Entering coordinates using control points / Entering coordinates using nodes / Entering coordinates using 3-D cursor</i>	
<b>Inputting coordinates using keyboard</b>	<b>2-60</b>
<i>Inputting the coordinates by offset distance / Keyboard input with combined use of the grid points / Repeating the last input</i>	
<b>Undoing coordinates input</b>	<b>2-61</b>
<b>Selection</b>	<b>2-62</b>
<b>Tools for object selection</b>	<b>2-63</b>
<i>Node selection tool / Curve selection tool / Surface primitive selection tool / Element selection tool / Surface mesh selection tool / Volume mesh selection tool</i>	
<b>Method of selection</b>	<b>2-65</b>
<i>Selecting a single object by a mouse click / Selecting objects using keyboard / Selecting an object in the rear side by command(Windows:Alt) click / Adding selected objects using shift click / Selecting multiple objects by rubber banding / Selecting all / Unselecting objects</i>	
<b>Chapter 3    Curves and Surface Primitives</b>	<b>3-1</b>
<b>Creating Curves and Surface Primitives</b>	<b>3-2</b>
<b>Creating straight lines</b>	<b>3-2</b>
<b>Creating circles or circular arcs</b>	<b>3-3</b>
<i>Arc / Clockwise Arc / Counter-CW Arc / Three Point Arc / Center &amp; Angle Arc / Clockwise Circle / Counter-CW Circle / Three Point Circle / Center &amp; Radius Circle</i>	
<b>Creating ellipses or elliptical arcs</b>	<b>3-7</b>
<i>Quarter / Half / Full</i>	
<b>Creating parametric curves</b>	<b>3-9</b>
<i>Cubic Spline / B-Spline / Bezier / Polynomial / Polyline</i>	
<b>Creating rectangles</b>	<b>3-11</b>
<b>Creating single points</b>	<b>3-11</b>
<b>Creating parametric surfaces</b>	<b>3-12</b>
<i>Flat Plane / B-spline Surface / Bezier Surface / Lagrangian Surface</i>	
<b>Creating spheres</b>	<b>3-15</b>

<i>Sphere by Radius / Sphere by 2 Points</i>	
<b>Creating cylinders, cones or truncated cones</b>	<b>3-17</b>
<b>Creating tori</b>	<b>3-18</b>
<b>Handling Curves and Surface Primitives</b>	<b>3-19</b>
<b>General editing commands</b>	<b>3-19</b>
<i>Deleting curves and surface primitives / Copying curves and surface primitives / Cutting curves and surface primitives / Pasting curves and surface primitives</i>	
<b>Reshaping and moving</b>	<b>3-19</b>
<i>Activating modification mode / Moving control points by mouse / Moving control points by keyboard input / Moving an entire curve or surface primitive</i>	
<b>Duplicating curves and surface primitives</b>	<b>3-22</b>
<i>Duplicating curves and surface primitives, and moving / Duplicating curves and surface primitives, and revolving / Mirroring curves and surface primitives / Modifying a curve or a primitive with its duplicate / Modifying attributes of a primitive surface</i>	
<b>Processing curves and surface primitives</b>	<b>3-27</b>
<i>Linking curves / Separating curves / Filleting two straight lines / Splitting curves by their intersection points / Obtaining intersection curves between surface primitives / Projecting curves / Building boundary curves of a surface primitive</i>	
<b>Curve Division</b>	<b>3-32</b>
<b>Dividing curves</b>	<b>3-33</b>
<i>Dividing curves individually / Dividing curves as a whole / Setting the number of divisions / Setting the weight of division density / Removing divisions from divided curves / Dividing curves using F keys</i>	
<b>Changing  Menu Items</b>	<b>3-37</b>
<i>Modifying the menu items for number of divisions / Changing the menu items for weight of division density</i>	
<b>Chapter 4    Mesh Generation</b>	<b>4-1</b>
<b>General procedure of mesh generation</b>	<b>4-1</b>
<b>Surface Mesh Generation</b>	<b>4-3</b>
<b>Automatic triangulation</b>	<b>4-4</b>
<i>Generating mesh on a plane by automatic triangulation / Setting a compatible region for automatic triangulation / Handling the incompatibility due to mismatching curve ends / Generating mesh on a surface primitive by automatic triangulation</i>	
<b>Surface mesh generation by mapping</b>	<b>4-12</b>
<i>Generating mesh using 2 edges / Setting 2 edges compatible for mesh generation / Shifting alignment of node pairing on 2 edges / Generating mesh using 3 edges / Generating mesh using 4 edges / Setting 4 edges compatible for mesh generation</i>	
<b>Surface mesh generation by sweeping operations</b>	<b>4-21</b>

Generating mesh by extrusion / Generating mesh by extrusion up to bounding curves / Generating mesh by translation / Generating mesh by revolution / Generating mesh by twisting	
<b>Duplicating surface meshes</b>	<b>4-36</b>
<b>Projecting surface meshes</b>	<b>4-37</b>
<b>Volume Mesh Generation</b>	<b>4-38</b>
<b>Volume mesh generation by automatic tetrahedronization</b>	<b>4-39</b>
Generating mesh by automatic tetrahedronization / Forming boundary meshes for automatic tetrahedronization	
<b>Volume mesh generation by mapping</b>	<b>4-42</b>
Generating mesh using box edges / Setting box edges compatible for mesh generation / Generating mesh using prism edges / Setting prism edges compatible for mesh generation / Generating mesh using tetrahedron edges / Setting tetra edges compatible for mesh generation	
<b>Volume mesh generation by sweeping operations</b>	<b>4-49</b>
Generating mesh by extrusion / Generating mesh by extrusion up to bounding surface primitives / Generating mesh by extrusion up to surface meshes / Generating volume mesh by translation / Generating volume mesh by revolution / Generating volume mesh by twisting	
<b>Duplicating volume meshes</b>	<b>4-62</b>
<b>Frame Element Generation</b>	<b>4-63</b>
<b>Creating frame elements using straight lines or curves</b>	<b>4-64</b>
Creating a frame element by inputting a straight line / Generating frame elements by dividing curves	
<b>Creating frame elements using mesh generation functions</b>	<b>4-66</b>
<b>Duplicating frame elements</b>	<b>4-67</b>
<b>Mesh Editing</b>	<b>4-68</b>
<b>General mesh editing commands</b>	<b>4-68</b>
Undoing mesh generation / Deleting meshes / Copying meshes / Cutting meshes / Pasting meshes / Merging meshes	
<b>Modifying nodal coordinates</b>	<b>4-70</b>
Dragging nodes using mouse / Modifying nodal coordinates by keyboard input / Changing node number / Absorbing nodes	
<b>Transforming meshes</b>	<b>4-73</b>
Moving meshes / Rotating meshes / Resizing meshes	
<b>Other mesh treatments</b>	<b>4-77</b>
Remeshing / Making cracktip elements	
<b>Surface normal direction</b>	<b>4-79</b>
Displaying surface normal direction / Displaying surface tangent directions / Reversing surface normal direction	

<b>Chapter 5</b>	<b>Data Assignment</b>	<b>5-1</b>
	<b>Overview of Data Assignment</b>	<b>5-2</b>
	<b>Basic composition of data</b>	<b>5-2</b>
	<i>Data for structural analysis / Data for analysis of heat conduction / Data for seepage analysis</i>	
	<b>General procedure of data assignment</b>	<b>5-3</b>
	<b>Functions common to all types of data assignment</b>	<b>5-4</b>
	<i>Functions handling data sets / Entering values of data items / Modifying values of data items / Assigning data sets / Checking data assignment / Clearing data assignment / Condensing data sets / Ending data assignment</i>	
	<b>Structural Element Properties</b>	<b>5-8</b>
	<b>Defining element properties</b>	<b>5-8</b>
	<i>Analysis class of element / Constitutive model / Isotropy of the properties / Data items of element properties</i>	
	<b>Defining element properties of truss and frame elements</b>	<b>5-11</b>
	<i>Defining cross section of a truss or a frame member</i>	
	<b>Assigning element properties</b>	<b>5-15</b>
	<i>Selecting objects to assign element properties / Overriding previous assignment of element properties / Representation of element property assignment</i>	
	<b>Mixing different structural types in one analysis</b>	<b>5-16</b>
	<i>Changing the analysis class of element / Applicable analysis classes of element</i>	
	<b>Using interface elements</b>	<b>5-18</b>
	<i>Characteristics of interface elements / Creating interface elements / Deleting interface elements</i>	
	<b>Using slip bars</b>	<b>5-20</b>
	<i>Characteristics of slip bar elements / Creating slip bar elements / Deleting slip bar elements</i>	
	<b>Using embedded bars</b>	<b>5-22</b>
	<i>Characteristics of embedded bar elements / Creating embedded bar elements / Deleting embedded bar elements</i>	
	<b>Color coding of property sets</b>	<b>5-25</b>
	<i>Turning on the property color mode / Changing or setting the property color</i>	
	<b>Heat Conduction and Seepage Properties</b>	<b>5-26</b>
	<b>Heat conduction properties</b>	<b>5-26</b>
	<i>Defining heat conduction properties / Assigning heat conduction properties</i>	
	<b>Seepage properties</b>	<b>5-27</b>
	<i>Defining seepage properties / Defining hydraulic conductivity functions / Assigning seepage properties</i>	
	<b>Structural Boundary Conditions</b>	<b>5-30</b>
	<b>Defining structural boundary conditions</b>	<b>5-30</b>
	<i>Nodal degrees of freedom / Data items of structural boundary condition / Entering data items of</i>	

<i>boundary conditions / Defining boundary conditions in local coordinates</i>	
<b>Assigning structural boundary conditions</b>	<b>5-35</b>
<i>Selecting objects to assign structural boundary conditions / Replacing or adding previous assignment / Representation of boundary condition assignment</i>	
<b>Heat Conduction and Seepage Boundary conditions</b>	<b>5-38</b>
<b>Defining heat conduction boundary conditions</b>	<b>5-38</b>
<i>Types of heat conduction boundary condition / Editable text items</i>	
<b>Assigning heat conduction boundary conditions</b>	<b>5-39</b>
<i>Selecting objects to assign heat conduction boundary conditions / Replacing previous assignment / Representation of heat conduction boundary condition assignment</i>	
<b>Seepage boundary conditions</b>	<b>5-41</b>
<i>Types of seepage boundary condition / Assigning open head boundary condition / Assigning confined head boundary condition / Assigning flux / Assigning point source / Defining the initial water table</i>	
<b>Load Conditions</b>	<b>5-45</b>
<b>Defining load condition sets</b>	<b>5-45</b>
<i>Data items of a load condition / Load types / Load direction / Editable text items</i>	
<b>Defining load condition sets for dynamic analysis</b>	<b>5-54</b>
<i>Setting the time dependency / Defining static load for dynamic analysis / Defining harmonic load for dynamic analysis / Defining transient load for dynamic analysis</i>	
<b>Assigning load condition sets</b>	<b>5-57</b>
<i>Selecting objects to assign load conditions / Assigning a load condition set to multiple objects / Assigning multiple load condition sets to a single object / Representation of load assignment</i>	
<b>Other functions related with assigning load conditions</b>	<b>5-58</b>
<i>Reversing the force direction / Exchanging the reference end of the curve / Suppressing load selection / Changing the placement of force symbol / Limiting the size of force symbol / Resetting the size of force symbol</i>	
<b>Load Combination</b>	<b>5-61</b>
<i>Defining load combinations / Handling load condition sets in “Load combination” dialog / Combining analysis results of multiple load conditions</i>	
<b>Dynamic Motions</b>	<b>5-63</b>
<b>Defining and assigning dynamic motion sets</b>	<b>5-63</b>
<i>Setting the time dependency of dynamic motion / Defining harmonic motions / Defining transient motions / Defining initial motions / Assign dynamic motions</i>	
<b>Defining and assigning nodal dynamic properties</b>	<b>5-66</b>
<i>Nodal dashpot / Nodal mass</i>	
<b>Frame Member Joint Conditions</b>	<b>5-68</b>
<b>Defining frame member joint conditions</b>	<b>5-68</b>

Assigning frame member joint conditions	5-68
<b>Sequentially Staged Modeling</b>	<b>5-69</b>
<b>Concept of sequentially staged modeling</b>	<b>5-69</b>
<i>The base model and the stage models / Data sharing and inheriting / Solution process of stage models /</i>	
<i>Procedure of sequentially staged modeling</i>	
<b>Creating stage models</b>	<b>5-74</b>
<i>Creating a new stage / Deleting a stage / Moving to the desired stage</i>	
<b>Building the geometry of stage models</b>	<b>5-74</b>
<i>Including the selected objects to the current stage / Excluding the selected objects from the current stage /</i>	
<i>Handling objects created after creation or some stages / "Model Display" options</i>	
<b>Property assignment in sequentially staged modeling</b>	<b>5-75</b>
<i>Assigning properties to the base model / Assigning properties to stage models</i>	
<b>Load assignment in sequentially staged modeling</b>	<b>5-77</b>
<i>Assigning loads to the base model / Assigning loads to stage models</i>	
<b>Control in processing of a sequentially staged model</b>	<b>5-79</b>
<i>Assumed stress field of a stage model / Prescribing the staged stress relaxation rate / Clearing displacements</i>	
<b>Chapter 6     Finite Element Processing</b>	<b>6-1</b>
<b>Getting Finite Element Solution</b>	<b>6-2</b>
<b>Solver of finite element analysis</b>	<b>6-2</b>
<b>Processing of structural analysis</b>	<b>6-3</b>
<i>Setting Analysis Options for linear static analysis / Setting Analysis Options for adaptive analysis /</i>	
<i>Setting Analysis Options for dynamic analysis / Setting Analysis Options for nonlinear analysis / Setting</i>	
<i>analysis options for sequentially staged modeling</i>	
<b>Processing of heat conduction and seepage analysis</b>	<b>6-14</b>
<i>Setting analysis options for heat conduction analysis / Setting analysis options for seepage analysis</i>	
<b>Processing stages</b>	<b>6-16</b>
<i>Progress of processing / Interrupting the processing / Abnormal termination of the processing</i>	
<b>Interactive real time processing</b>	<b>6-20</b>
<b>Other Functions Related with the Processing</b>	<b>6-22</b>
<b>Optimizing node numbering</b>	<b>6-22</b>
<b>Optimizing element numbering</b>	<b>6-23</b>
<b>Setting output items</b>	<b>6-23</b>
<b>Specifying integration scheme</b>	<b>6-24</b>
<b>Displaying analysis information</b>	<b>6-25</b>

<b>Chapter 7 Postprocessing of Continuum Analysis</b>	<b>7-1</b>
<b>Visualizing Scalar Data by Contours</b>	<b>7-3</b>
<b>Setting contouring options</b>	<b>7-4</b>
<i>Selecting the data item / Designating the contouring object / Setting the number of contour bands /</i> <i>Selecting the contouring method / Selecting the style of boundary surface rendering / Turning the</i> <i>shading effect on or off / Displaying the boundary of cut or cross planes / Displaying the cube</i> <i>surrounding the entire model / Limiting the range of the contour scale by actually displayed values</i>	
<b>Setting the contour scale</b>	<b>7-10</b>
<i>Editing the scale values / Setting the format of the contour scale / Aligning a contour to the specified</i> <i>value / Truncating the scale values at the specified decimal point / Getting symmetrically arranged scale</i> <i>values / Spacing the scale values with weight / Possible combination of contour scale options / Saving</i> <i>contour scale values / Reading contour scale values</i>	
<b>Setting a cut plane for contouring</b>	<b>7-16</b>
<i>Activating the cut plane setting mode / Setting a cut plane / Contouring on a cut plane / Splitting</i> <i>objects using the cut plane</i>	
<b>Setting parallel planes for contouring</b>	<b>7-20</b>
<i>Activating the parallel plane setting mode / Setting parallel planes interactively / Setting parallel planes</i> <i>by custom input / Contouring on parallel planes</i>	
<b>Setting cross planes for contouring</b>	<b>7-25</b>
<i>Activating the cross plane setting mode / Setting cross planes / Contouring on cross planes</i>	
<b>Other functions related with contouring</b>	<b>7-28</b>
<i>Contour marking over a contour image / Sampling contour value by specifying the coordinates / Turning</i> <i>on and off the contour scale bar</i>	
<b>Visualizing Scalar Data by Iso-surface and others</b>	<b>7-32</b>
<b>Visualizing scalar data using iso-surfaces</b>	<b>7-32</b>
<i>Setting the iso-surface display options / Selecting the data item to be represented by iso-surfaces /</i> <i>Selecting the data item to be represented by contours / Setting the number of iso-surfaces / Setting the</i> <i>number of contour bands / Selecting the type of boundary surface rendering / Designating the iso-</i> <i>surfaces as the boundary of truncated model / Limiting the range of iso-surface scale by actually displayed</i> <i>values / Setting the iso-surface level / Setting the contour scale</i>	
<b>Curve plotting of scalar data</b>	<b>7-39</b>
<i>Initiating curve plotting / Modifying curve plotting / Displaying the numerical value at the sampling</i> <i>point / Resizing the graph / Setting the options for curve plotting / Terminating curve plotting</i>	
<b>Surface plotting of scalar data</b>	<b>7-42</b>
<i>Setting the surface plotting options / Selecting the data item / Selecting the type of rendering for the</i> <i>plotted surface / Selecting the type of rendering for the source plane / Setting the scale of height / Setting</i> <i>the datum of the plotted surface</i>	

<b>Visualizing Vector Data</b>	<b>7-45</b>
<b>Visualizing vector data by arrows</b>	<b>7-45</b>
<i>Setting the arrow display options / Selecting the data item / Selecting the type of arrow rendering /</i>	
<i>Selecting the style of surface or boundary surface rendering / Setting the placement focus of arrows /</i>	
<i>Setting the position of arrow heads / Setting the length of arrows / Setting the thickness of arrows /</i>	
<i>Displaying selectively sampled arrows / Overlaying vector images</i>	
<b>Visualizing displacements by deformed shape</b>	<b>7-52</b>
<i>Setting the display options / Selecting the style of the deformed shape rendering / Overlaying the</i>	
<i>deformed shape image over the screen image / Retainint the deformed shape in future rendering /</i>	
<i>Excluding the rigid body displacements from deformed shape / Setting the deformation scale / Visualizing</i>	
<i>the displacements by animation / Restoring undeformed shape</i>	
<b>Getting numerical values of nodal data</b>	<b>7-56</b>
<b>Visualization of Multi-step Analysis Data</b>	<b>7-57</b>
<b>Stepwiae rendering of multi-step analysis results</b>	<b>7-57</b>
<i>Stepwise rendering of multi-step analysis results / Selecting the method of stepwise rendering / Ending</i>	
<i>stepwise rendering</i>	
<b>Stepwise rendering of sequentially staged analysis results</b>	<b>7-59</b>
<b>Visualization of Dynamic Analysis Data</b>	<b>7-60</b>
<b>Visualizing dynamic response</b>	
<i>Visualizing dynamic analysis results using multi-step view / Visualizing dynamic analysis results in</i>	
<i>time history form / Plotting time history records of nodal value in spatial coordinates / Plotting time</i>	
<i>history records in modal coordinates / Animation of dynamic motion / Displaying the trace of nodal</i>	
<i>movement / Changing the time step / Dragging the time step / Adjusting the animation speed and the</i>	
<i>deformation scale / Resizing "Dynamic Responce" window / Ending tie history display</i>	
<b>Visualizing dynamic mode shape</b>	<b>7-67</b>
<i>Starting and controlling mode shape display / Selecting the mode to display /Scale of display / Speed of</i>	
<i>animation / Ending dynamic mode display</i>	
<b>Visualization of combined Load Case Results</b>	<b>7-69</b>
<i>Characteristics of the analysis results with multiple load cases / Starting multi-loading view / Visualizing</i>	
<i>the analysis results of each load case / Visualizing the synthesized analysis results</i>	
<b>Visualization of Seepage Analysis Data</b>	<b>7-71</b>
<b>Visualization of scalar and vector data</b>	<b>7-71</b>
<i>Contouring of scalar data / Representing vector data</i>	
<b>Vesualization of phreatic surface</b>	<b>7-73</b>
<i>Rendering of phreatic surface in 2-D seepage analysis / Rendering of the phreatic surface in 3-D seepage</i>	
<i>analysis</i>	
<b>Visualization of flow path</b>	<b>7-76</b>



*Interactively displaying a flow path / Clearing flow paths / Getting flow velocity*

## **Image and Animation 7-83**

### **Image handling 7-83**

*Retrieving an image saved in a file / Saving the screen image in a file*

### **Animation 7-86**

*Creating an animation / Writing the animation script / Playing an animation*

## **Chapter 8 Diagrams for Frame Analysis 8-1**

### **Diagrams for Truss and Rigid Frame 8-2**

#### **Visualizing analysis data of 2-D and 3-D trusses 8-2**

*Displaying axial force diagram / Displaying axial force value / Displaying reactions at supported nodes / Displaying displacements*

#### **Visualizing analysis data of 2-D and 3-D rigid frames 8-4**

*Displaying axial force diagram / Displaying shear force diagram for 2-D rigid frame / Displaying bending moment diagram for 2-D rigid frame / Displaying deformed shape of rigid frame / Displaying support reactions of rigid frame / Displaying shear force diagram for 3-D rigid frame / Displaying bending or torsional moment diagram for 3-D frame*

### **Diagram Related Functions 8-9**

*Manipulating the diagrams / Displaying more than one diagrams / Clearing diagram / Redrawing diagram / Reversing diagram directions as a whole / Reversing diagram directions of selected element(s) / Adjusting the scale of diagrams*

#### **Setting display options 8-12**

*Instant redrawing mode / Curve normal mode / Turning on - off text of diagram values / Temporarily turning off text of diagram values / Popping up the hidden texts / Selectively turning on text of diagram values / Displaying part of the diagram for selected members only*

## **Chapter 9 Data Interface with External Softwares 9-1**

### **Overview of File Contents 9-2**

#### **File position 9-3**

#### **Input data for external solver 9-3**

#### **Output from external solver 9-4**

#### **Other data for user interface, graphical modeling and rendering 9-5**

### **Contents of Data Items 9-6**

#### **Modeling data 9-6**

*Modeling data master record / Modeling data position record / Header record / Node data / Element data / Boundary condition data / Element property data / Temperature data / Member joint data / Equivalent nodal force data / Equivalent nodal heat data / Nodal dynamic data / Integration scheme record / Solver*

*option record / Analysis output item record / Dynamic analysis setting record / Curve end point data / Curve data / Primitive surface data / Surface mesh data / Volume mesh data / Load condition data-header record / Load condition data-attribute record / Heat boundary condition data / View transformation data / Construction plane data / Symbol size record*

**Analysis data****9-44**

*Analysis data master record / Adaptive analysis data position record / Nonlinear analysis data position record / Dynamic analysis data position record / Custom contour menu data / Custom vector menu data / Analysis data items*

# **Chapter 1**

## **Overview**

## *Chapter 1 Overview*

# Chapter 1 Overview

VisualFEA is an integrated software for finite element analysis, which is an advanced numerical technique to solve and analyze physical problems arising in many fields of science and engineering. Numerous commercial or academic programs for finite element analysis have been developed and distributed all over the world so far. However, most of them are not much accessible for many potential users of the method, owing to various reasons: complexity of usage, high expenses, restricted portability, functional limitations and so on. The main objective of VisualFEA is to overcome such barriers between the user and the software, and make itself easily accessible and affordable for everyone who needs finite element analysis not only for practical use but also for educational purposes. The greatest advantage of VisualFEA is its ease of use and user friendliness. Its usage is simple, natural and intuitive for all the diverse and ample functionality it provides. This makes the software most unique and attractive, because finite element analysis involves complicated procedures, and accordingly finite element programs in general are far from user-friendly.

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

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

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

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

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

## Finite Element Analysis and VisualFEA

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

### Procedure of finite element analysis using VisualFEA

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

#### ■ Stages of finite element analysis

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

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

analysis results and presenting them in more understandable forms.

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

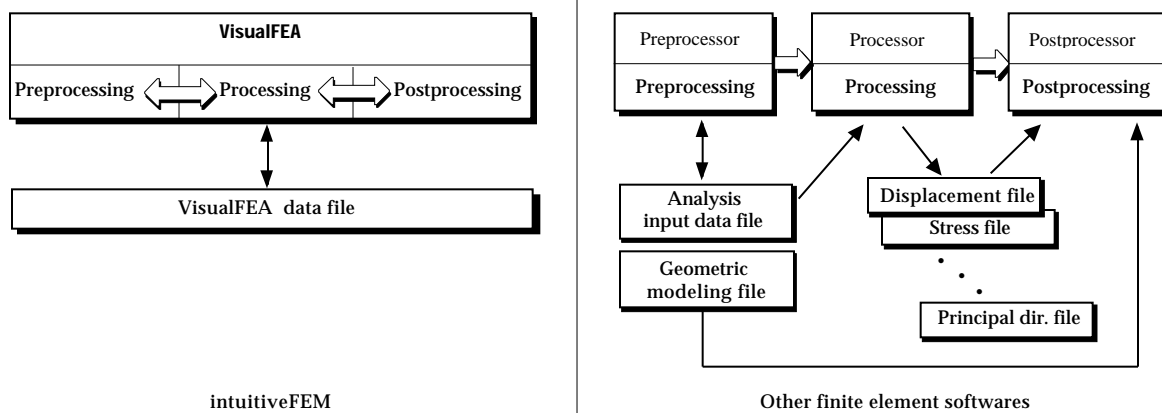
Some stages and procedures may be added or dropped depending on the analysis subject and the project contents.

### ■ Finite element analysis using VisualFEA

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

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

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



< Comparison of software and data construction >

### ■ Mixed use of VisualFEA and external solvers

Although the entire procedure of finite element analysis can be completed by a single execution of VisualFEA as described above, there is an alternative procedure in which an external solver is used together with VisualFEA.

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

### Creating a finite element model using VisualFEA

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

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

The above items are in fact the input data items used at the processing stage. There are other data items which are necessary only for pre- and postprocessing purposes in VisualFEA.

A finite element model is completed by creating nodes and elements and assigning attributes, boundary conditions and external effects.

### ■ Node and element data

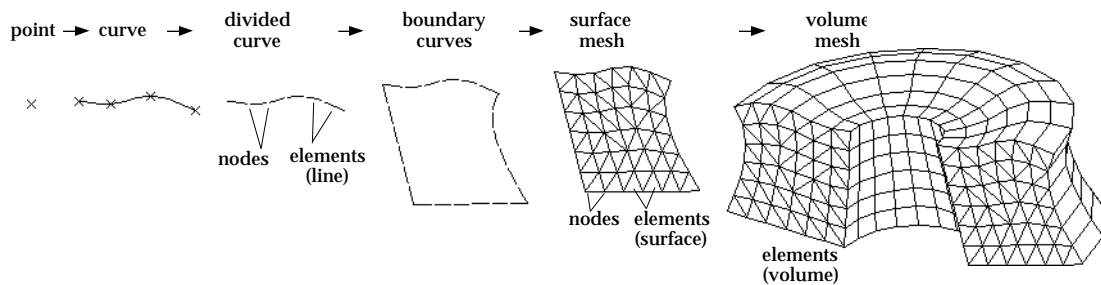
The geometry of a finite element model is defined by nodes and elements in VisualFEA. An element has a specified number of nodes. The node-element relations are established as a part of a finite element mesh. Some nodes are created by dividing curves used as a seed for mesh generations, and the others are created by mesh generation.

There are line elements for frame analysis, surface elements for plane or shell analysis, and volume elements for 3-dimensional solid analysis. Line elements are



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

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



< Hierarchical construction of line, surface and volume elements >

## ■ Other data assignment

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

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

## Visualizing analysis results using VisualFEA

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

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

### ■ Visualization of scalar data

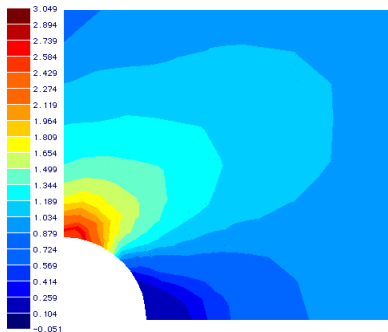
Stresses, strains and temperatures are examples of scalar data obtained from finite element analysis. The most widely used method of visualizing scalar data is contouring. The data distribution is represented by a number of contour lines or bands. However, contouring is limited to representing the data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

### ■ Visualization of vector data

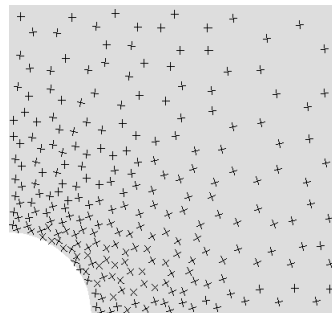
Examples of vector data are displacements, reaction forces, heat fluxes and principal directions. The most widely used method of visualizing vector data is arrow representation. Different types of arrow representation are available for planes, curved surfaces, or 3 dimensional volumes. Vector data like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

### ■ Visualization of truss and frame data

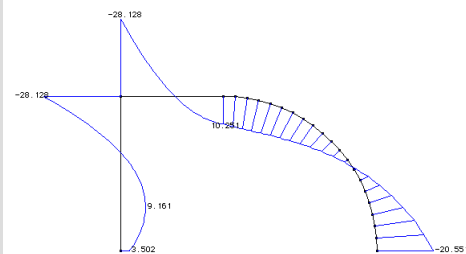
The analysis results of trusses and frames are best expressed by diagram images. Member forces such as bending moments, shear forces and axial forces are represented by diagrams. Diagrams are drawn in continuous curves to represent the variation of the values across the frame members.



Visualization of scalar data



Visualization of vector data



Visualization by diagram

< Different methods of visualizing analysis results >

## Framework of VisualFEA

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

### Basic user interface

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

#### ■ Project and file

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

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

#### ■ View control

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

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

### ■ 3-D geometric data input system

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

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

### ■ Model selection and control

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

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

### ■ Model rendering

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

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

## Preprocessing

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

## ■ Curve and surface primitive modeling

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

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

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

## ■ Mesh generation

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

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

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

## ■ Data Assignment

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

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

- Flux
- Nodal source
- Initial water table

## ■ Others

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

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

## Processing

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

## ■ Structural analysis

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

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

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

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

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

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

#### ■ Heat conduction

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

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

#### ■ Seepage

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

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

## **Postprocessing**

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



## ■ Data visualization

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

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

## ■ Visualization aids

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

- Parallel planes
- Cross planes
- Slicing
- Capping

## ■ Image handling

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

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

## ■ Printing

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

- Text
- Screen image

### ■ Others

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

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

## Educational aids

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

### ■ Element stiffness computation

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

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

### ■ Assembly and solution process

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

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

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

### ■ Shape function and interpolation

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

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

### ■ Eigen mode

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

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

### ■ Stress recovery and smoothing

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

and visualize graphically the stress recovery and smoothing procedure. The computational aspects and characteristics of the procedure can be studied and understood easily using this educational function.

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

#### ■ Adaptive analysis process

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

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

#### ■ Structural behavior

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

## **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.

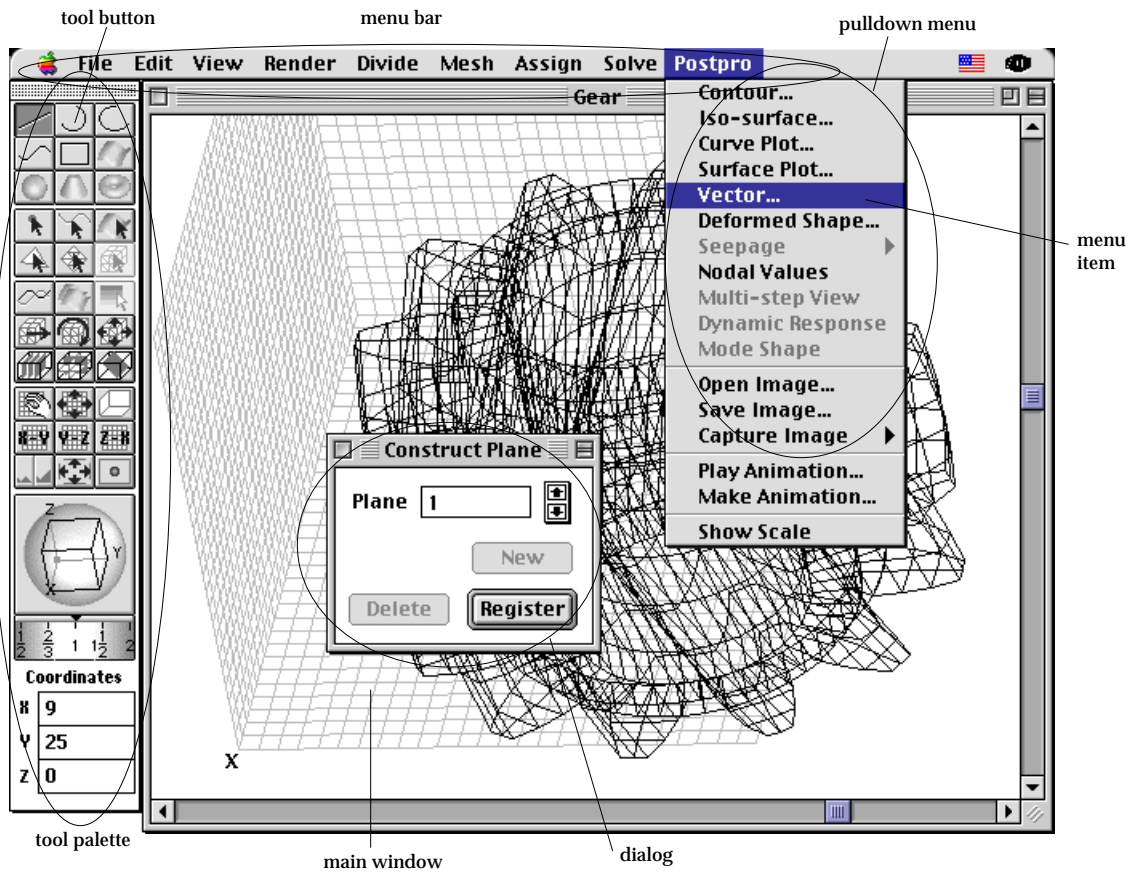
View of the model on the screen can be controlled and adjusted interactively either using the mouse or keyboard input. There are various tools for view transformation including view rotating, zooming and panning. In addition, the direction of view or the part of zooming can be specified directly. Usage 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.



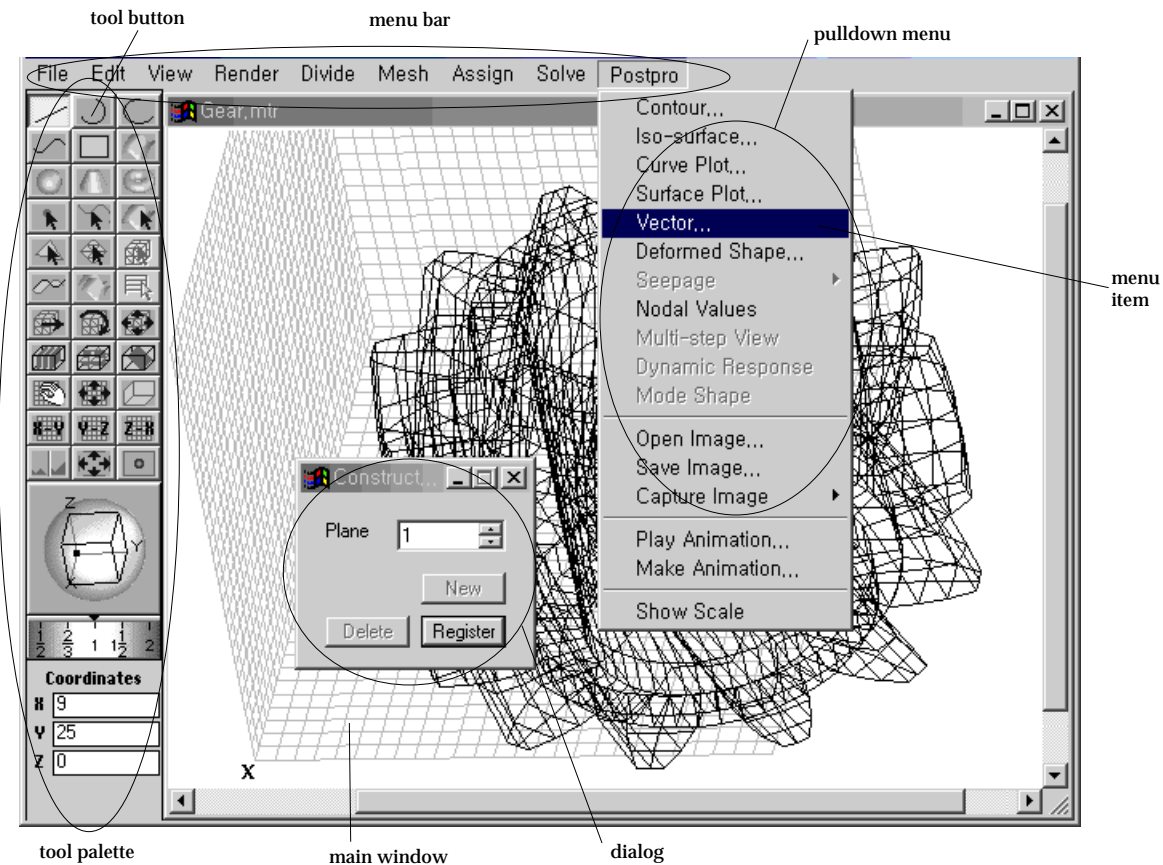
< Basic elements of user interface (Mac OS version) >



## 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.

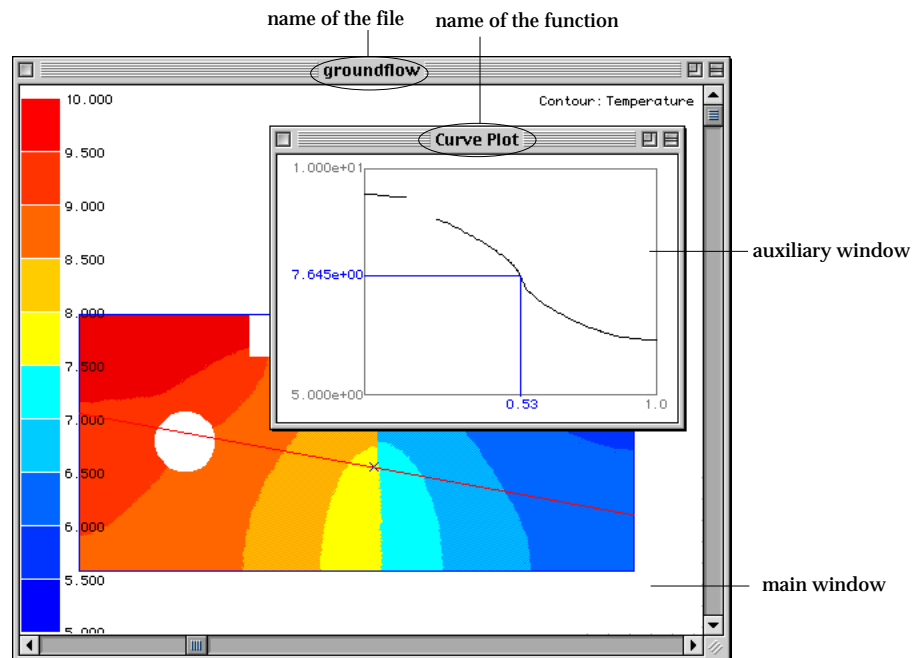
## Window and dialog

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.

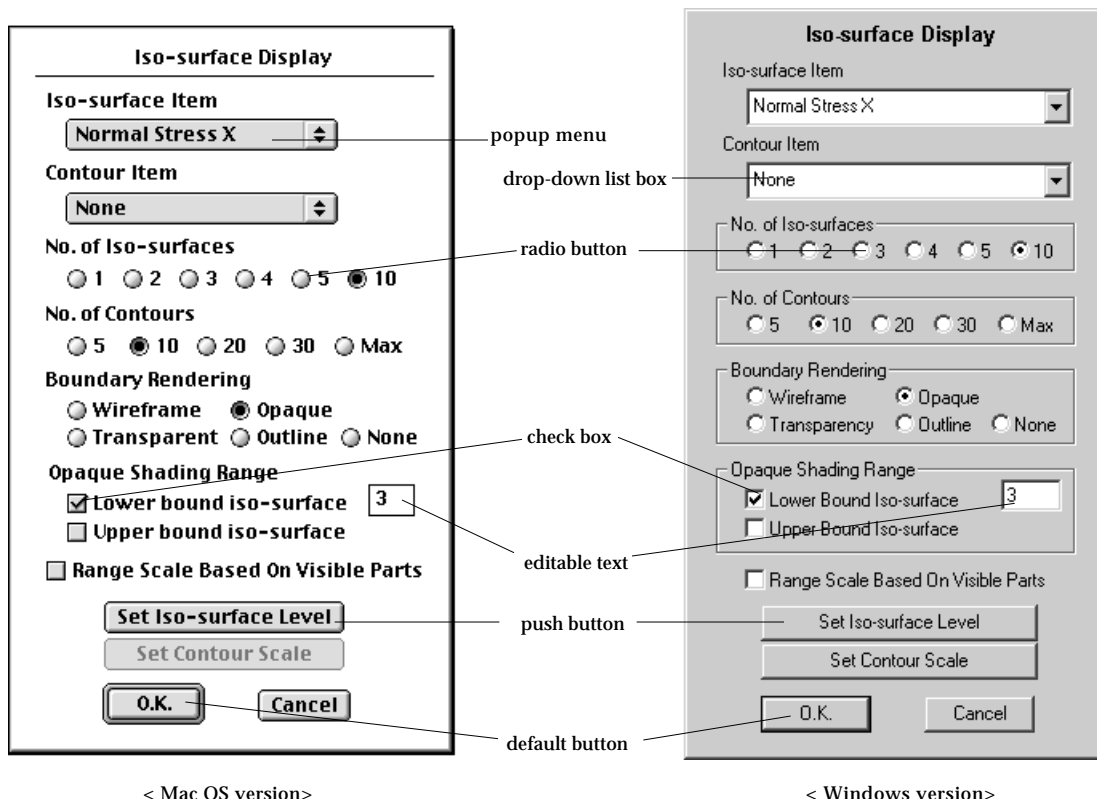


< Main window and auxiliary window >

## ■ 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 >

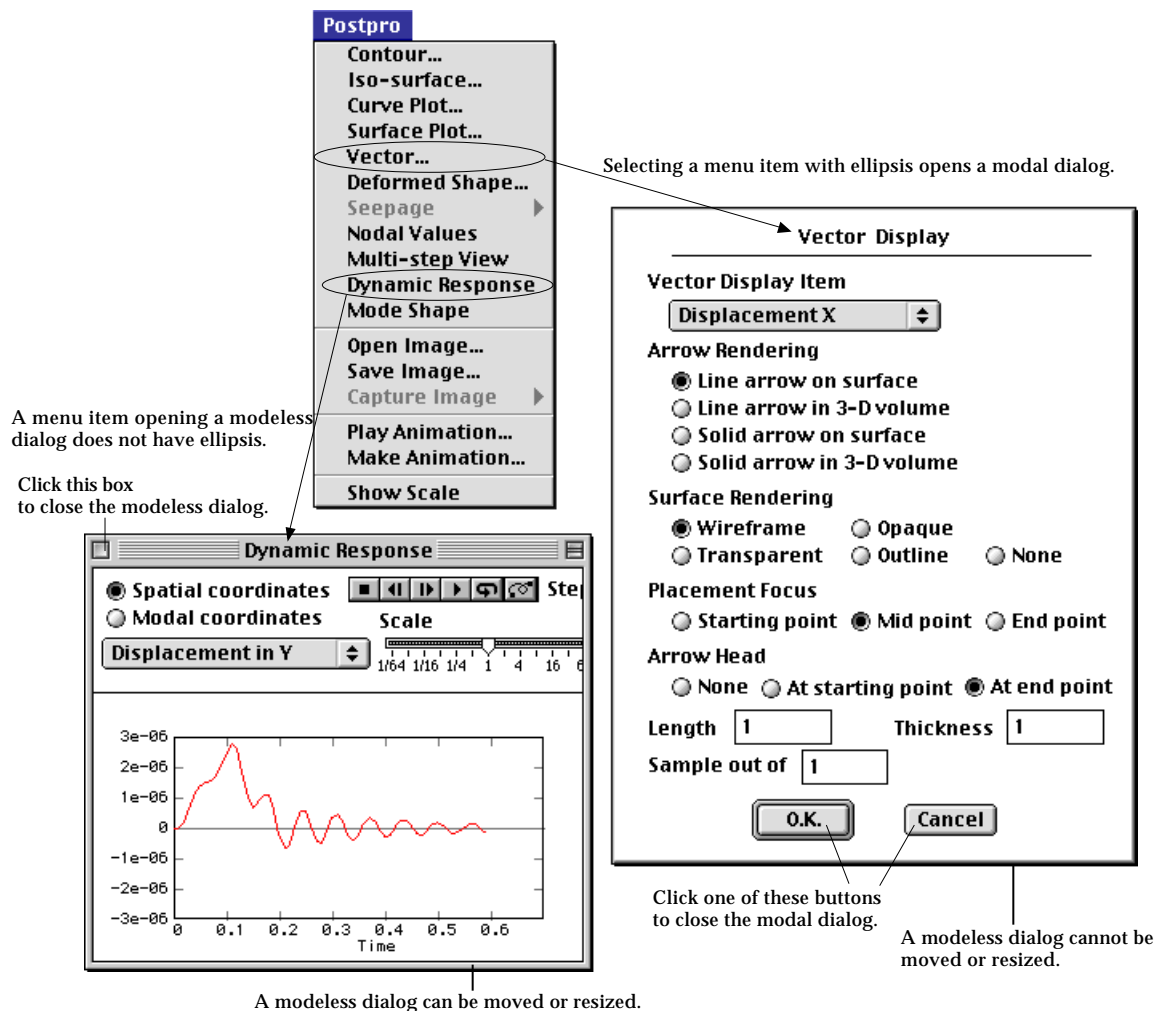
< Windows version >

< 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 **O.K.** or **Cancel**. 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.



< Comparison of a modal dialog and a modeless dialog >

## 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

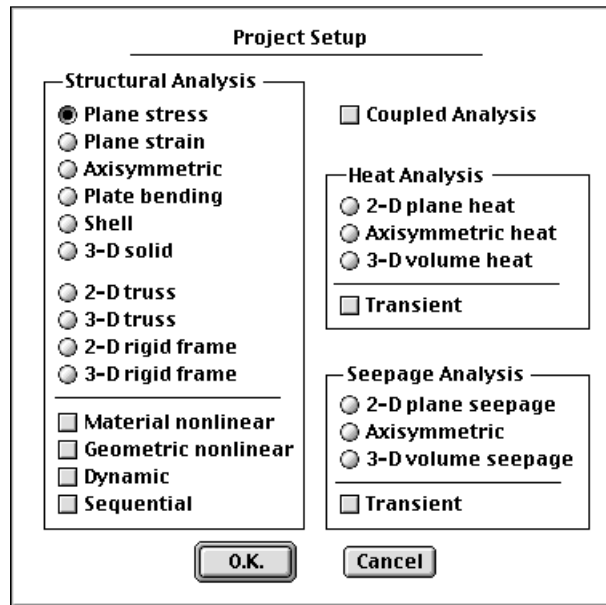
VisualFEA can be started by double clicking either VisualFEA icon or a project file icon. The data file name has extension .mtr for Windows version. In case a project file is double clicked, the file is opened with the launching of the software. This can also be achieved either by clicking one of those 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.

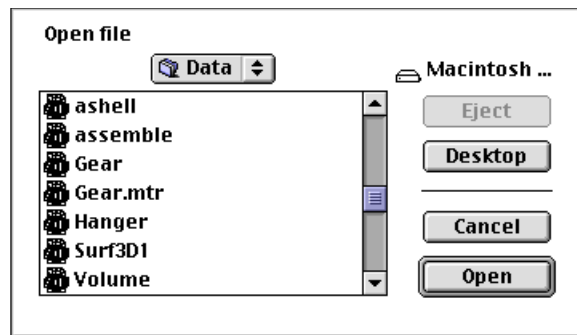
There are 2 options respectively for heat conduction and seepage analysis:

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

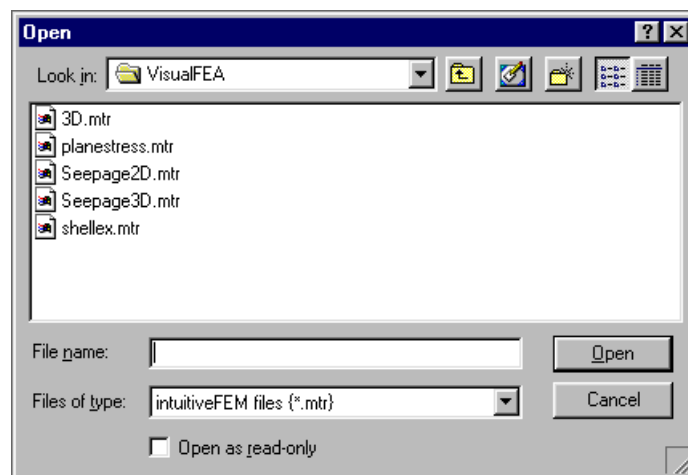


## ■ Opening a project file

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) >

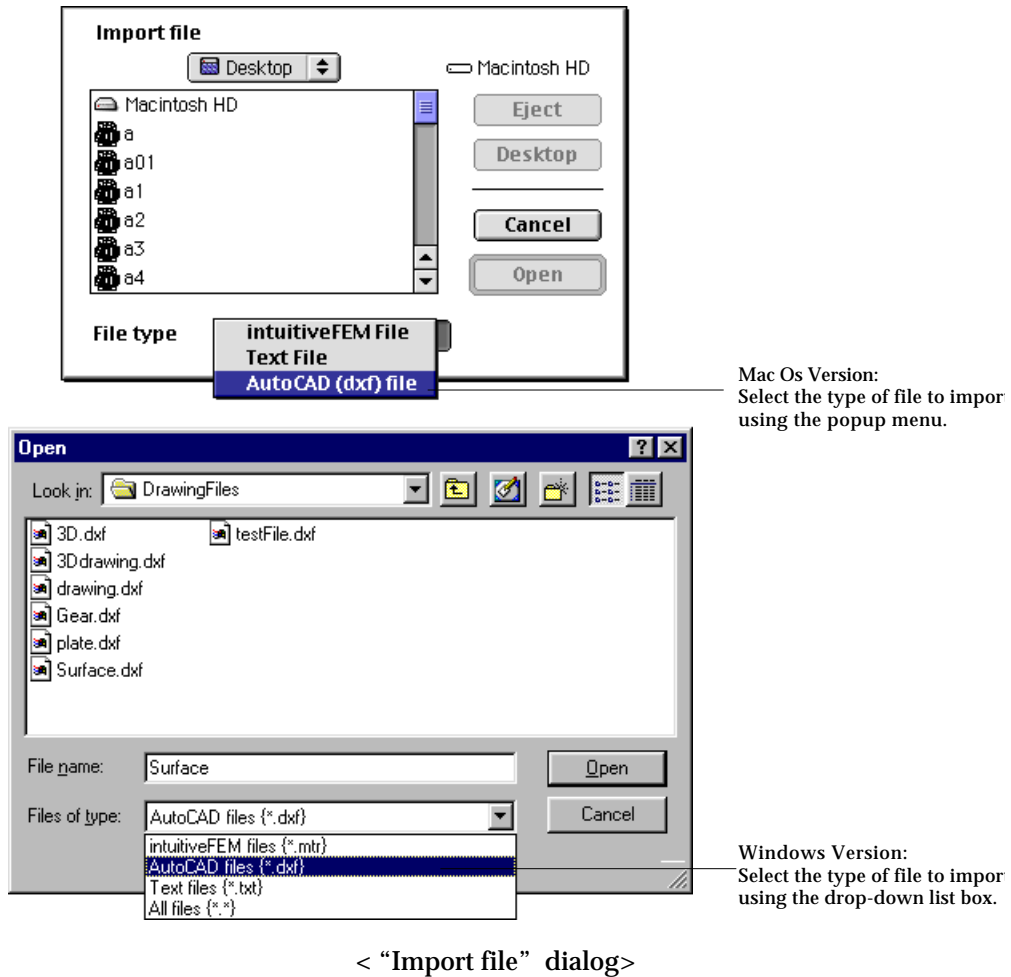


< “Open” dialog (Windows version)>

## ■ 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.

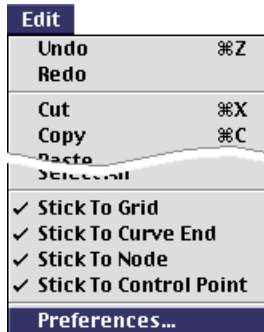
### ■ Printing

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.

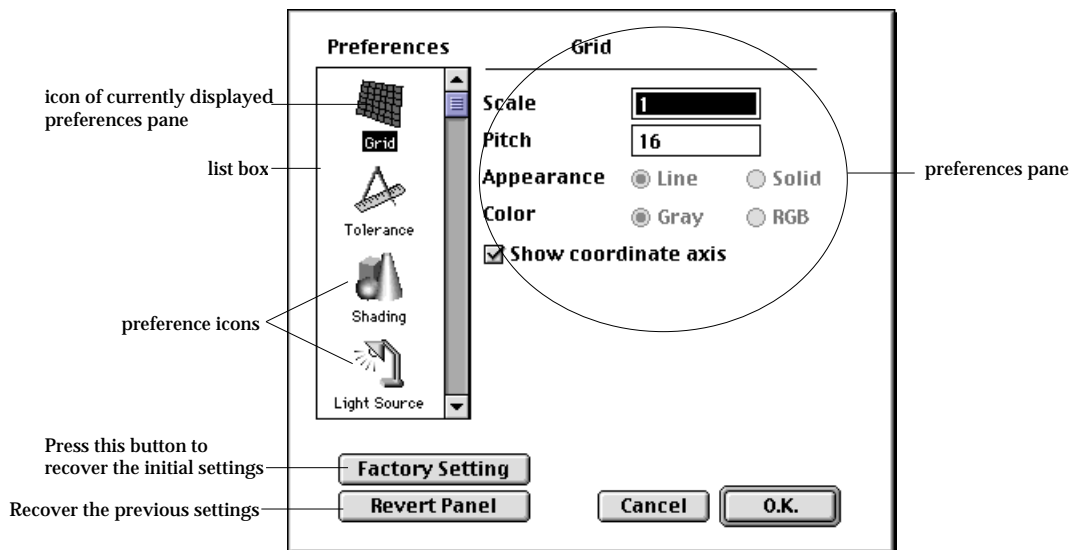
### ■ Quitting VisualFEA

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.

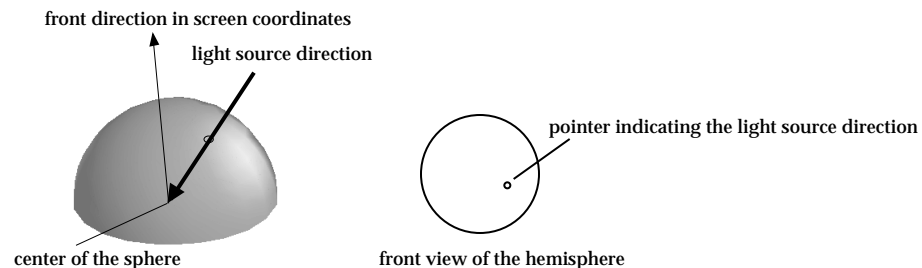
- “No. of Gray Levels”: Shading of an object is represented by gray levels. this option determines how many gray levels are used in shading. This option is also applied to contouring with shading.
- “Rendering Method”: Two methods of rendering are used for shading or contouring in VisualFEA. One is rendering 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

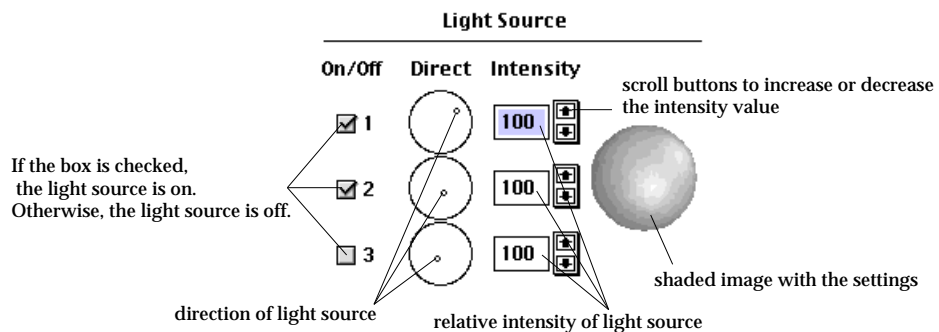
Parallel point light sources are used in shading. There are 3 light sources, with different directions and intensities, each of which can be turned on or off. The light source pane also shows an image of a sphere shaded with the settings.

- “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.

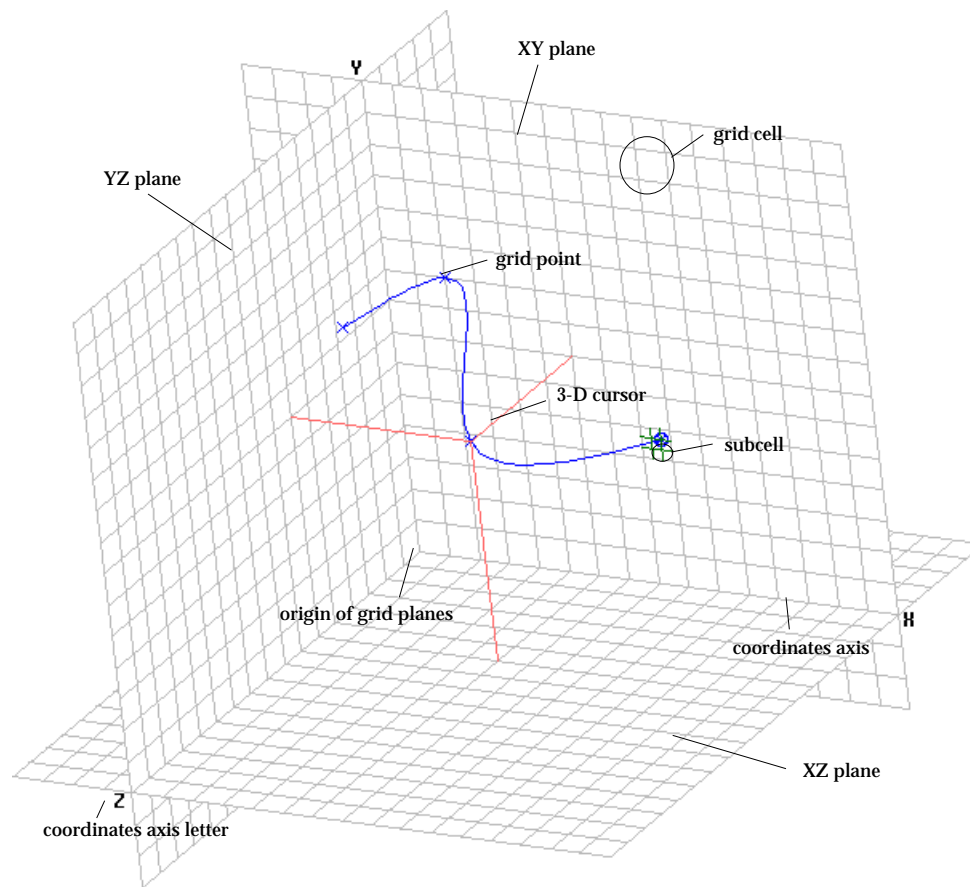
- “Gradient Recovery and Smoothing”: There are 4 radio buttons to select the method of computing secondary variables such as stresses, strains, heat fluxes and so on. For the first 3 methods, the nodal values are averaged between elements connected at the node so that the values are continuous across 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 >

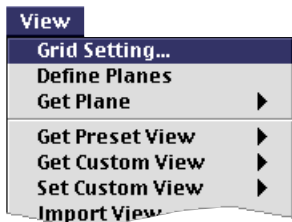
## Grid



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.

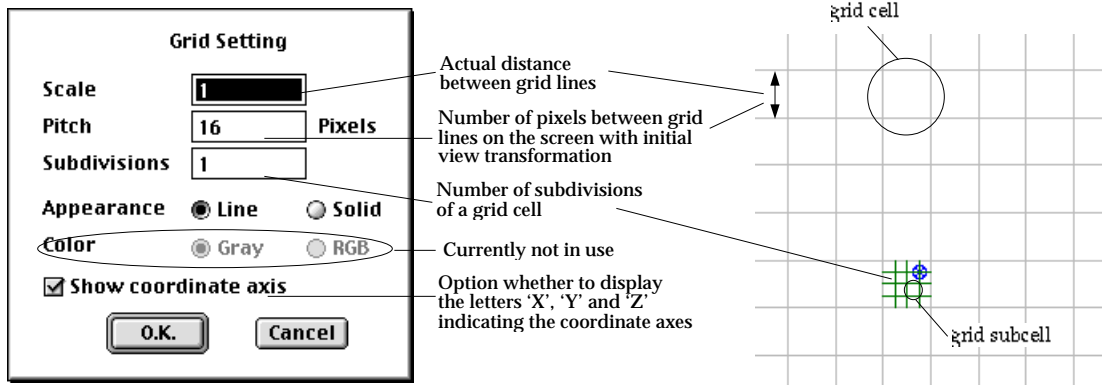
### ■ Setting grid

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.

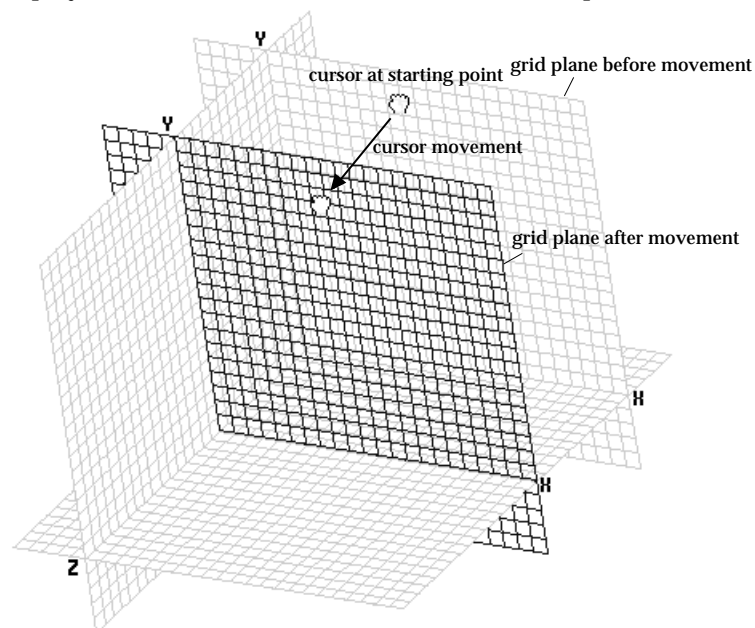
## ■ Moving grid planes



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:

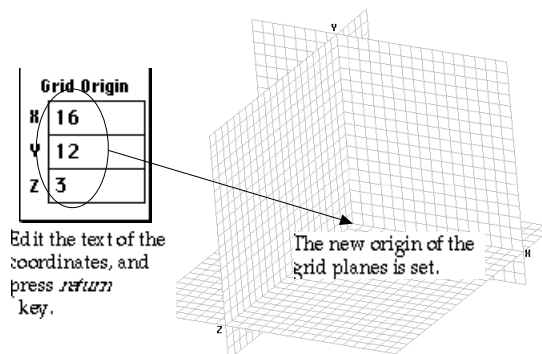
- 1) Start the grid movement tool by pressing the tool button , if it is not yet activated.
- 2) Do the view transformation if necessary.  
It is easy to move a plane facing obliquely to the front of the screen.
- 3) Place the screen cursor over the grid plane to move.  
The cursor shape changes into .
- 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.







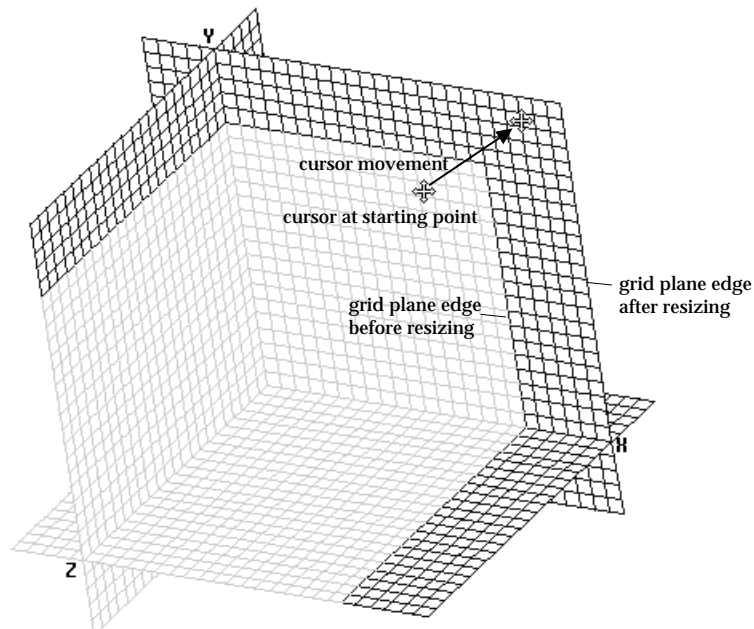
&lt; Moving grid planes by keyboard &gt;



### ■ 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 .

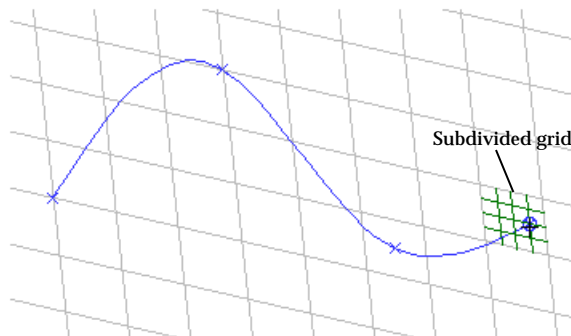
- 3) Drag the  mark to the subdivided grid point to be used as the input point.

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

- 4) Release the mouse button.

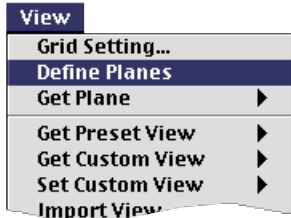
The coordinates of the point with  mark is entered as the new input point.

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

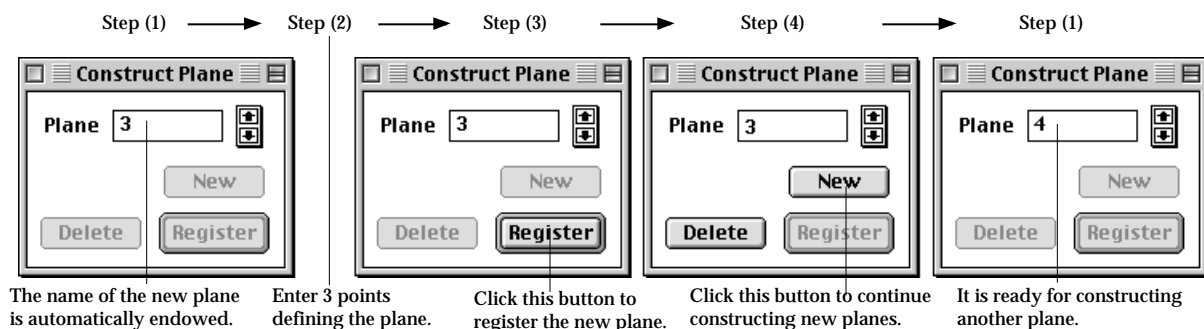


The system defined XYZ grid plane consists of three planes, i.e., XY, YZ and ZX planes which are respectively normal to Z, X and Y axes. User can define grid planes other than these ones. Their origin and their normal direction may be set arbitrarily as desired. It is very convenient, in some cases, 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.

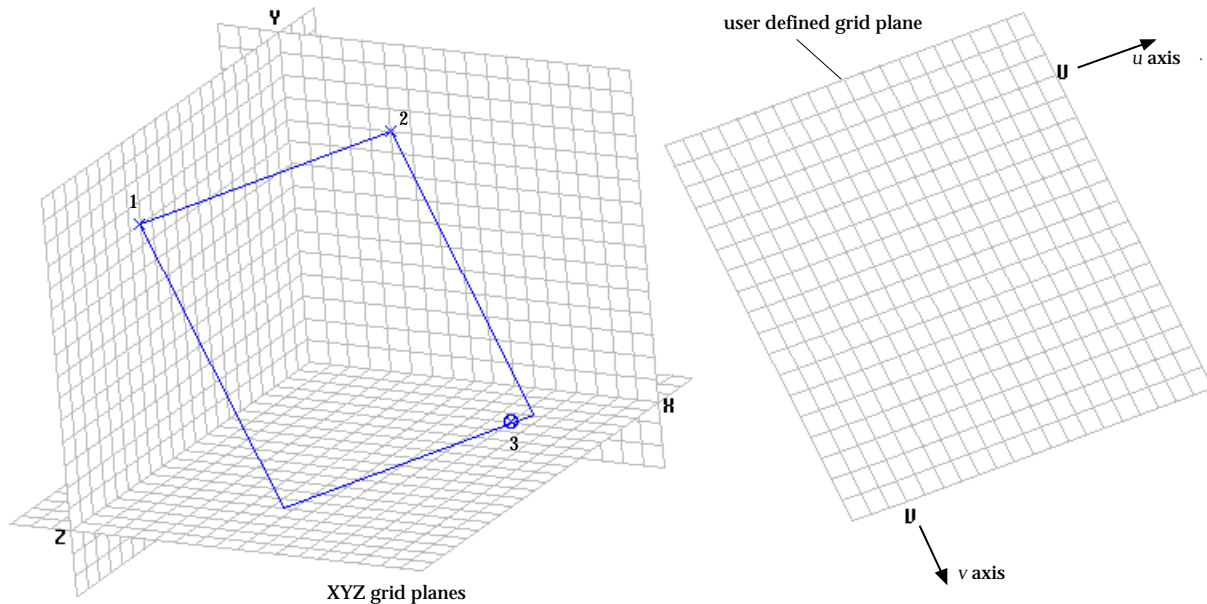


< "Construct Plane" dialog in each step of defining grid planes >



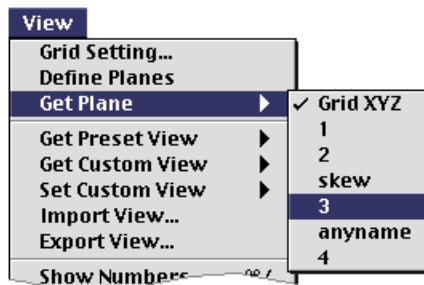
- 4) Close the "Construct Plane" dialog in order to end constructing grid planes and start entering coordinates using the grid planes. Or, click **New** button, and repeat step 2) and 3) to add more grid planes.

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

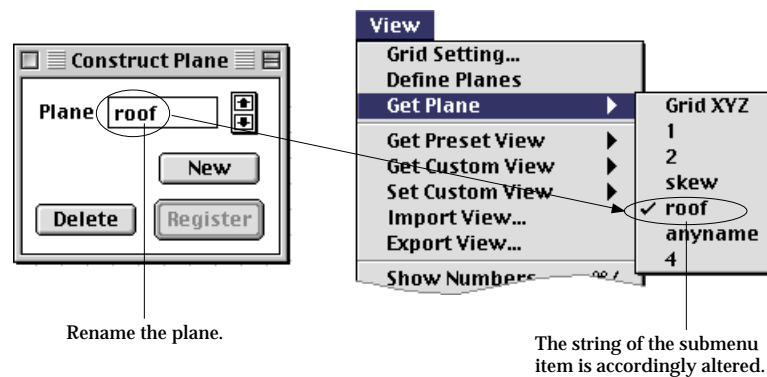
User defined grid planes can be deleted by the following steps.

- 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.

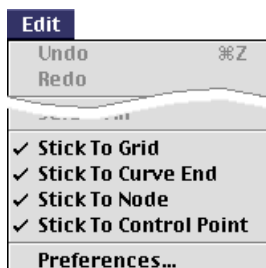
- 3) Press the mouse button.
- 4) Drag the 3-D cursor foot to the desired position by keeping the mouse button and *control* key pressed, while moving the screen cursor.

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

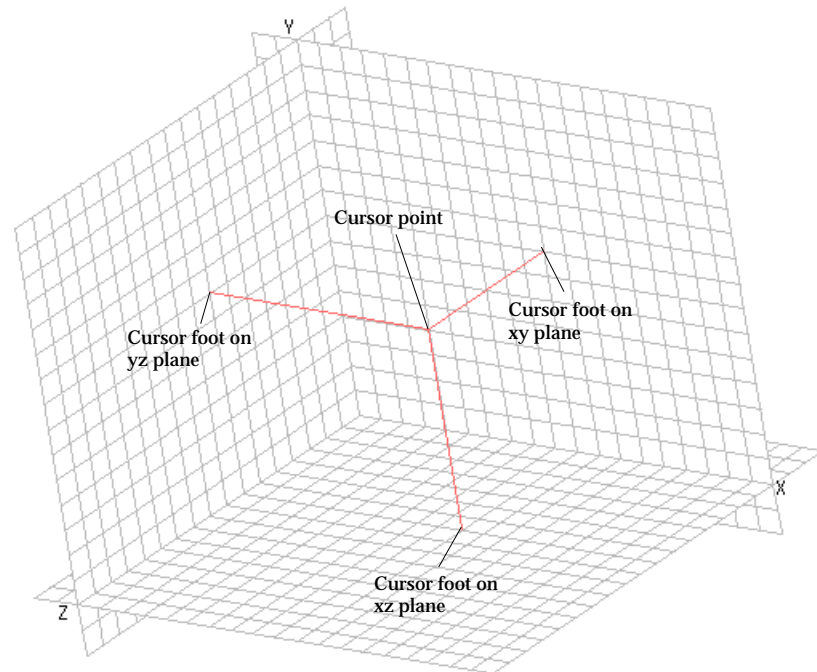
- 5) Release the mouse button.

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

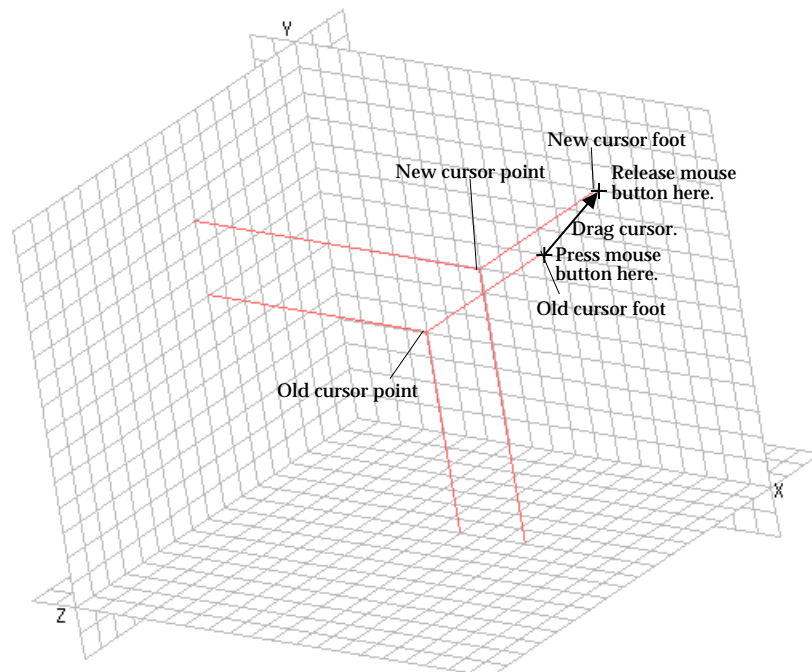
- 6) Move the screen cursor over to another 3-D cursor foot, and repeat 1)-5) until the 3-D cursor point settles at the desired position.



"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.

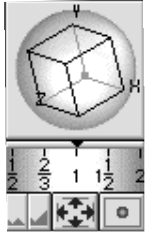


&lt; Composition of 3-D cursor &gt;



&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.*

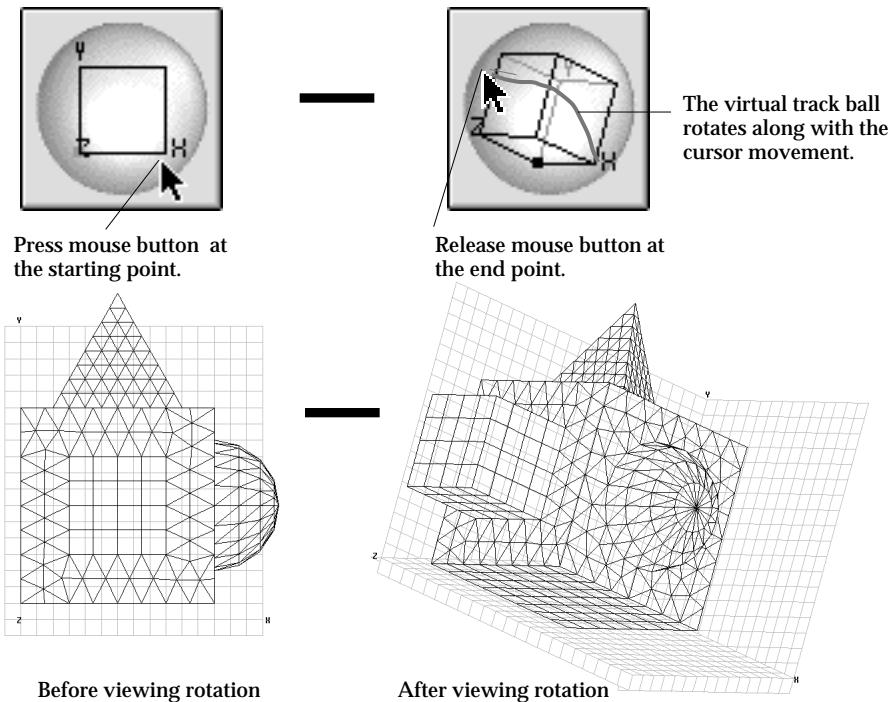
### ■ Rotating view using virtual track ball

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.

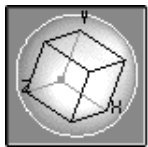
Any action in progress is not interrupted by operation of virtual track ball. For example, if you entered part of control points of a spline curve before rotating view, you can complete the curve by entering the remaining control points after the rotation.





<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

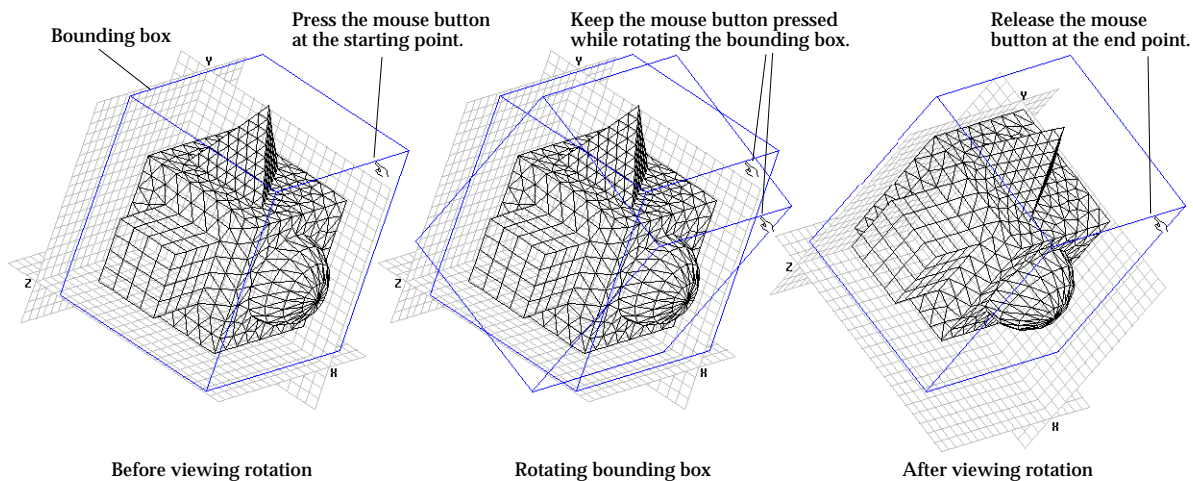
- 1) Lock the virtual track ball by clicking it with  key (Windows:  key) pressed, if it is not locked.

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.

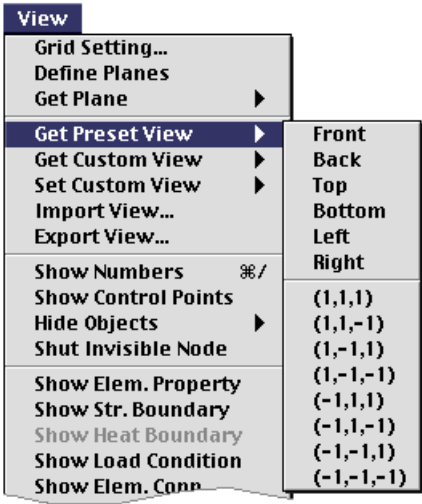


< Rotating view using bounding box >

■ 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 .

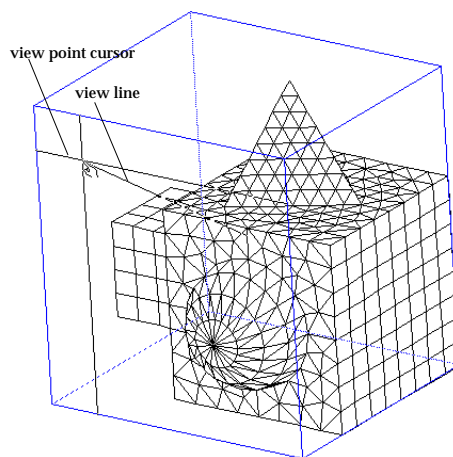
The new view point is marked by cross hairs, and the view line is represented by a straight line connecting the point and the center of the bounding box.

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

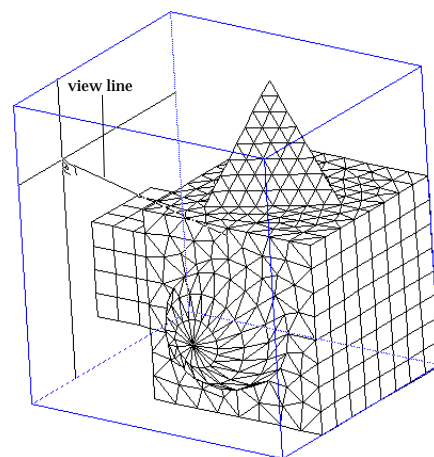
- 3) Double click the mouse button.

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

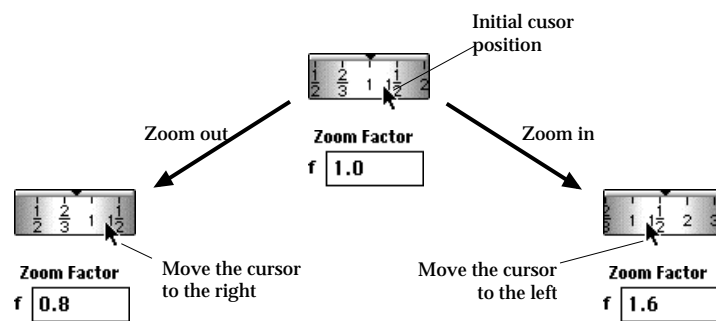


The modeling objects can be drawn in different scales depending on the required detail. Enlarging and reducing the display scale are termed here as "zooming in" and "zooming out" respectively. This zooming operation can be achieved in many different ways as explained below.

### ■ 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.



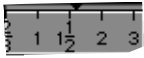
< Zooming in and out using the zoom dial >

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.

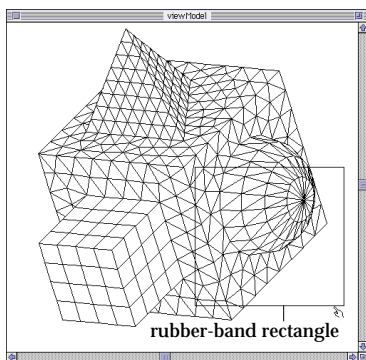
- 3) Press the mouse button, and draw a rubber-band rectangle by moving the mouse diagonally across the screen.

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.

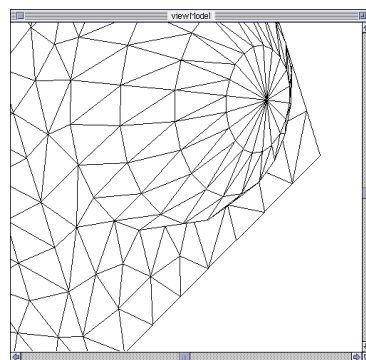
- 4) Release the mouse button.

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.

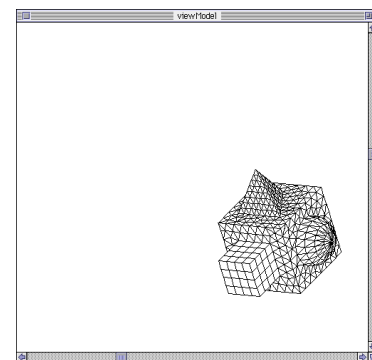
The zoom dial is still effective in locked state. and may be used for zooming. In order to unlock the zoom dial, click it once again with  key (Windows :  key) pressed, or start any other tool..



Before zooming




After zooming in  
(without pressing control key)

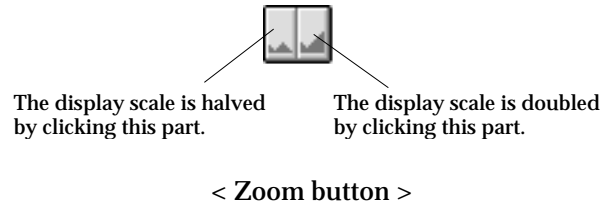


After zooming out  
(with pressing control key)


< 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

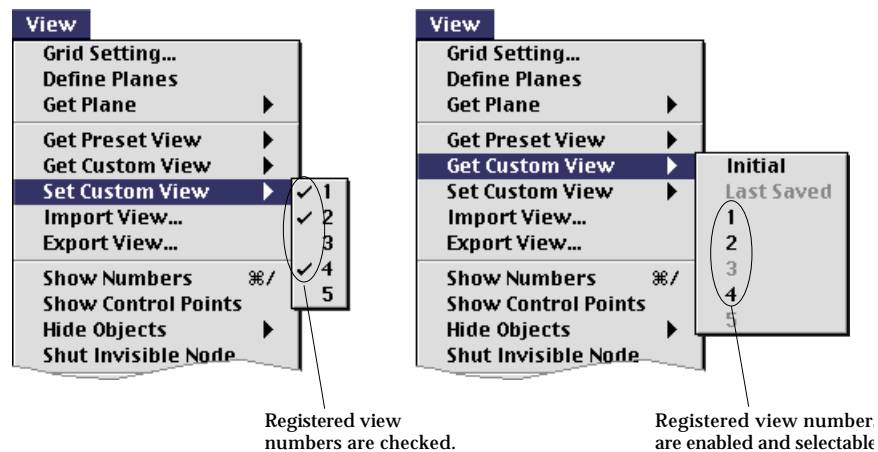
You may sometimes want to get back to the current view state after going through many view transformations. For this reason, there are functions for setting a few custom views and getting one of them later if necessary. VisualFEA allows setting up to 5 custom views.

### ■ 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

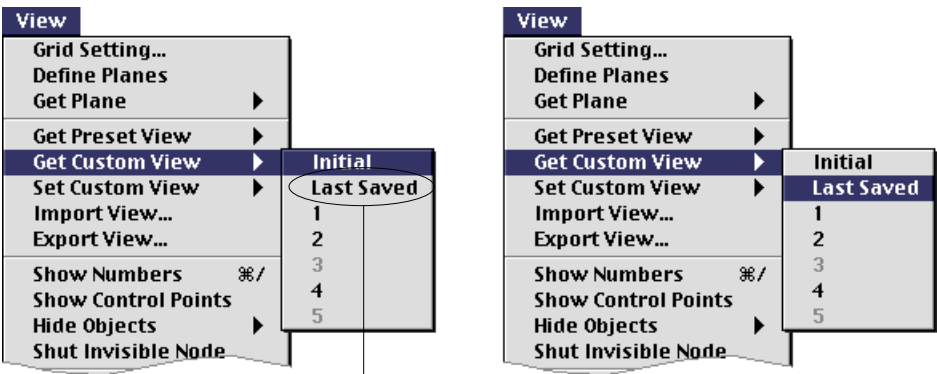
The initial view is the one set when the project begins, and the last saved view is the one set when the project file is opened or saved. It is useful to retrieve one of these view states, especially when you get lost after improper viewing transformations.

### ■ 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.

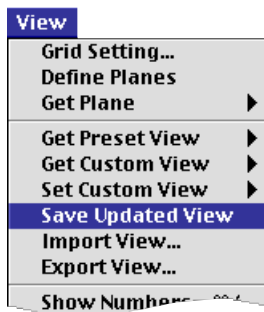
< Menu items for the initial view and the last saved view >

## 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:

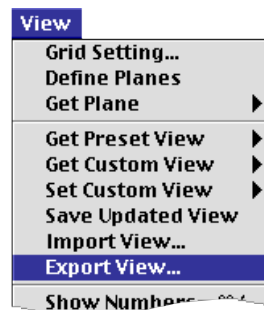
- Current viewing transformation : rotation, zoom, pan
- Custom views
- Grid settings : grid scale, grid pixel and grid origin
- User-defined grid planes

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



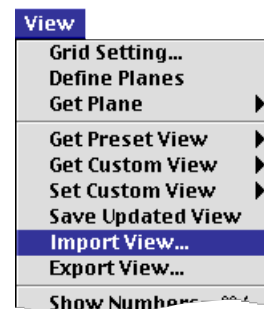
### ■ Updating view data in the working file

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.



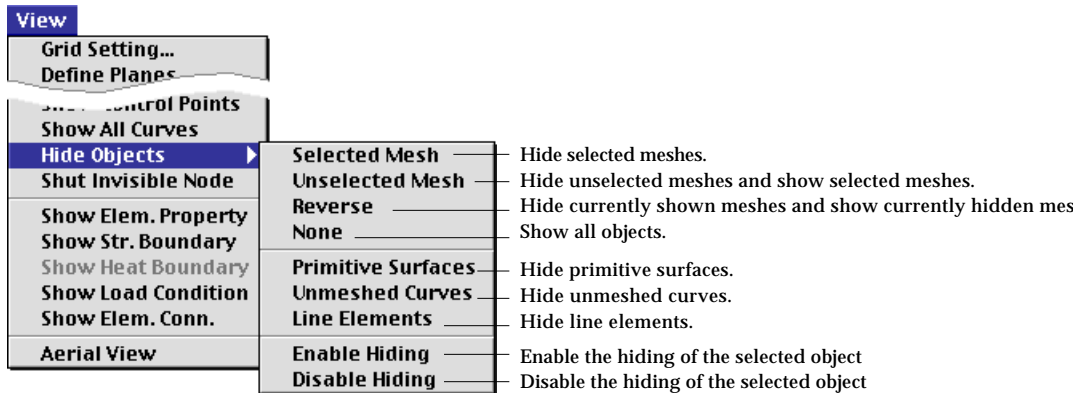
### ■ Importing views

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

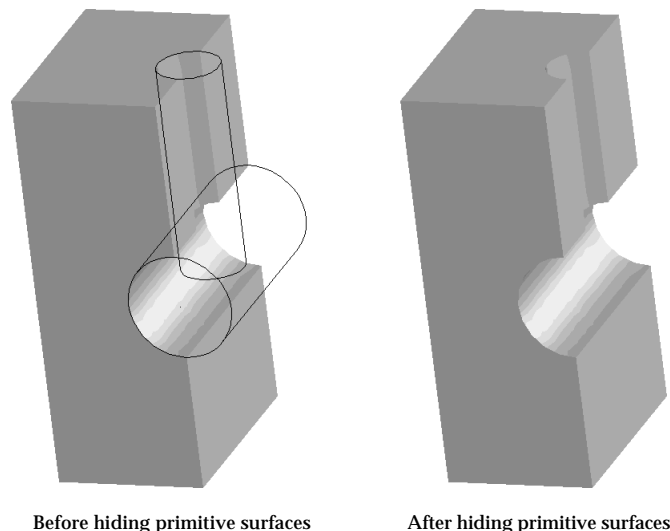
If you choose "Reverse" item from **Hide Objects** submenu, currently invisible meshes become visible, and visible meshes become invisible.

### ■ 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.



Before hiding primitive surfaces

After hiding primitive surfaces

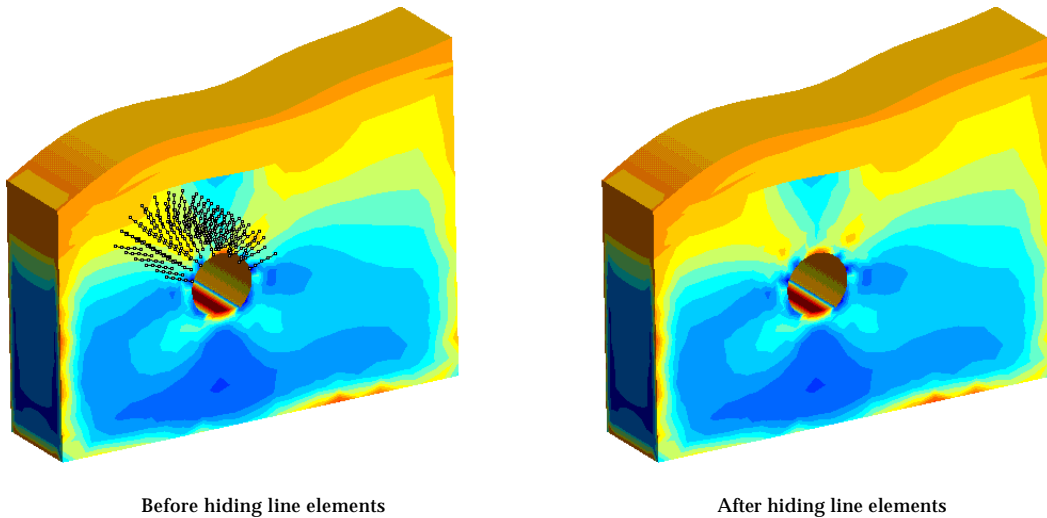
<Example of hiding primitive surfaces>

### ■ 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

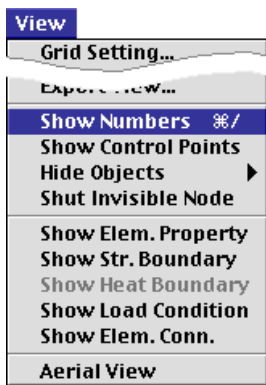
Surface meshes as components of a volume mesh become visible or invisible depending on the visibility of the volume mesh. Likewise, the visibility of the curves which belong to a surface mesh is determined by the visibility of the surface mesh. However, some curves or surfaces need be made visible regardless 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.*

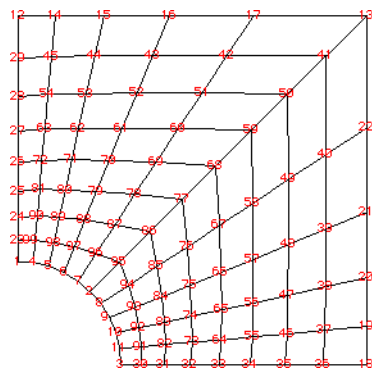
- 2) Select objects whose numbers are to be shown.

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.

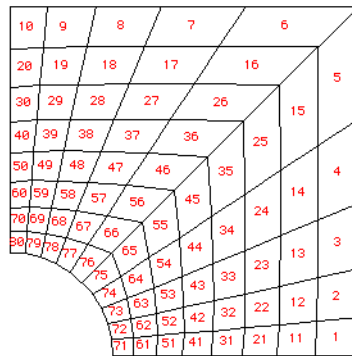
- 3) Choose "Show Numbers" item from **View** menu.

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

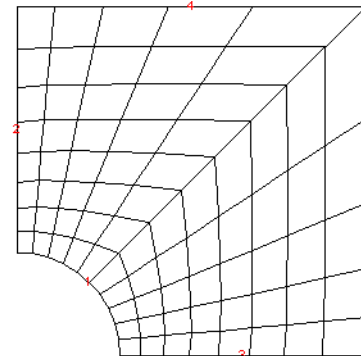
*Only element assigned with element property are numbered. Therefore, numbers will not be shown for elements without property assignment.*



Node number



Element number



Curve number

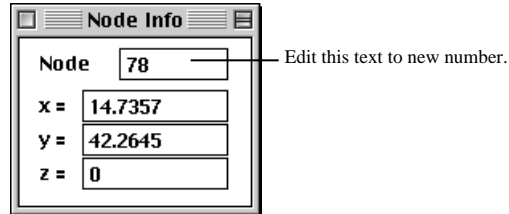
<Examples of displayed numbers>

### ■ 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.

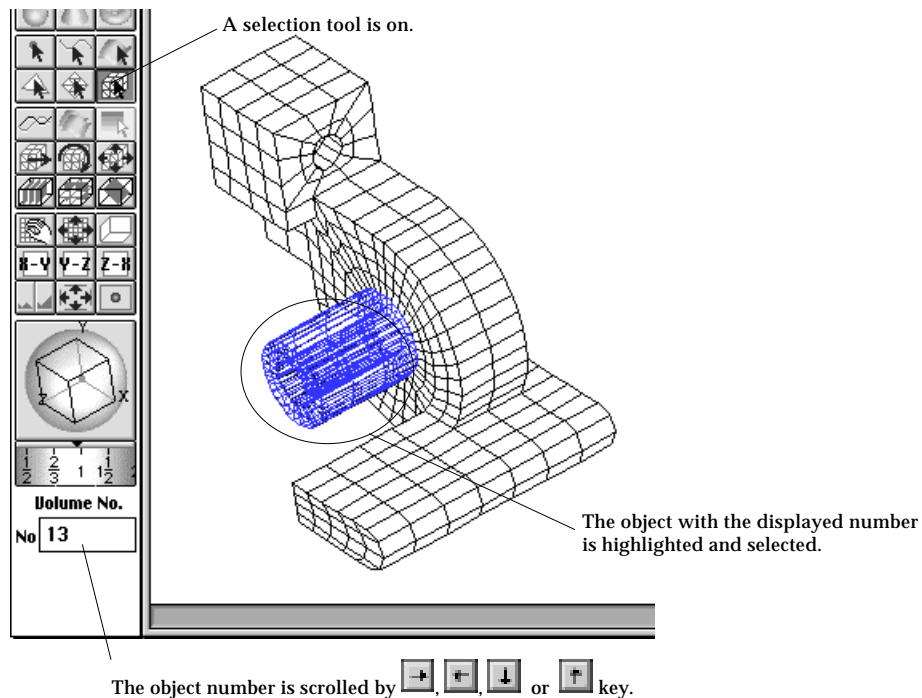


### 3) Change the number text.

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

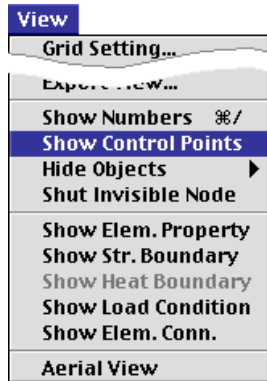
An object can be selected by its number. When a selection tool is on, scrolling the object number using **→**, **←**, **↓** or **↑** key highlight the object with currently displayed number in the editable text item at the bottom of the tool palette. The highlighted object is included in the selection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

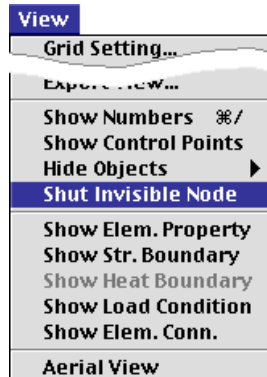


### ■ Displaying control points of curves

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.

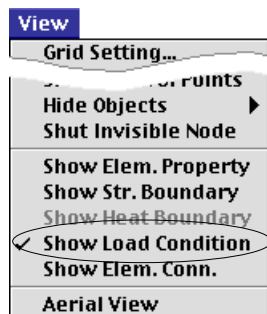
### ■ Making invisible nodes unselectable

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

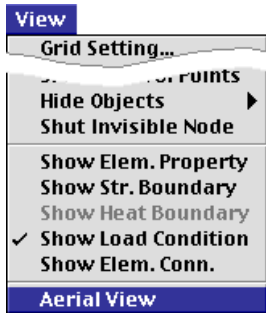
The assignments of various attributes, such as load conditions, boundary conditions and so on are displayed only when the assignment is going on and thus the corresponding dialog is on. It is sometimes necessary to make those assignments constantly displayed regardless of the modeling state. This can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

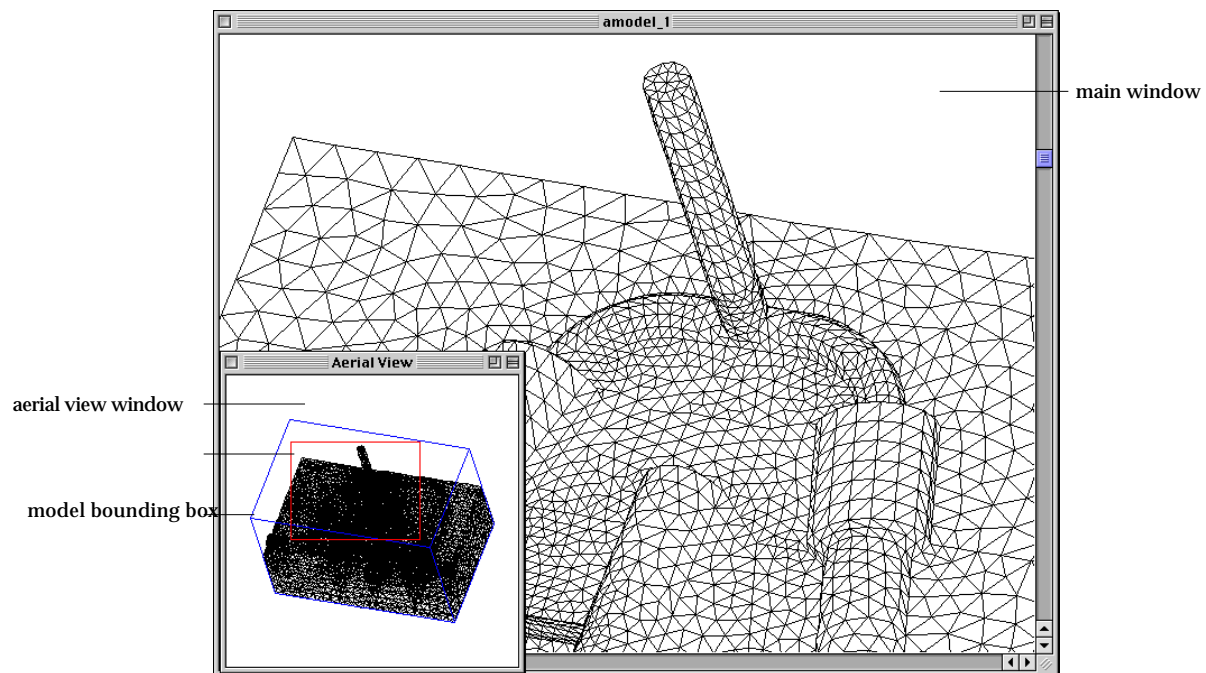
- 2) Drag the window bounding rectangle to pan the display of the model.

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.

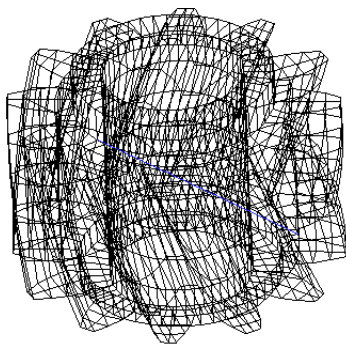
- wireframe mesh without hidden line removal
- wireframe mesh with hidden line removal
- outline
- shading
- transparency shading
- broken mesh

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.

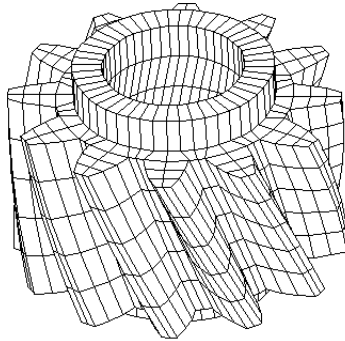
#### ■ Rendering by wireframe with or without hidden line removal

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.

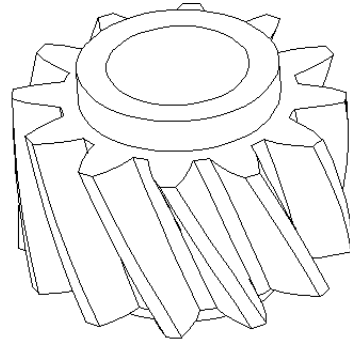




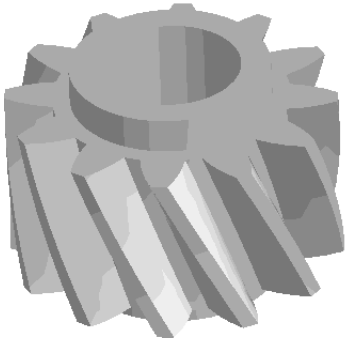
Wireframe



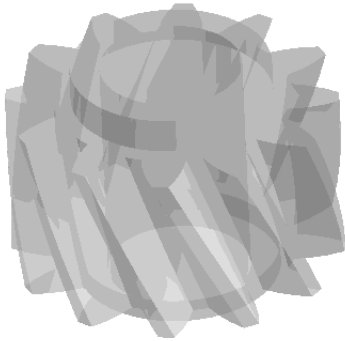
Wireframe with hidden lines removed



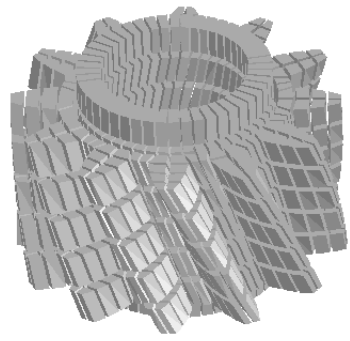
Outline



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.

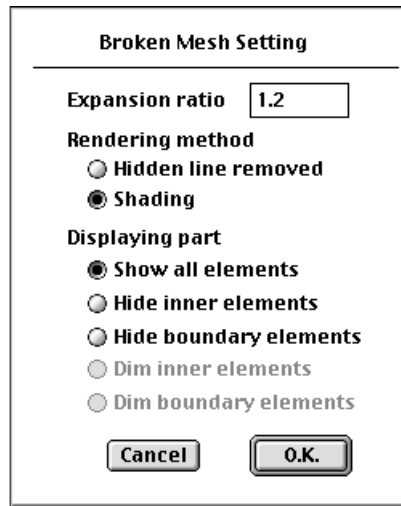
### ■ Rendering by shading or transparency shading

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.

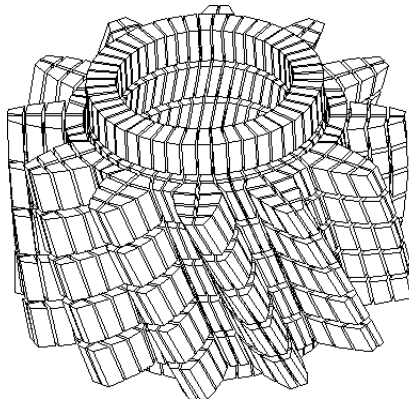
### ■ Rendering by broken mesh

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.

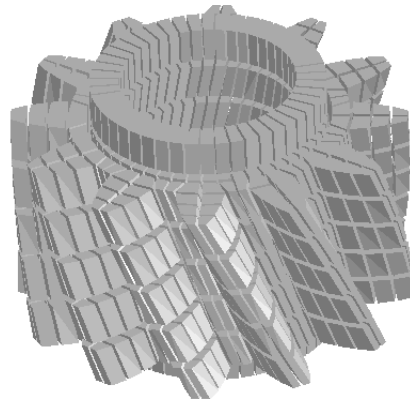


The dialog has following items:

- 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:

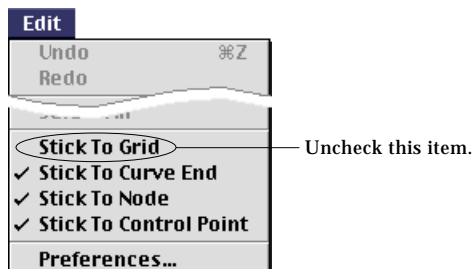
- a point constrained on grid planes
- a grid point
- a control point of an existing curve or a surface primitive
- an existing node point
- 3-dimensional cursor point

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.

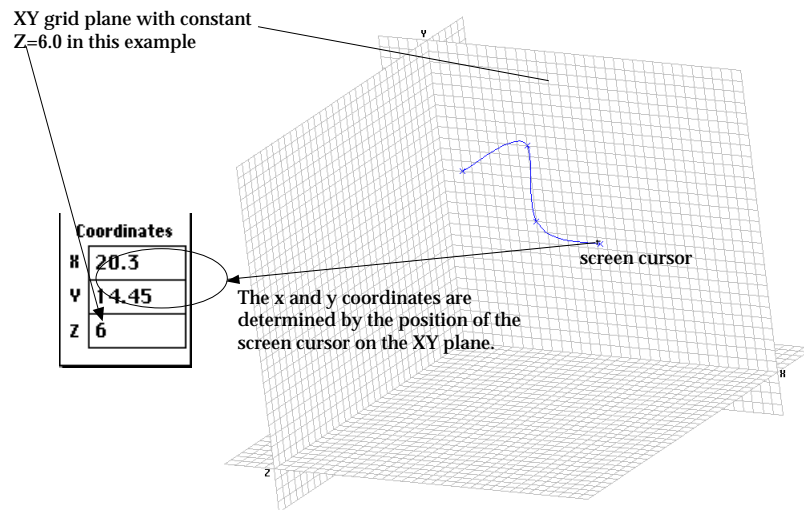
While the mouse is pressed, the coordinates at the cursor point are echoed in the text boxes at the bottom of the tool palette. If you move the mouse

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.

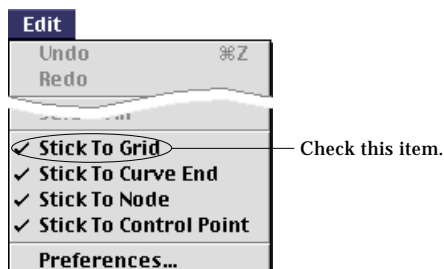


< Example of entering coordinates using the grid planes >



## ■ 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.

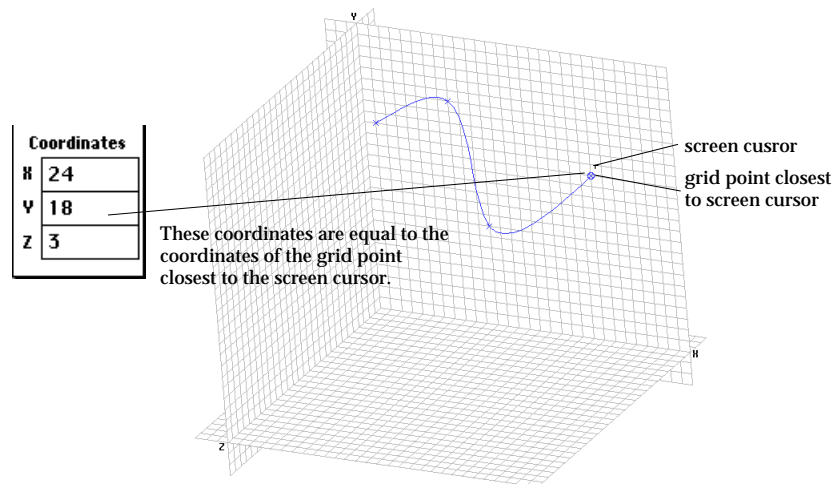


- 3) Move the cursor close to a grid point.
- 4) Press the mouse button.

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

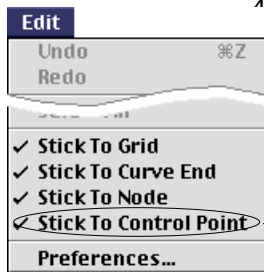
The coordinates can be entered using control points of previously created curves or surface primitives by the following steps.

- 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.

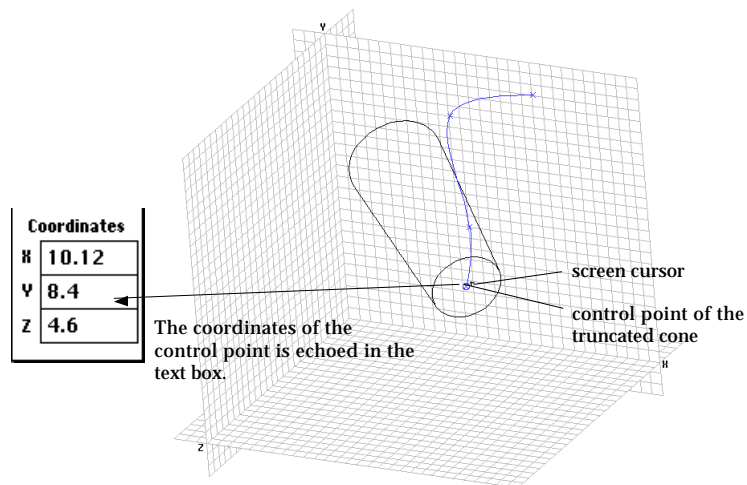
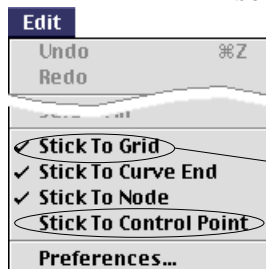
At the moment the mouse button is pressed, the control point closest to the screen cursor is marked by  and the coordinates of the control point are echoed in the text boxes at the bottom of the tool palette.

- 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.

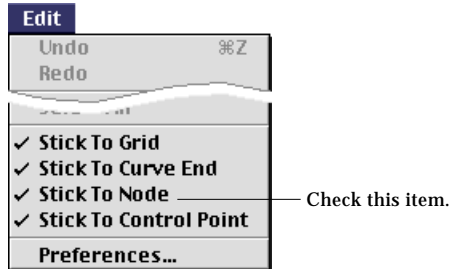


< Example of entering coordinates using a control point of a primitive surface >


### ■ 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.

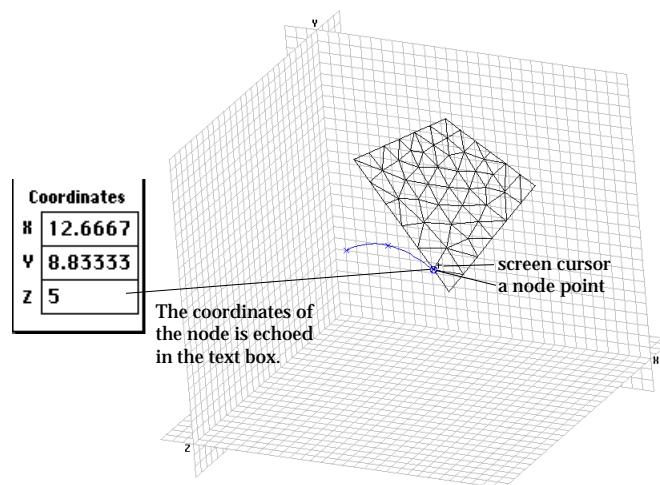


- 2) Move the cursor close to a node.
- 3) Press the mouse button.

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.

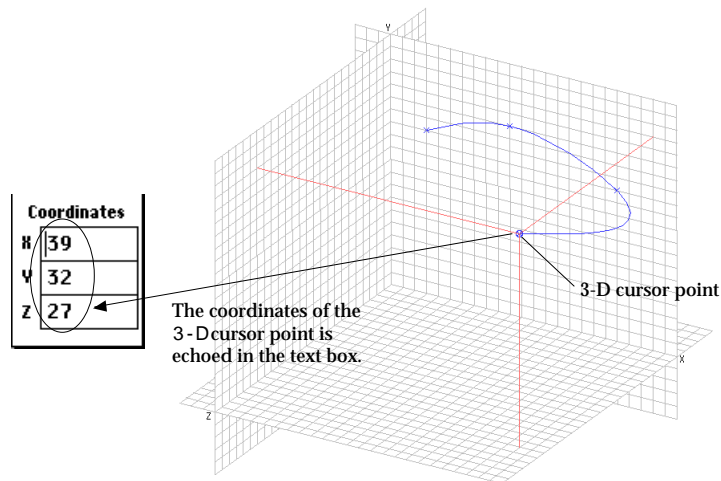
- 2) Move the 3-D cursor point to the desired position.

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.



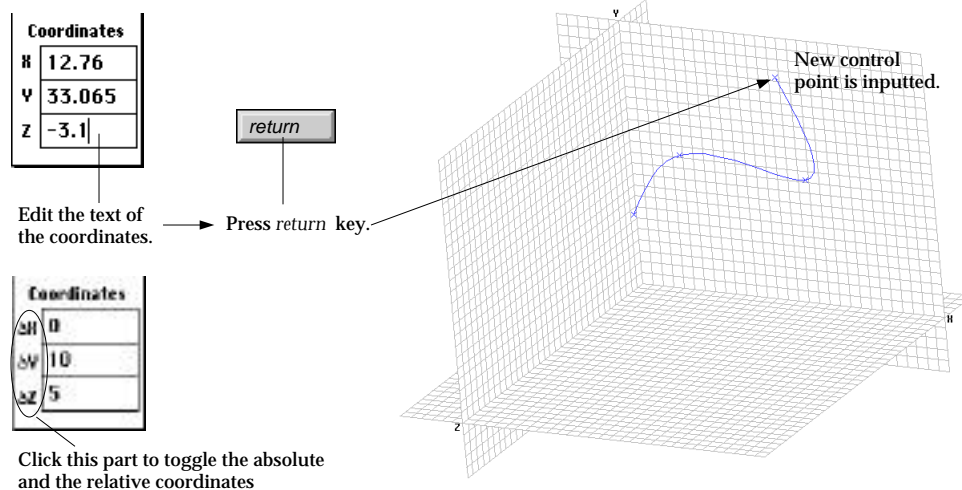
< Example of entering coordinates using 3-D cursor >

## 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.

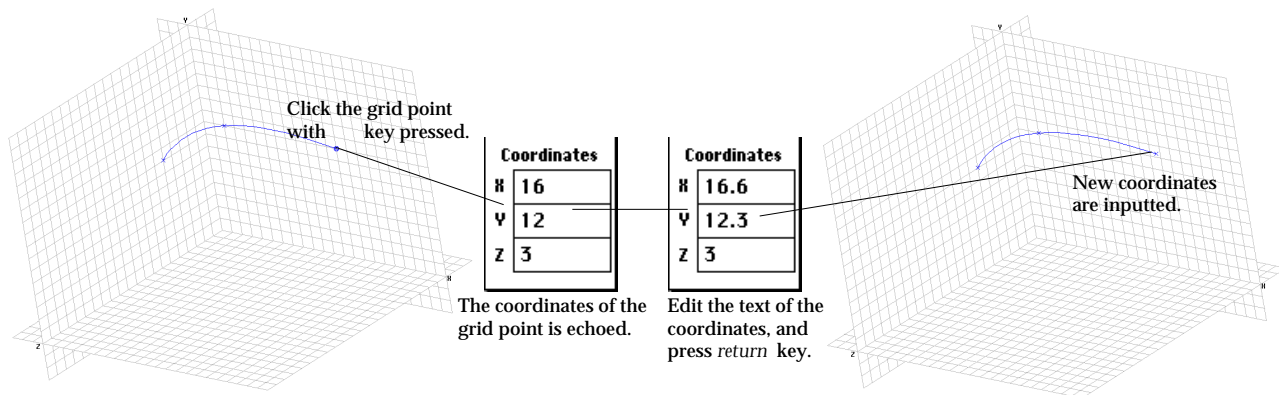
### ■ Inputting the coordinates by offset distance

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.

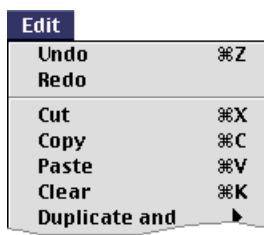



< Example of keyboard input with combined use of the grid points >

## ■ 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

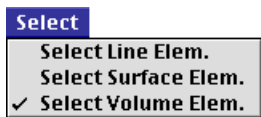
The surface primitive selection tool  is used to select surface primitive(s). When the tool button is pressed, a text box for inputting or displaying a surface primitive number appears at the bottom of the tool palette. The selected surface primitive number is shown in this text box only when a single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



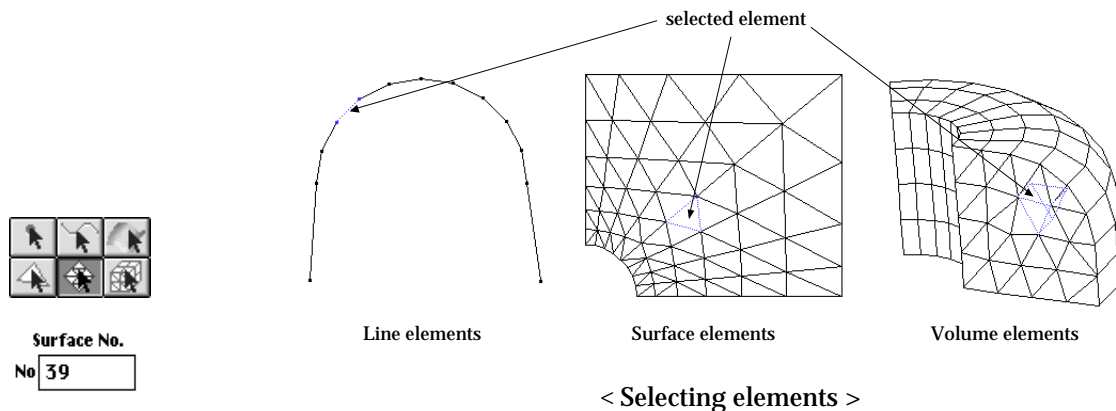
Element No.  
No 141

### ■ 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:

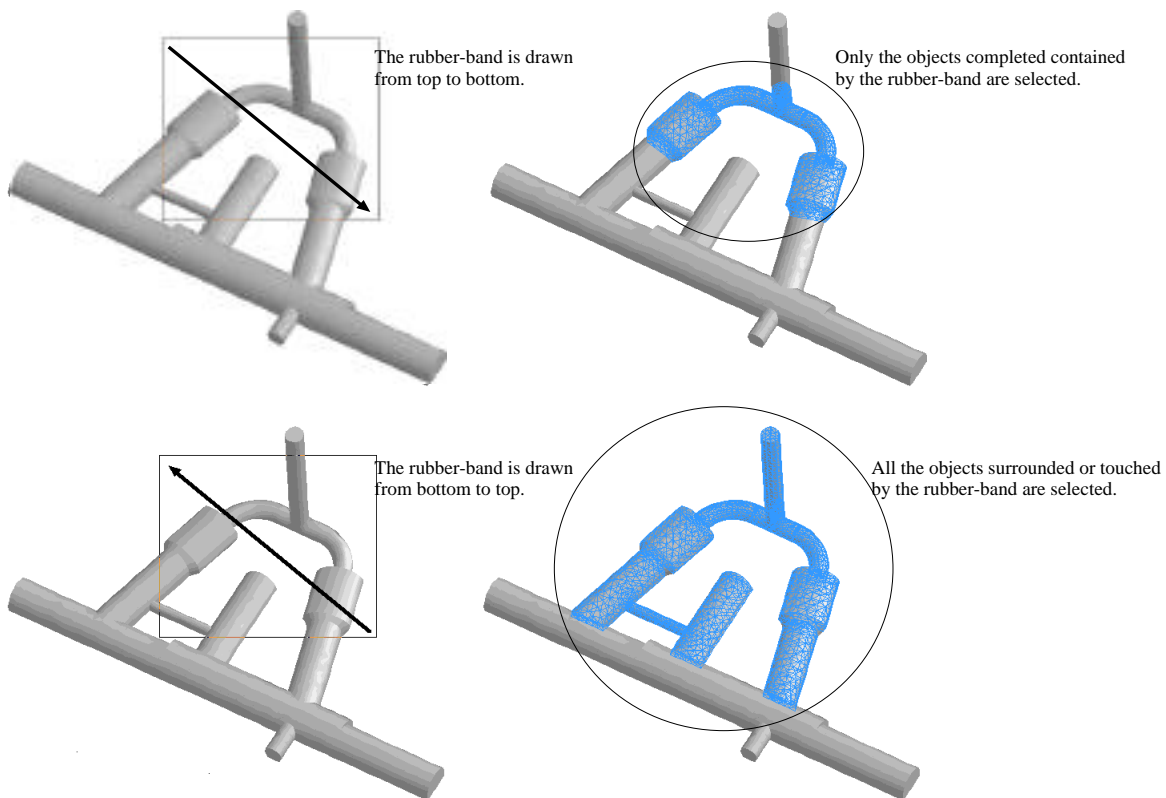
- 1) Position the screen cursor at one point.  
Imagine 4 corners of a rectangle on the screen. Place the cursor on one of the corners.
- 2) Press the mouse button and drag the cursor diagonally.  
You are now moving the cursor from one corner to the opposite corner. A rectangle is drawn. The rectangle 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.



## **Chapter 3**

### **Curves and Surface Primitives**

## *Chapter 3   Curves and Surface Primitives*

## Chapter 3 Curves and Surface Primitives

Curve is used here as a generic term designating various kinds of lines, from straight lines to parametric curves. Curves are used as the most basic objects defining the geometry of the finite element model. The following types of curves are supported in VisualFEA.

- straight line
- circular arc, circle
- elliptic arc, ellipse
- cubic spline curve, B-spline curve, Bezier curve, polynomial curve
- polyline
- rectangle

In VisualFEA, curves are used extensively for mesh generation, in various forms as follows:

- as seed curves generating nodal points
- as boundary curves enclosing the mesh
- as the axis of revolution or the path of sweeping for mesh generation
- as curve divisions controlling the mesh density

Surface primitive implies analytic or parametric surface defined as a constraint for mesh generation, and is termed here as such to distinguish it from surface mesh. The following types of curves are supported in VisualFEA.

- sphere
- cylinder, cone, truncated cone
- torus
- plane, B-spline surface, Bezier surface

The surface primitives may be used as a mapping space or bounding surface for mesh generation. They may also be used to create curves in 3-dimensional space by their intersection.

The curves and surface primitives are created interactively by inputting control points using either mouse or keyboard. Once created, curves or surface primitives can be modified later. General editing commands such as coping, duplicating, deleting are available for them. They can also be processed by various operations such as intersecting, linking, filleting and projecting.

Curves and surface primitives are meaningful as parts of a geometric model for finite element analysis. They are not directly involved in actual processing of finite element analysis, but are used as modeling objects defining nodes and elements.

Curve division is used for creation of line elements as well as for controlling the mesh density.

## Creating Curves and Surface Primitives


Curves and surface primitives are created by inputting a certain number of control points. The number of points that define a curve or a surface primitive is determined by the type and, in some cases, by the method of creation. A curve is constructed by inputting the control points in sequence. When the specified number of control points are entered, the curve is completed automatically, and the computer is ready for creating the next curve.

However, the number of control points are not specified in case of parametric curves such as spline or Bezier curves. In such cases, the curve is completed when termination of the input is informed by entering  key (Windows :  key) or clicking the last point twice.

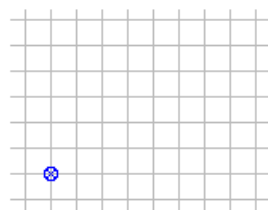
To create a type of curve or surface primitive, the corresponding tool button should be pressed. As long as the button is pressed, inputting of the control points can proceed without interruption. For example, a view angle or rendering mode can be changed without terminating the input procedure.

### Creating straight lines

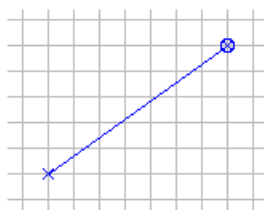


Press the line tool button  in order to create straight lines. The coordinates of the two end points defining a straight line can be entered either by clicking the grid points or by inserting the text of the coordinate values in the text box at the bottom of the tool palette. Creation of straight lines can be repeated as many times as desired, while the line tool button is in its pressed state.

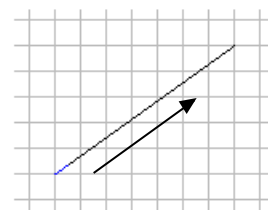
*A straight line has a direction. This direction is determined by which one of the two end points is first defined. That is, a straight line is directed from the first end point to the other. The direction of a line makes sense in such cases as dividing a line and assigning load conditions. However, these applications can be properly handled by the software regardless of the direction.*



Inputting the first point



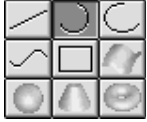
Inputting the 2nd point





Inputting the 3rd point

< Procedure of inputting a circular arc >

## Creating circles or circular arcs



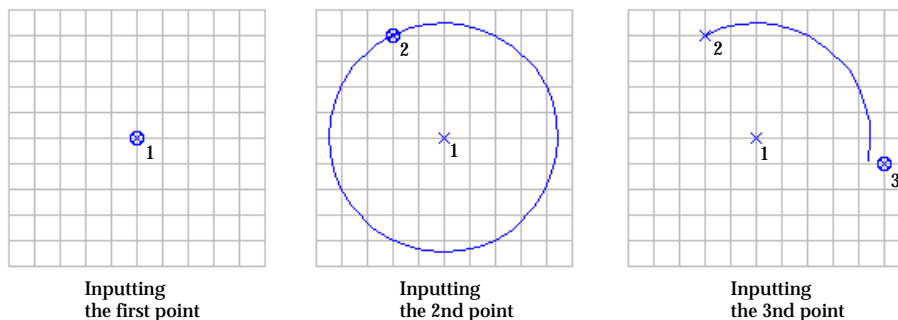
Press the circle tool button  in order to create circles and circular arcs. The **Circle** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to set whether to make circle or to make arc, and how to input. The first 5 items are used to create circular arcs and the others to create circles. The  marked option is effective for future creation of circles or circular arcs. The option can be changed by selecting the menu item before or while the arc or circle being created.

Circle	
<input checked="" type="checkbox"/> Arc	Arc with the central angle less than $180^\circ$ .
Clockwise Arc	Arc directed clockwise from the starting point to the end point.
Counter-CW Arc	Arc directed counter clockwise from the starting point to the end point.
Three Point Arc	Arc defined by three points on it.
Center & Angle Arc	Arc defined by the center, the starting point and the central angle.
Clockwise Circle	Circle directed clockwise from the starting point to the end point.
Counter-CW Circle	Circle directed counter clockwise from the starting point to the end point.
Three Point Circle	Circle defined by three points on it.
Center & Radius Circle	Circle defined by the center and the radius.

### ■ Arc

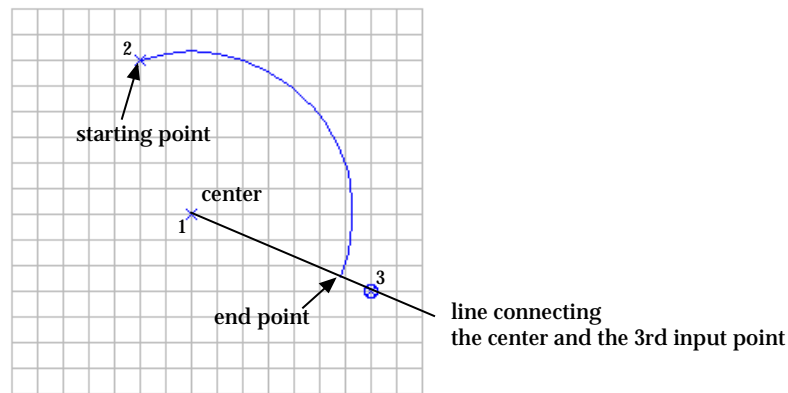
While this option is active, an arc is created by entering its center and two end points. The direction of the arc is determined so that its central angle become less than  $180^\circ$ .

The following figure shows an example of creating a circular arc by application of this option. When the center and the starting point of the arc are entered, a full circle is drawn. This circle represents the possible path of the arc, which is finally set by entering the third point, i.e., the end point of the arc.



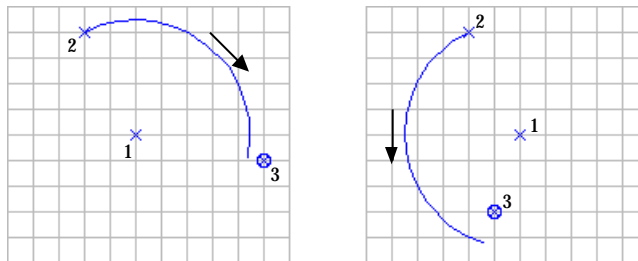
< Procedure of inputting a circular arc >

The end point may not coincide with the third input point as shown in this example. Then, the coordinates of the end point are determined so that the arc is trimmed by the line extending the center and the third input point.



&lt; Construction of an arc &gt;

As shown below, the central angle of a arc made by this option is always less than  $180^\circ$ , and accordingly, the arc is directed from the starting point to the end point.

< The arcs are made so that their central angles are less than  $180^\circ$  >

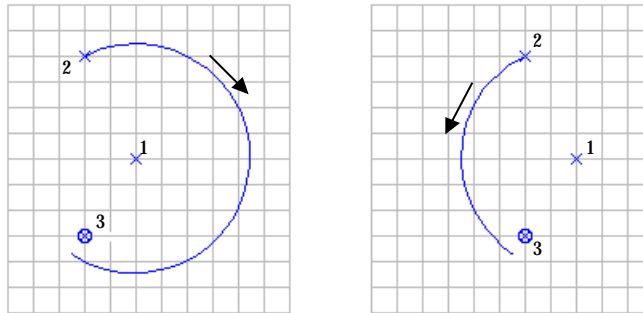
### ■ Clockwise Arc

While this option is active, a circular arc is created by entering its center and two end points. The arc is directed clockwise from the first end point to the second. The meaning of "clockwise" or "counter clockwise" is relative to the view direction in 3-dimensional space. That is, the direction is based on the current view of the screen. Therefore, a clockwise arc may look counter clockwise, when the screen view is reversed by rotating the view direction. However, the absolute direction of the arc in 3-dimensional space remains the same as created, regardless of the view transformation.

### ■ Counter-CW Arc

While this option is active, a circular arc is created by entering its center and two end points. The arc is directed counter-clockwise from the first end point to the second. Here, the direction of the arc is based on the current view of the screen, as in the case of "Clockwise arc".

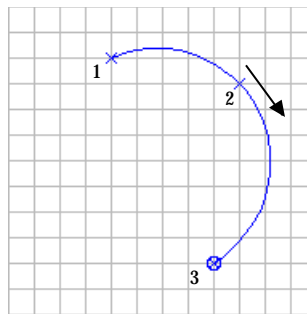




&lt; Clockwise arc and counter-CW arc &gt;

### ■ Three Point Arc

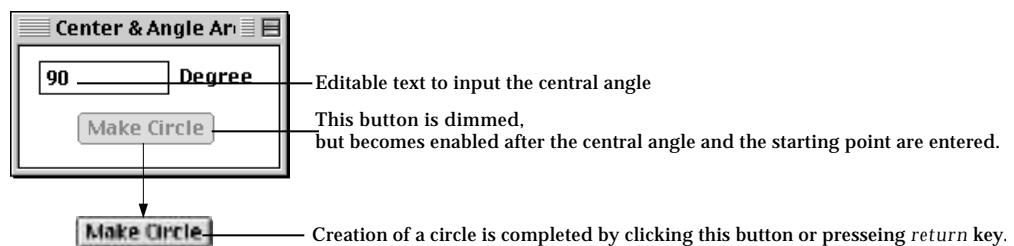
While this option is active, a circular arc is created by entering three points on the arc consecutively, i.e. the starting point, a mid-point, and the terminal point. The arc is directed from the starting point through the mid-point to the terminal point.



&lt; Three point arc &gt;

### ■ Center & Angle Arc

A dialog box appears when this option is selected. A circular arc is created by entering its center and the starting point, and specifying its central angle using the text box of the dialog. The central angle is set initially as 90° in the text box. The central angle can be modified as desired by editing this text. **Make Circle** is dimmed at the beginning, and is not in clickable state. Only after the center and the starting point of an arc are entered, this button becomes enabled. A circle is created by clicking this button, or pressing **return** key (Windows : **Enter** key).



### ■ Clockwise Circle

While this option is active, a circle is created by entering its center and two other points. The first of the two points specifies the starting point of its circumference. The circle is directed clockwise from this point and back to the point. The second point is a point on the plane of the circle, but not on the line connecting the first and the second point. The plane of the circle is determined by entering this point.

*The first two points are enough to define a circle on a plane, although one more point determining the plane of the circle is necessary in 3-dimensional modeling. However, VisualFEA asks for the 3rd point even in the case of planar modeling, because 3-dimensional data input is assumed regardless of the actual dimension of modeling.*

### ■ Counter-CW Circle

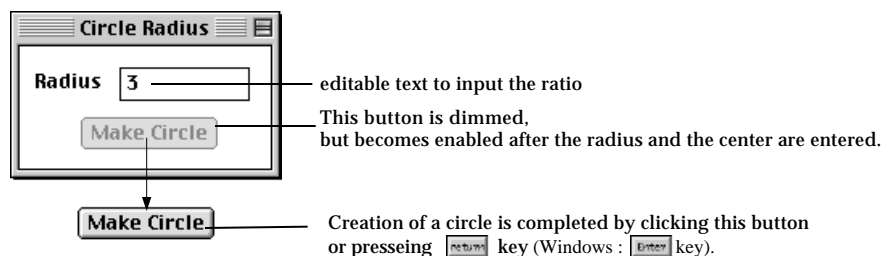
While this option is checked, a circle is created by entering three points in the same manner as in the case of "Clockwise Circle." The circle is directed counter-clockwise

### ■ Three Point Circle

While this option is checked, a circle is created by entering three points on the circle consecutively, i.e. the starting point and two other points on the circumference. The circle is directed from the starting point toward the second.



### ■ Center & Radius Circle

A dialog box appears when this option is selected. A circle is created by entering its center and specifying its radius using the text box of the dialog. At the beginning, the radius is set as 1 in the text box. This radius can be modified as desired by editing this text. **Make Circle** is dimmed at the beginning, and is not in clickable state. Only after the radius and the center of a circle are entered, this button becomes enabled. A circle is created by clicking this button, or pressing **return** key (Windows : **Enter** key).

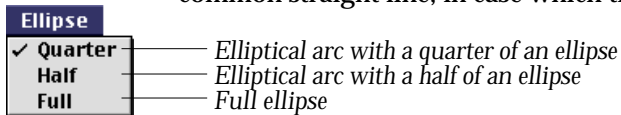


## Creating ellipses or elliptical arcs



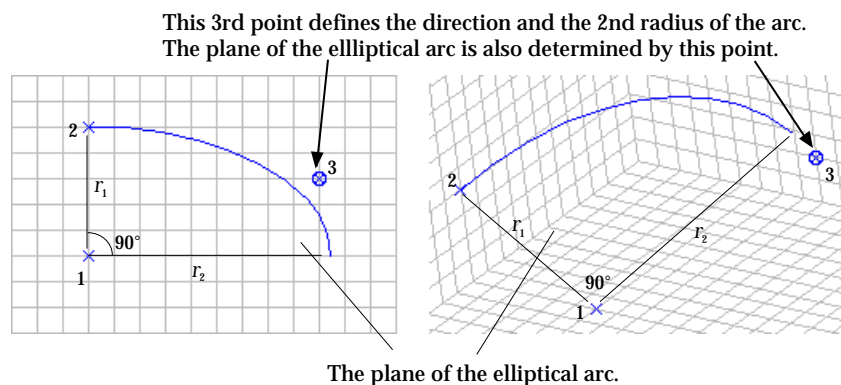
Press the ellipse tool button  in order to create ellipses or elliptical arcs. The **Ellipse** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to select the type of the ellipse, i.e., quarter, half or full ellipse. The  marked option is used for creation of ellipses and elliptical arcs. The option can be changed by selecting the menu item before or while the ellipse or elliptical arc being entered.

The ellipse or elliptical arc is completed by entering 3 points, regardless of the type of the ellipse. The first point is the center of the ellipse or elliptical arc, and the second point is the starting point of the circumference. The line connecting these two points becomes the first radius,  $r_1$  as shown below. The third point is not necessarily on the circumference of the ellipse or elliptical arc, but determines the second radius,  $r_2$  and the plane on which the ellipse or elliptical arc lies.  $r_2$  is set as equal to the distance between the first and the third points. The three points form the plane on which the ellipse or elliptical arc lies. But they should not lie on a common straight line, in case which the ellipse or elliptical arc cannot be defined.



### ■ Quarter

While this option is checked, an elliptical arc, with a quarter of an ellipse, is created by entering three points, as in the manner described above. The elliptical arc of this option has circumference with a central angle of  $90^\circ$  starting from the second input point and ending at the end of the second radius as shown below. When the second point is entered, a circle is drawn temporarily, with its center at the first point and with its radius connecting the first and the second points. The plane and the part of the ellipse consisting the arc are determined by the third point.

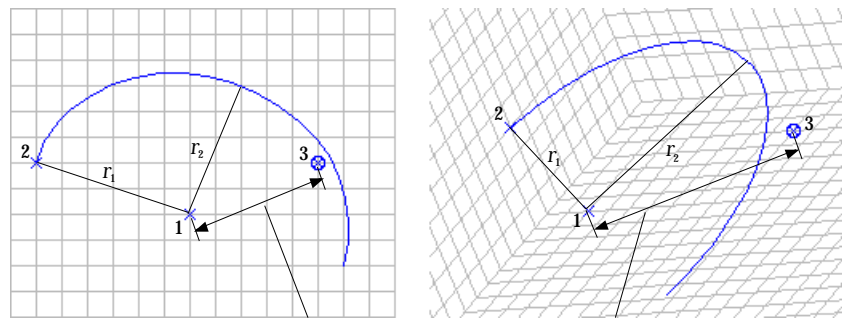


< Quarter ellipse >

### ■ Half

While this option is checked, an elliptical arc, with a half of an ellipse, is created by entering three points, in the manner described above. The half ellipse has the circumference starting from the second input point and is symmetric about the second radius as shown below.

When the second point is entered, a temporary circle is drawn, with its center at the first point and with its radius connecting the first and the second points. The plane and the part of the ellipse consisting the arc are determined by the third point.

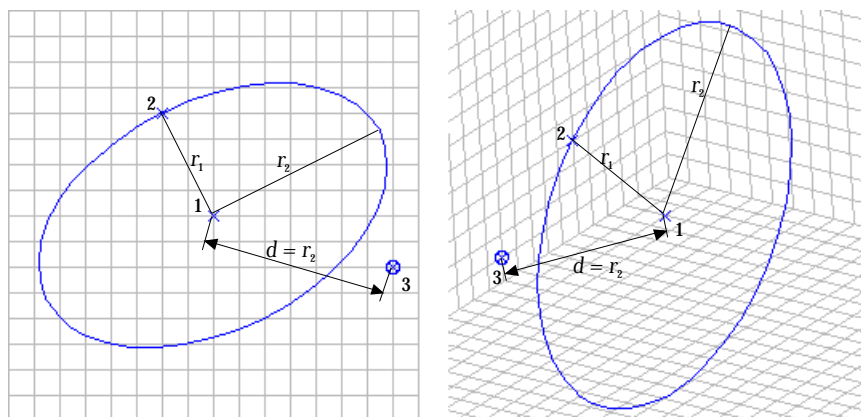


The length of the 2nd radius,  $r_2$  is equal to this distance.

< Half ellipse >

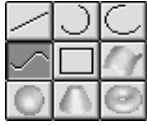
### ■ Full


While this option is checked, an ellipse is created by entering three points, as in the manner described above. The ellipse created by this option has circumference starting from the second input point and turning around back to this point. When the second point is entered, a circle is drawn, with its center at the first point and with its radius connecting the first and the second points. The plane and the second radius of the ellipse are determined by the third point.

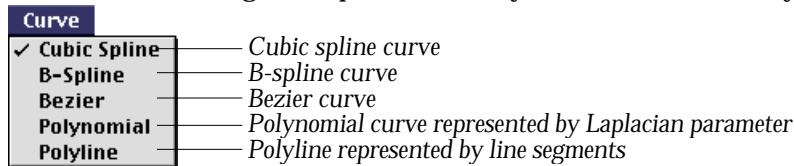


< Full ellipse >

## Creating parametric curves

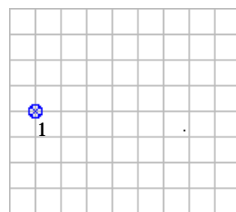


Press the curve tool button  in order to create parametric curves, which are defined by three or more control points. The **Curve** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to select the type of the curve, i.e., cubic spline, B-spline, Bezier, parabola and polyline. The marked option is used for creation of parametric curves. The option can be changed by selecting the menu item before or while control points being entered. The shape of the curve is formed as the control points are consecutively entered. There is no limit to the number of control points on a curve. In order to terminate entering the control points and complete a curve, click the last control point once again, or press **return** key (Windows : **Enter** key).

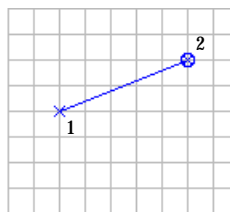


### ■ Cubic Spline

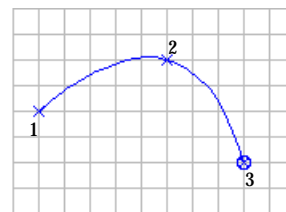
While this option is checked, a cubic spline curve is created by entering four or more points. When the first and the second point are entered, a straight line is drawn, connecting these two points. The third point makes a parabola. A cubic spline curve is formed after the fourth and the consecutive points are entered. The shape of the curve is continuously varied as the control points are added, and finalized only when the last control point is entered.



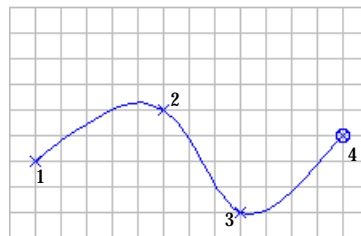
The first point is entered.



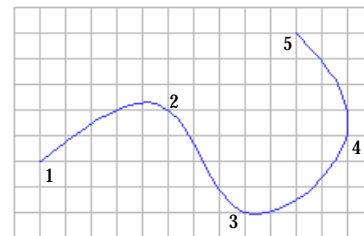
Straight line is formed by the 2nd point.



A parabola is formed by the 3rd point.



A cubic spline curve is formed by the 4th points.



A cubic spline curve is completed by clicking the last point or entering "return" key.

< Progress of creating cubic spline >

### ■ B-Spline

While this option is checked, a B-spline curve is created by entering three or more control points. When the first and the second point are entered, a straight line is drawn, connecting these two points. A B-spline curve is formed when the third and the consecutive points are entered. The process of creating B-spline curves is similar to that of creating cubic spline curves. A closed B-spline curve is obtained by overlapping the last control point over the first one.

### ■ Bezier

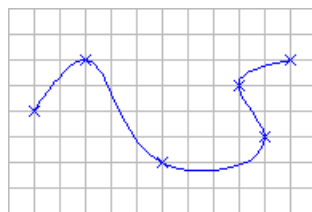
While this option is checked, a Bezier curve is created by entering three or more control points. A Bezier curve is formed when the third and the consecutive points are entered. The process of creating Bezier curves is similar to that of creating cubic spline curves.

### ■ Polynomial

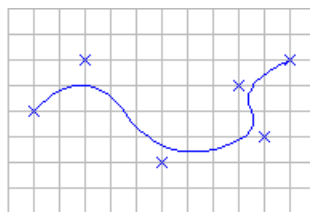
While this option is checked, a polynomial curve is created by entering three or more control points. A polynomial curve represented by Laplacian parameters is formed when the third and the consecutive points are entered.

### ■ Polyline

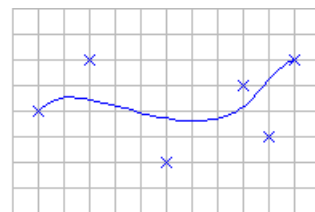
While this option is checked, a polyline is created by entering three or more control points. A polyline is represented by line segments connecting consecutive two points. A polygon is obtained by overlapping the last control point over the first one.



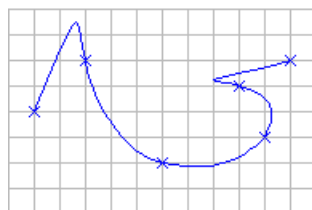
Cubic spline curve



B-spline curve



Bezier curve



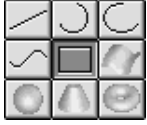
Polynomial curve




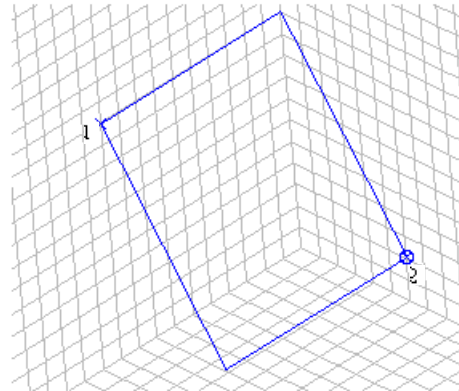
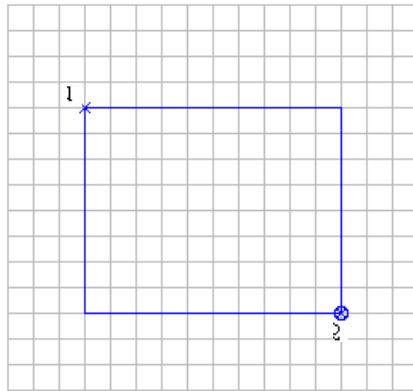
Polyline

< Parametric curves >

## Creating rectangles



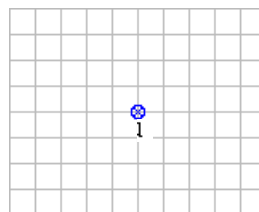
Press the rectangle tool button  in order to create rectangles, which are defined by entering two diagonal corner points of the rectangle. If these two points are on a grid plane, the rectangle is created on the same plane. For example, the first and the second points have the equal  $z$  coordinate value, the plane of the rectangle is perpendicular to the  $z$  axis. If the corner points do not form a common plane normal to any one of  $x$ ,  $y$  and  $z$  axis, the plane of rectangle is determined so that the shorter edge of the rectangle is made parallel to one of the coordinate axes. A rectangle is treated as one curve rather than four line segments.



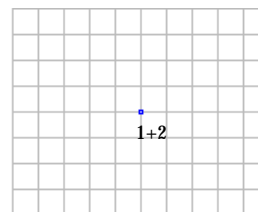
< Rectangle >

## Creating single points

A single point, which does not belong to a curve, can be created by entering identical coordinates twice. Using a grid, this can be achieved simply by entering a point and clicking the point once again. This procedure is valid only when the straight line tool is activated. A single point is useful as a reference point for inputting various curves. For example, when a circular arc is created, its center is not marked visually. So, it is sometimes difficult to overlay a control point of other curves over the identical point. In this case, the process can be facilitated by first inputting a single point at the center and using this as a reference.



Inputting the first point.







Creating a single point by clicking the first point again.

< Single point >


## Creating parametric surfaces

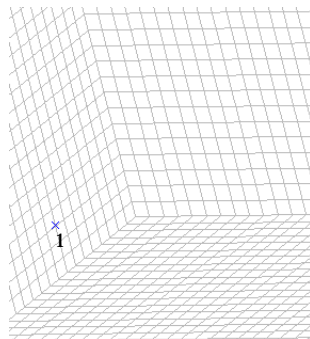


Press the surface tool button  in order to create parametric surfaces. The types of parametric surface include flat plane, B-spline surface, Bezier surface and Lagrangian surface. In order to set the type for creation, select one of the items from the **Surface** menu shown below. Currently effective type is marked by . The default type is initially set as a Bezier surface.

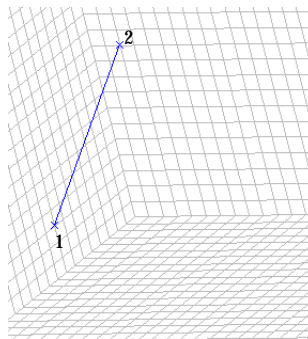
Surface	
Flat Plane	Create flat planes.
B-Spline Surface	Create B-spline surfaces.
 Bezier Surface	Create Bezier surfaces.
Lagrangian Surface	Create Lagrangian surfaces.
 Display Control Curve	Display the control curves of the surface.
Display Control Net	Display the control nets of the surface.
Display Hidden Removed	Display the control curves of the surface with hidden lines removed.

### ■ Flat Plane

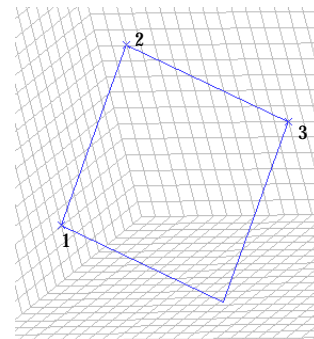
While this type is  marked, a flat plane is created by entering 3 points. As exemplified in the following figure, the flat plate is represented by a diamond shape. The first and the second input point form an edge of the diamond, and the second and the third point form another one.



Inputting 1st point










Inputting 2nd point



Inputting 3rd point

< Creating a flat plane >

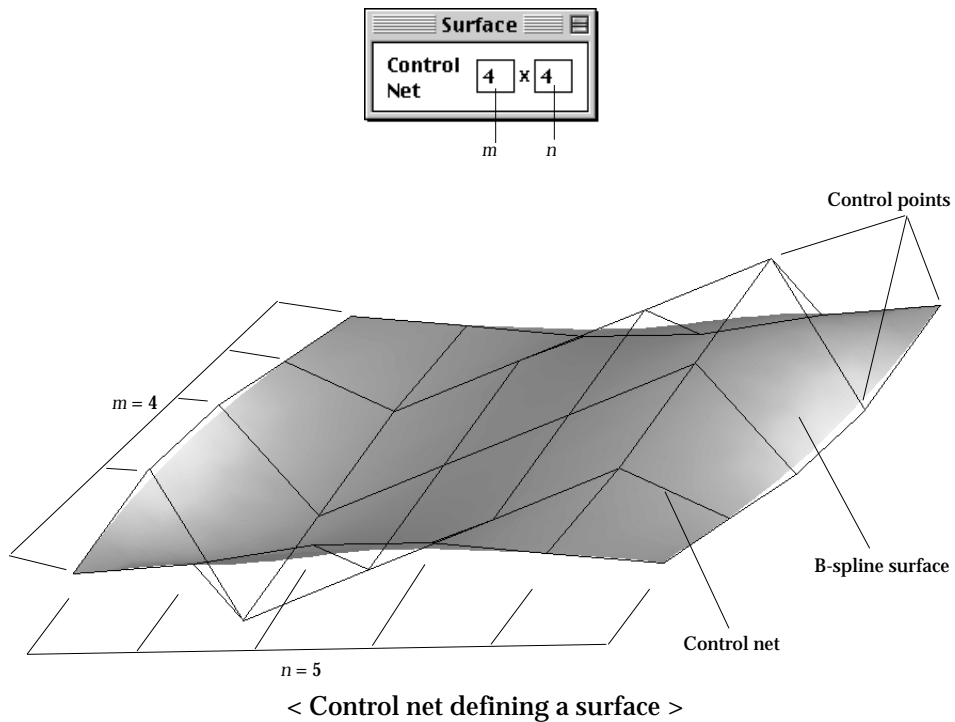
Modification mode is automatically activated right after a flat plane is created. Any of the three points can be selected and modified without sequence. Place the screen cursor on one of the points marked with , and press the mouse button. Then, the mark will be changed into . Keep the mouse button pressed while moving the cursor to the desired point. The  mark moves along with the cursor. Release the mouse button. Then, the selected point moves to the last point of  mark. The new coordinates of the selected point can also be entered directly using the keyboard. When one of the three points is selected and marked by , the coordinates of the point are displayed in the editable text boxes at the bottom of the tool palette. Edit the coordinate values and press  key (Windows :  key). Then, the point moves to a new location corresponding to the entered



coordinates.

### ■ B-spline Surface

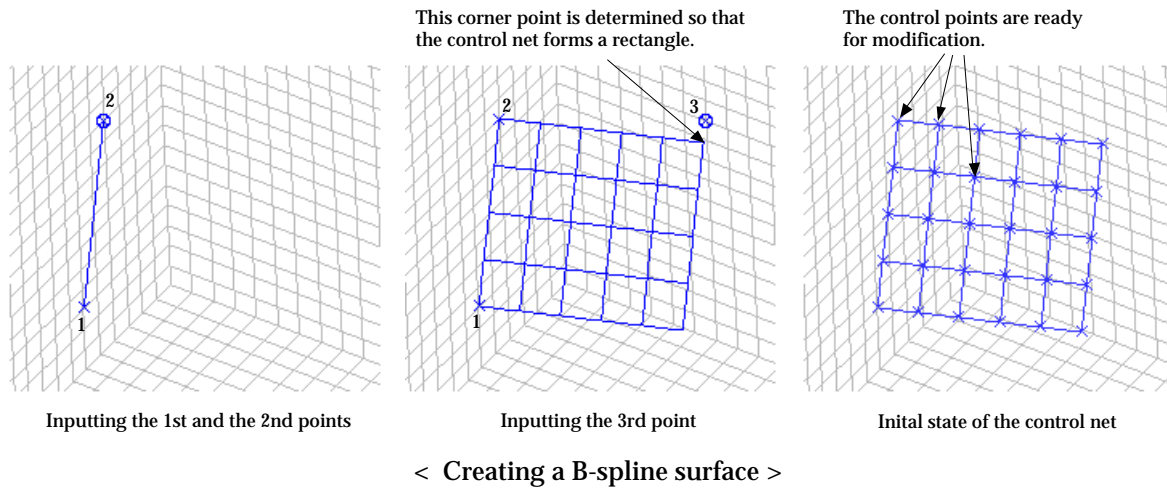
While this type is marked, a B-spline surface is created by specifying the size of the control net, entering 3 corner points and consecutively editing the coordinates of the control points. The size of the control net is defined in the form of  $m \times n$  and entered using "Surface" dialog. Here, the control net is a lattice of  $(m+1) \times (n+1)$  control points in array of  $(m+1)$  rows and  $(n+1)$  columns.




To create a B-spline surface,

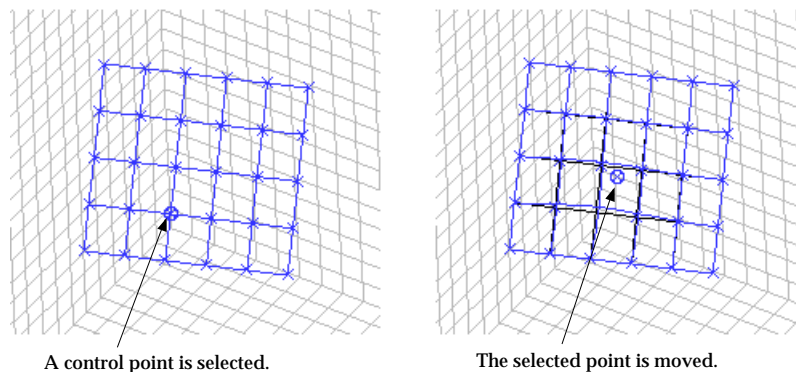
1. Enter  $m \times n$ , the size of the control net in the "Surface" dialog.
2. Input the first and the second point.  
They form a line connecting two corners of the control net. The line is also an edge of the control net, consisting of  $(m+1)$  control points.
3. Input the third point.  
The initial state of the control net is constructed by entering the third point. Another edge of the control net is aligned with the second and the third point. But, the corner of the control net may or may not coincide with the third point. The actual coordinates of the corner point are determined so that the control net forms a rectangle.

All the control points on the control net lie initially on a plane. In order to construct a desired surface, the control points should be moved, one by one, to the



desired positions as follows.

1. Select the control point to move, by clicking the point.  
The selected control point is highlighted by  mark.
2. Move the selected control point.  
The control point can be moved either by dragging the point on the screen, or by entering the new coordinates in the text boxes of the tool palette.
3. Repeat the step 2, and 3 for all the control points which should be modified.



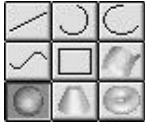
< Modifying the control net >


### ■ Bezier Surface

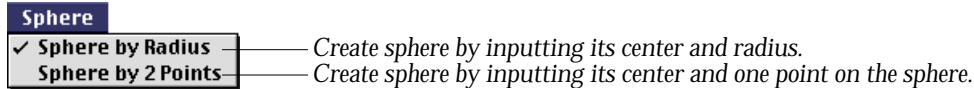
### ■ Lagrangian Surface

When these options are effective, Bezier surfaces and Lagrangian surfaces are respectively created by constructing and consecutively editing the control net of the surface in the same manner as that of B-spline surface.

## Creating spheres







Press the sphere tool button  in order to create spheres. A sphere can be defined either by inputting its center and radius, or by inputting its center and one point on the sphere. The **Sphere** menu shown below is provided with options to select one of the two methods.



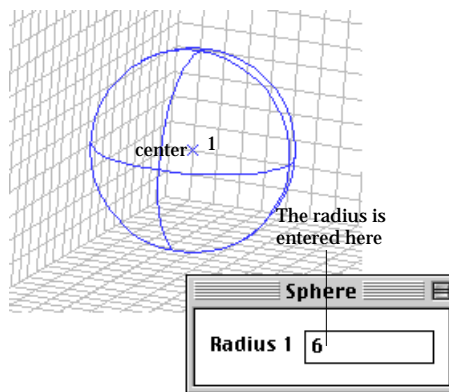
### ■ Sphere by Radius

"Sphere" dialog appears as shown above, when this option is selected. The radius of the sphere can be entered using this dialog. A sphere is created by inputting its center and the radius.

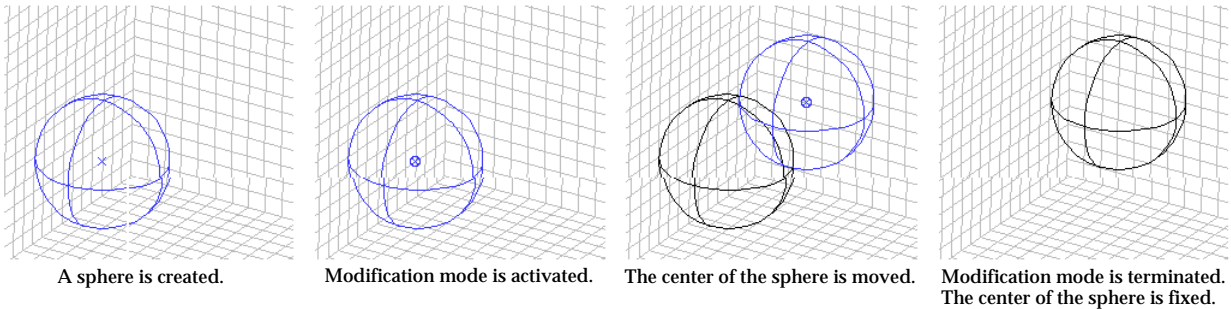
Modification mode is automatically activated right after a sphere is created. Its center and radius can be modified consecutively. Place the screen cursor on the center marked with , and press the mouse button. Then, the mark will be changed into . Keep the mouse button pressed while moving the cursor to the desired point. The  mark moves along with the cursor. Release the mouse button. Then, the center of the sphere moves to the last point of  mark. This operation on the newly created sphere can be repeated as many times as wanted. The modification mode is terminated and the center of the sphere is finalized by a light click at any point other than the sphere center.

The radius of the sphere can be modified by entering the radius in the "Sphere" dialog.

*Here "light click" implies touching mouse button slightly. "Hard click", implies pressing mouse button for a little while, say one second, and releasing it. Hard click will give a different result. Not only the movement of the center is terminated, but also a new sphere with its center at the point of the click will be created.*



< Creating a sphere by its center and radius >



< Moving the center of the newly created sphere >

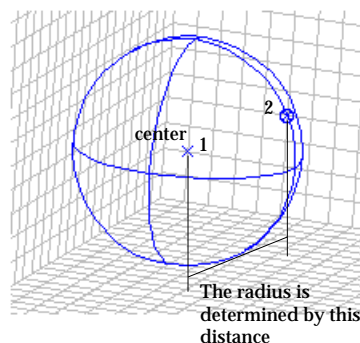
### ■ Sphere by 2 Points

While this option is effective, a sphere is created by inputting 2 points, i.e., its center and one point on the sphere. "Sphere" dialog is not shown. The first input point becomes the center of the sphere. And the second point defines the surface of the sphere. The radius of the sphere is determined by these 2 points.

Dragging the mouse with its button pressed while entering the second point will visualize the changing size of the new sphere. The second point moves along with the cursor, and accordingly the radius of the sphere changes. The creation of the sphere is completed at the moment when the mouse button is released.

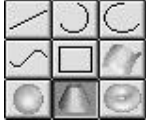
Modification mode is automatically activated right after a sphere is created. And, its center and the second point can be modified consecutively in the same way as explained for "Sphere by radius". While moving one of the two points, the other point stays unchanged, and accordingly the radius of the sphere changes.


In order to move the whole sphere without changing its radius, press the "shift" key while doing the above operations.



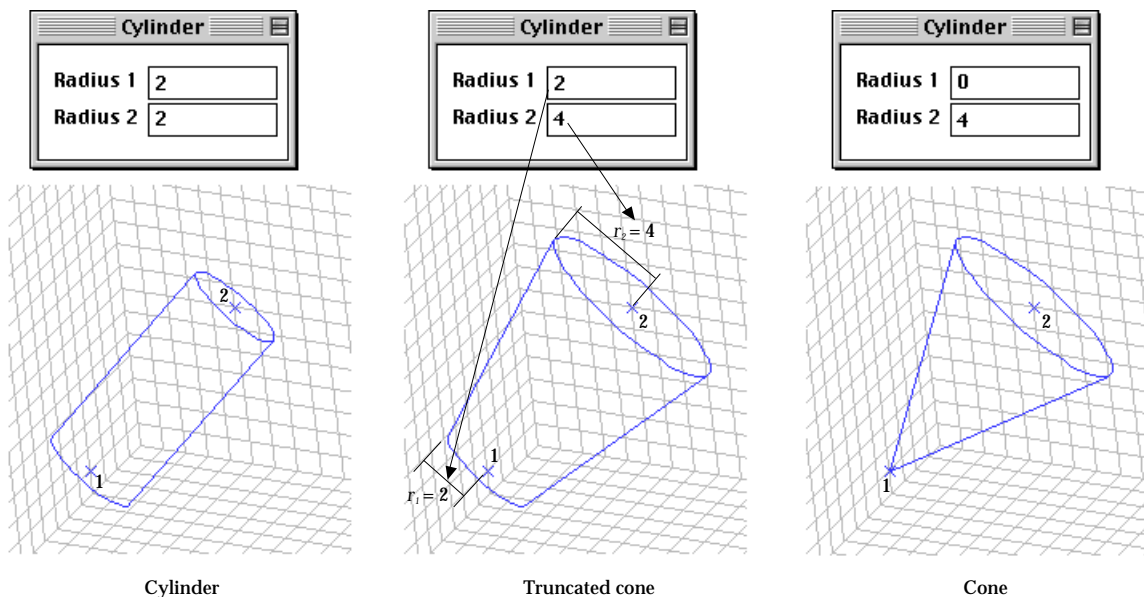
< Inputting a sphere by its center and one point on the sphere >

### Creating cylinders, cones or truncated cones



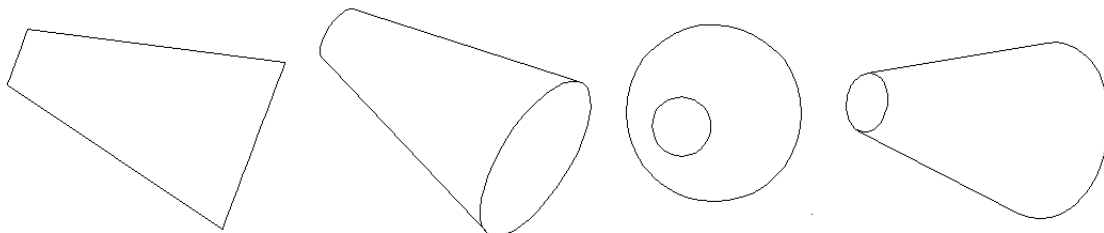
Press the cylinder tool button  in order to create cylinders, cones or truncated cones, which are treated as the same kind of primitive surfaces in VisualFEA. These primitive surfaces are defined by two end points which form an axis, and two radii. A cylinder is created when the two radii are the same. As shown in the figures below, a truncated cone is made when the two radii are different, and a cone is obtained when one of them is zero. The two radii are entered using the "Cylinder" dialog. The texts entered for "Radius 1" and "Radius 2" define the radii at both ends of the surfaces. The texts can be entered before, after or during inputting the end points.

Modification mode is automatically activated right after a sphere is created. And, its center and the second point can be modified consecutively in the same way as explained for "Sphere by radius"



#### < Inputting a cylinder, a cone or a truncated cone >


The surface is represented by outlines. Only the visible part of the outlines is displayed so that the screen drawing is simplified. The visible part changes depending on the view direction as shown below.



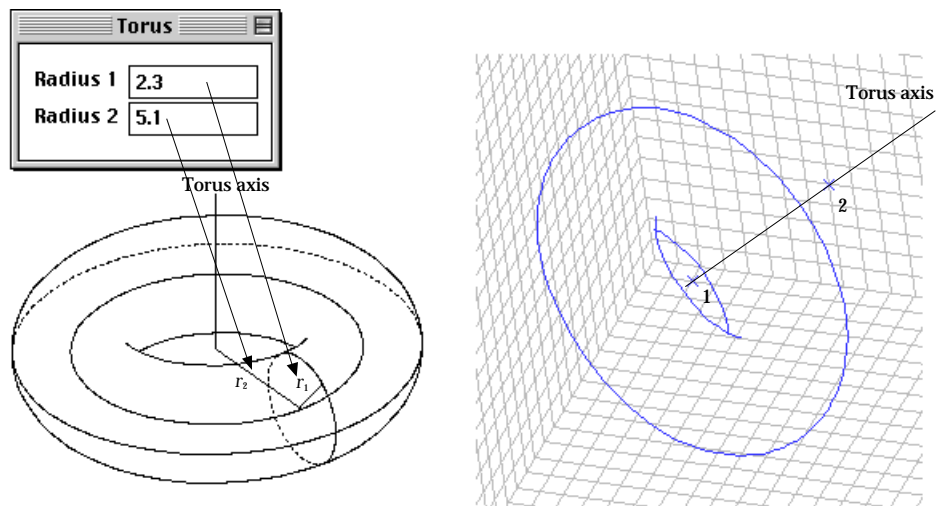
#### < Different display of a truncated cone depending on the view direction >

## Creating tori



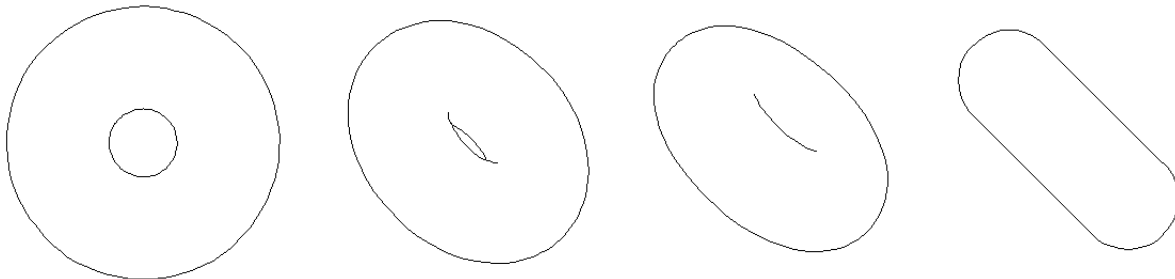
Press the torus tool button  in order to create tori, each of which has an axis and two radii. The axis is defined by two points. The texts entered for "Radius 1" and "Radius 2" define respectively the radius  $r_1$  of the small circle and  $r_2$  of the large one as indicated in the figure below. The texts can be entered before, after or during inputting the axis points.

Modification mode is automatically activated right after a torus is created. And, its center and the second point can be modified consecutively in the same way as explained for "Sphere by radius."



< The axis and two radii of a torus >

The surface is represented by outlines. Only the visible part of the outlines is displayed so that the screen drawing is simplified. The visible part changes depending on the view direction as shown below.



< Different display of a torus depending on the view direction >

## Handling Curves and Surface Primitives

Once created, curves and surface primitives are handled as modeling objects. They can be edited, or processed by various operations.

### General editing commands

General editing commands such as coping, duplicating, deleting are applicable for curves and surface primitives.

#### ■ Deleting curves and surface primitives

In order to delete curves or surface primitives, first select the objects to delete, and select "Clear" item from **Edit** menu, or press *delete* key. When a curve or a surface primitive is deleted, the control points on it are also deleted. But, the control points shared by other curves or surface primitives are not deleted. Likewise, all the nodes allocated on the deleted curve are removed, but nodes shared by other curves or meshes cannot be deleted. In this context, a meshed curve cannot be deleted. In order to delete a meshed curve, all the meshes associated with the curve should be deleted in advance.

#### ■ Copying curves and surface primitives

In order to copy curves or surface primitives, select the objects to copy, and select "Copy" item from **Edit** menu. The copied objects are stored in the computer memory and can be pasted later.

#### ■ Cutting curves and surface primitives

Cutting deletes the selected objects, and at the same time, stores the deleted objects in the computer memory. This is done by "Cut" command in **Edit** menu, and has the same effect as deleting after copying the selected objects.

#### ■ Pasting curves and surface primitives

"Paste" command in **Edit** menu pastes the objects which have been stored in the computer memory by "Cut" or "Copy" command.

### Reshaping and moving





It is sometimes necessary to modify curves or surface primitives, which were previously created. Modification mode should be activated in order to start modification. Under modification mode, control points can be moved to other positions, and some attributes may be altered using dialog boxes. But adding or

deleting control points are not allowed.

You may reshape a curve or a surface primitive by modifying its control points individually. Or, you may move an entire curve or an entire primitive by a single operation.

*A divided curve has nodes on it. When a curve is modified, its nodes are also moved on to the new path of the curve. Meshes are generated on the basis of curves and surface primitives. Any curve or primitive surface associated with meshes cannot be modified.*





### ■ Activating modification mode

Modification mode is activated by modification tools. Click the curve modification tool button  to modify curves and the surface modification tool button  to modify the surface primitives. The modification mode is deactivated when any other tool is activated.

*Modification mode is also activated automatically right after creating a surface primitive. Modifying surface primitives under this situation is described in each section of creating primitive surfaces.*

### ■ Moving control points by mouse



A control point of a curve or a surface primitive can be dragged to another position by using mouse as follows:

- 1) Select a curve or a surface primitive to modify.  
The selected curve or primitive is highlighted in blue color, and all the control points of the curve or primitive are marked by .
- 2) Click the control point to move.  
The clicked point gets  mark. The coordinates of the point are displayed on the text boxes at the bottom of the tool palette.
- 3) Place the screen cursor over the  mark, and press the mouse button.  
Now the point is ready for movement.
- 4) Drag the point to the desired position, and release the mouse button.  
As you move the mouse, the  marked point follows the cursor movement. The point is dragged to the position where the mouse button released. Accordingly, the shape of the curve or primitive is modified.
- 5) Repeat 2), 3) and 5) for all the control points which should be modified.  
While this modification process is going on, the original shape as well as the modified shape is displayed. The original shape is represented in black and the modified one in blue.
- 6) Complete the modification by clicking any point not on the curve or primitive.  
When modification is completed, the original shape disappears, and the modified shape settles with moved control points. The color of the modified shape turns into black. Now, another curve or surface primitive can be modified by starting from step 1 again.



### ■ Moving control points by keyboard input


The control point can be moved also by entering coordinates using the keyboard as follows:

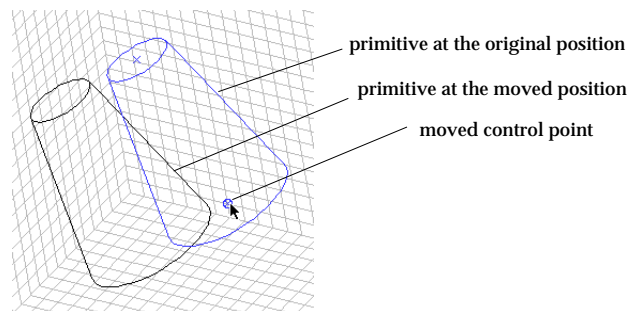
- 1) Select a curve or a surface primitive to modify.
- 2) Click the control point to move.
- 3) Edit the coordinates of the selected point in the text boxes in the tool palette.  
The coordinates of the selected point are displayed in the editable text boxes. Select and edit the X, Y and Z coordinates as desired.
- 4) Press  key (Windows :  key).  
The modified coordinates are entered, and the selected point is moved to the new point.
- 5) Complete the modification by clicking any point not on the curve or primitive.  
Now, another curve or surface primitive can be modified by starting from step 1 again.

Mouse and keyboard may be used alternately while modifying a curve or a surface primitive.

### ■ Moving an entire curve or surface primitive

You may move an entire curve or an entire primitive by specifying the movement of a point on it.

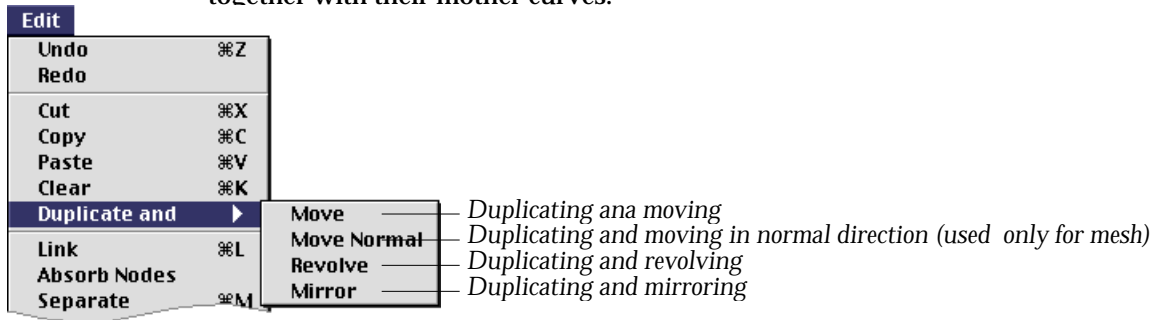
- 1) Select a curve or a surface primitive to modify.
- 2) Click the control point to move.
- 3) Press *shift* key.
- 4) Place the cursor over the  marked point, and press the mouse button.
- 5) Drag the point to the desired position, and release the mouse button.  
The entire curve or primitive moves along with the movement of the selected point.
- 6) Complete the modification by clicking any point not on the curve or primitive.  
The object settles at the moved position. Another curve or surface primitive can be modified by starting from step 1 again.



< Movement of an entire surface primitive >

## Duplicating curves and surface primitives

It is sometimes convenient to create objects by duplicating the selected objects. The final position or orientation of the duplicated objects may be set by moving, revolving or mirroring. The nodes on divided curves are automatically duplicated together with their mother curves.

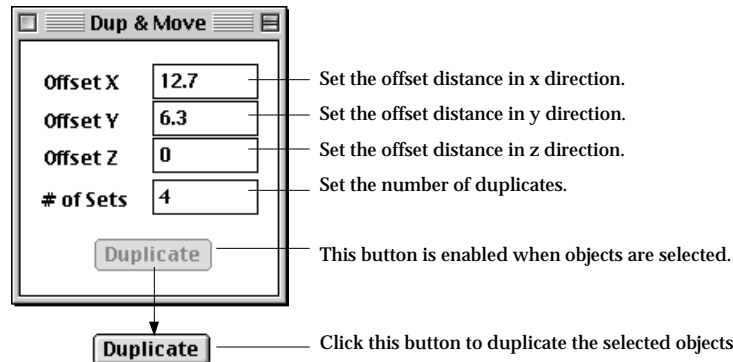


### ■ Duplicating curves and surface primitives, and moving

You may make as many duplicates of the selected curves or surface primitives as you want, and position them with specified offset distance by the following procedures.

- 1) Choose "Move" from **Duplicate and** submenu.

One of the selection tools is activated if not currently, and "Dup and Move" dialog box appears.

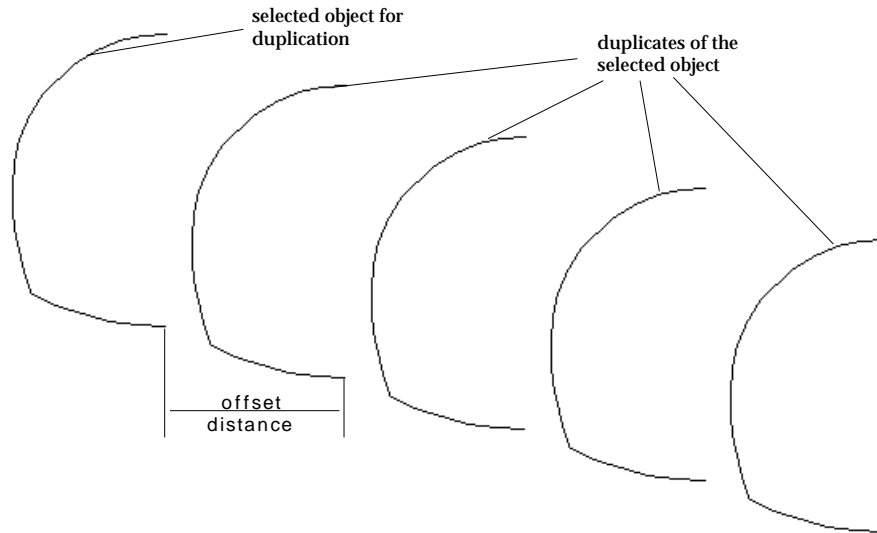


- 2) Set the offset distances in x, y, z directions.  
The duplicates are spread in parallel with the specified offset distances. If all the offset distances are set to 0, no duplication will be done.
- 3) Set the number of duplicates to make.  
The number of duplications is set by entering in this text box. The number should be greater than or equal to one.
- 4) Select the curves or surface primitives to duplicate.  
Click the selection tool corresponding to the type of objects to duplicates, and select the objects. The initially dimmed **Duplicate** button is enabled when

one or more objects are selected.

- 5) Click **Duplicate** button.

Duplication is fulfilled, and duplicates of the selected objects are created.



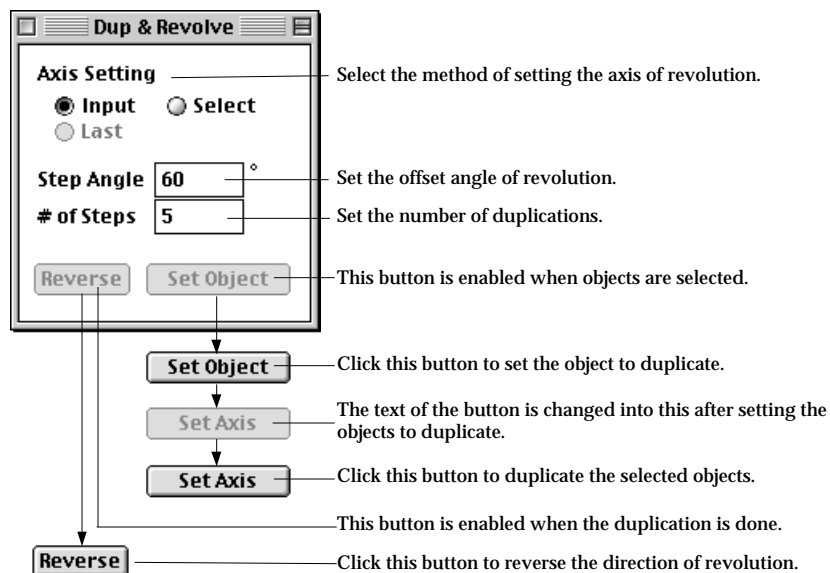
<Duplicating the selected curves and moving >

### ■ Duplicating curves and surface primitives, and revolving

You may set the position and the orientation of the duplicates by revolving them about a specified axis by the following procedures.

- 1) Choose "Revolve" from **Duplicate and** submenu.

One of the selection tools is activated if not currently, and "Dup and Revolve" dialog box appears.



- 2) Select the method of setting the axis of revolution.

The axis of revolution for duplication may be set by one of the following 3 methods.

- **Input** : input the axis of revolution by entering a new straight line just as creating a new line.
- **Select** : select an existing straight line.
- **Last** : apply the axis of revolution which was applied in the last duplication. The “Last” button is enabled only when “Duplicate and Revolve” is processed at least once.

- 3) Set the offset angle of revolution for duplication.

Each set of the duplicates are revolved with the specified angle. This value may be either positive or negative.

- 4) Set the number of duplicates to make.

The number of duplications is set by entering in this text box. The number should be greater than or equal to one.

- 5) Select the curves or surface primitives to duplicate.

Click the selection tool corresponding to the type of objects to duplicates, and select the objects. The initially dimmed **Set Object** button is enabled when one or more objects are selected.

- 6) Click **Set Object** button.

The selected objects are set for duplication. The text of the button is changed into **Set Axis**. If “Axis Setting” is set to “Last”, the button is enabled. Otherwise, it is disabled.

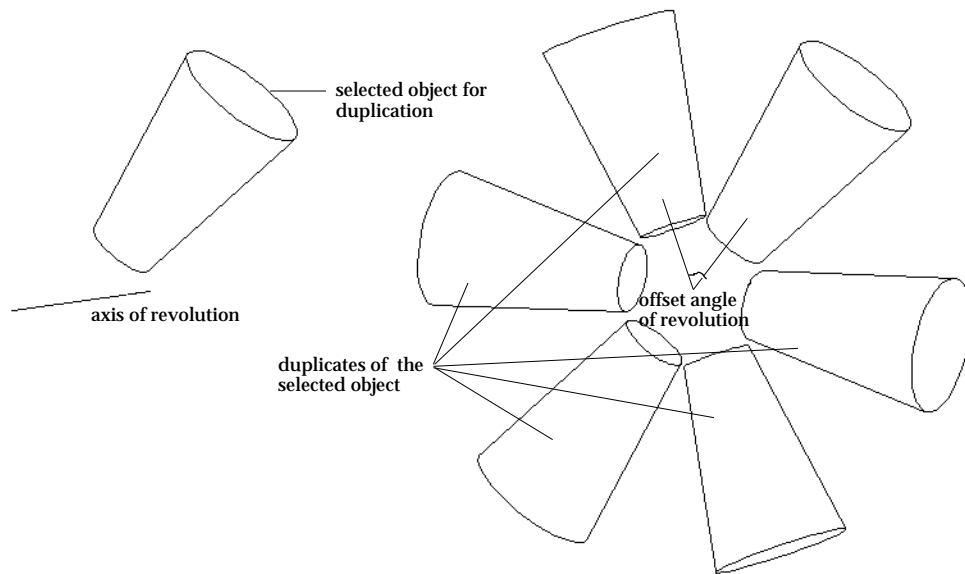
- 7) Input or select the axis of revolution.

Input or select the axis of revolution depending on the “Axis Setting.” Then, **Set Axis** button is enabled. If “Last” button is on, skip this step.

- 8) Click **Set Axis** button.

The duplication is processed, and the specified number of duplicates are created. The text of the button is changed into **Set Object** which indicates it is ready again for another duplication process.

When a duplication process is completed, **Reverse** button is also enabled. The orientation of revolution can be reversed by clicking **Reverse** button.



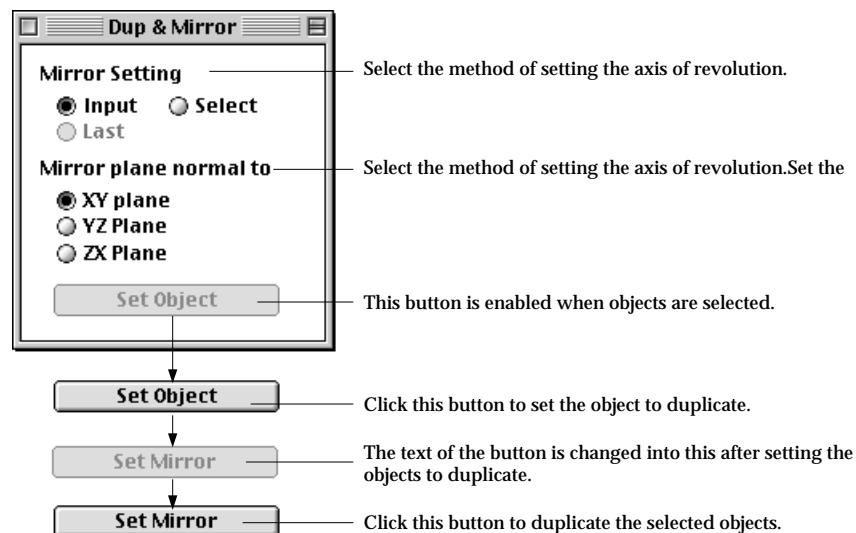
&lt; Duplicating a surface primitive and revolving &gt;

### ■ Mirroring curves and surface primitives

Duplicate of selected objects can be created by mirroring them.

- 1) Choose "Mirror" from **Duplicate and** submenu.

One of the selection tools is activated if not currently, and "Dup and Mirror" dialog box appears.



- 2) Select the method of setting the mirror.

The mirror plane for duplication may be set by one of the following 3 methods.

- Input : input the mirror plane by entering a new straight line just as

creating a new line.

- Select : select an existing straight line.
- Last : apply the mirror plane which was applied in the last duplication. The “Last” button is enabled only when “Duplicate and Mirror” is processed at least once.

3) Select the plane normal to the mirror

One of xy, yz and zx coordinate planes should be selected. The mirror plane is determined by this plane and the line as described in step 6). The mirror is normal to this plane and contains the line.

4) Select the curves or surface primitives to duplicate.

Click the selection tool corresponding to the type of objects to duplicates, and select the objects. The initially dimmed  button is enabled when one or more objects are selected.

5) Click  button.

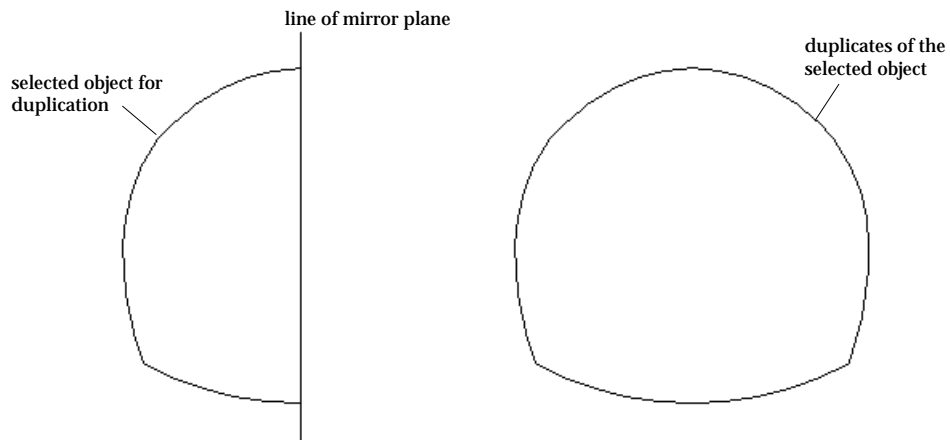
The selected objects are set for duplication. The text of the button is changed into . If “Mirror Setting” is set to “Last”, the button is enabled. Otherwise, it is disabled.

6) Input or select a straight line, which defines the mirror plane normal to the plane selected in step 3).

Input or select a straight line depending on the “Mirror Setting.” The mirror plane is defined so that the line is on the plane. Then,  button is enabled. If “Last” button is on, skip this step.

7) Click  button.

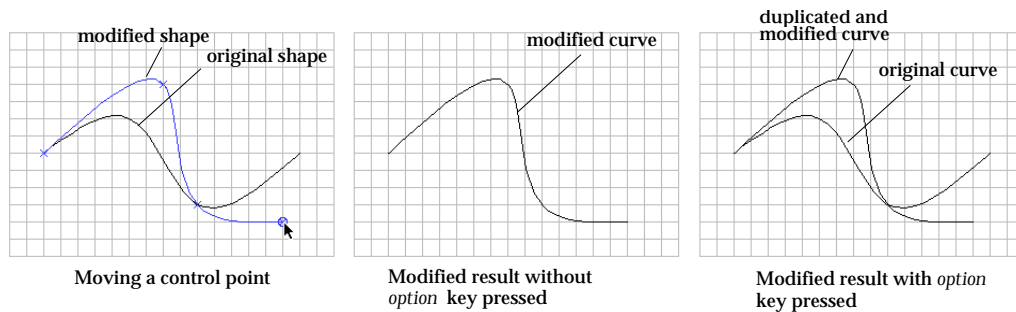
The duplication is processed, and the text of the button is changed into  which indicates it is ready again for another duplication process.



< Mirroring a curve >

### ■ Modifying a curve or a primitive with its duplicate

Instead of modifying the selected curve or a primitive, you may modify its duplicate. Press *option* key and follow the steps described in "Moving control points by mouse." A duplicate of the curve or primitive is created prior to modification process. The original curve or primitive is intact, and its duplicate is modified.



< Modifying a curve or its duplicate >


### ■ Modifying attributes of a primitive surface

There are a few kinds of surface primitives whose attributes may be modified. Namely, they are spheres, cylinders (including cones and truncated cones) and tori. If you click a surface primitive of these types under modification mode, a dialog box appears on the screen. The dialog box has editable text item(s) for editing attributes. The shape of the surface primitive is immediately modified by editing these text items.

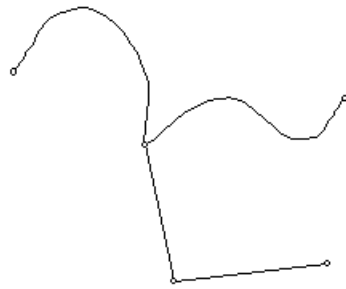
## Processing curves and surface primitives

### ■ Linking curves

Linking is connecting more than two curves serially and forming a linked curve. A linked curve is treated as a single curve in various actions such as selecting, deleting, cutting, copying, intersecting, projecting, linking and so on. The following steps are taken to link a curve:

- 1) Activate curve selection tool , if it is not in action.
- 2) Select curves to link.
- 3) Choose "Link" from **Edit** menu.

Linking has nothing to do with the order of selecting curves. All the selected curves should be serially connectable. Otherwise, they cannot be linked. In such cases as exemplified in the following figure, a warning message "Incompatible Link" is issued, and the "Link" command is ignored.



The selected curves are not connected serially.



The selected curves are not connected.


< Example of curves that cannot be linked into a linked curve >

### ■ Separating curves

Linked curves can be separated into individual curves of original state prior to linking. In order to separate linked curves, select them, and choose "Separate" from **Edit** menu. More than one linked curve can be separated at once.

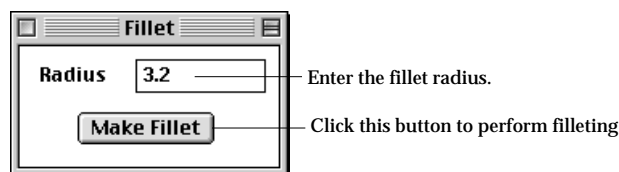
### ■ Filletting two straight lines

Filletting is used to connect two straight lines with an arc of a specified radius.

- 1) Activate curve selection tool , if it is not in action.
- 2) Select two straight lines for filletting.  
"Fillet" item of **Edit** menu is enabled only when 2 straight lines are selected.
- 3) Choose "Fillet" from **Edit** menu.  
"Fillet" dialog appears on the screen.
- 4) Enter the fillet radius using the "Fillet" dialog.
- 5) Click **Make Fillet** button of the "Fillet" dialog.

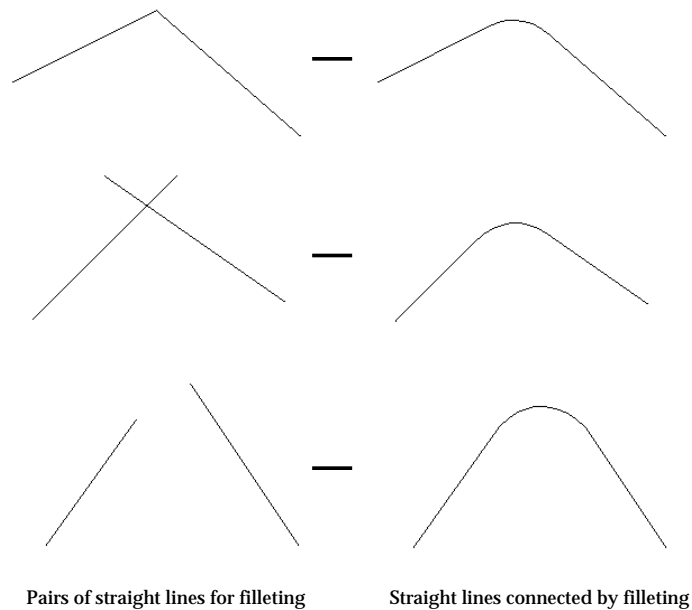
In order to continue filletting with other straight lines, repeat 2), 3) and 5) instead of starting from step 1). Filletting can be performed as long as "Fillet" dialog is on the screen. Filletting action can be terminated by closing the dialog or by starting any other tool in the tool palette.

Filletting is performed if and only if two straight lines are selected. The two lines do not have to touch in order to perform filletting, but must be capable of intersecting.



< Fillet dialog >






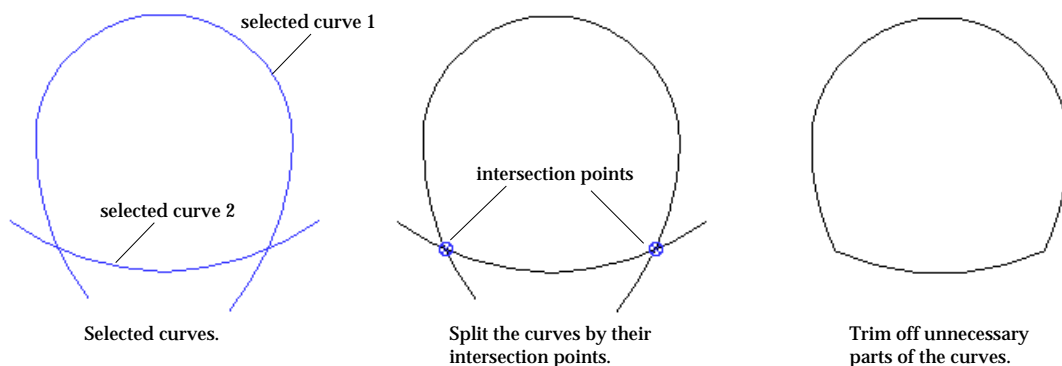
< Examples of filleting >

### ■ Splitting curves by their intersection points

If there are two or more curves crossing with each other, then there are intersection points. You may need split each of them at their intersection points. This can be achieved by the following steps:

- 1) Activate curve selection tool , if it is not in action.
- 2) Select curves for intersection.
- 3) Choose "Intersect" from **Edit** menu.


In case there are one or more divided curves among the selected curves, they are involved in determining the intersection points, but will not be split. Every segment split from a curve forms an independent curve.



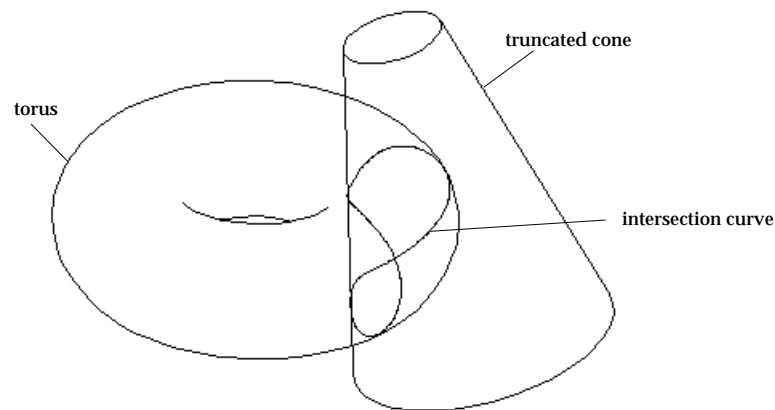
Example of processing curves using their intersection points

### ■ Obtaining intersection curves between surface primitives

If there are two or more surface primitives crossing with each other, then there are intersection curves.

- 1) Activate surface primitive selection tool  if it is not in action.
- 2) Select surface primitives for intersection.
- 3) Choose "Intersect" from **Edit** menu.

A surface primitive can not be split by their intersection curves. But, intersection curves are useful in setting the boundary of mesh region on surface primitives.

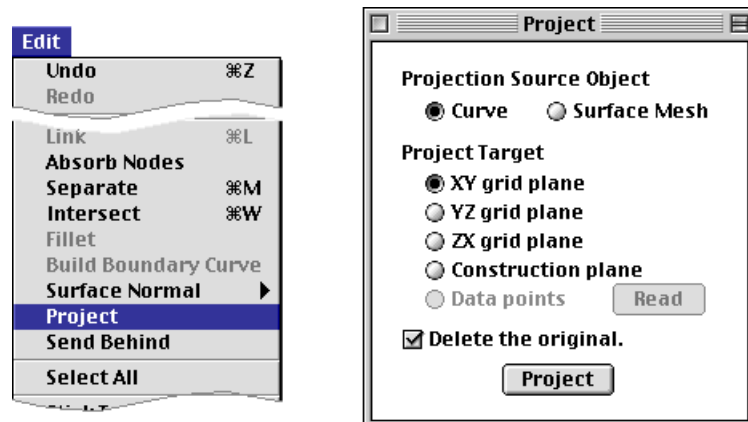



< Example of intersecting surface primitives >

### ■ Projecting curves

New curves may be created by projecting other curves on the specified grid plane.

- 1) Choose "Project" item from **Edit** menu.  
"Project" dialog opens. Curves or surface meshes can be projected to the designated target using this dialog.



- 2) Turn on the radio button "Curve" under "Projection Source Object."  
The curve selection tool  is automatically activated.

- 3) Set the projection target.

There are radio buttons "XY grid plane", "YZ grid plane", "ZX grid plane" and "Construction plane" which represent projection targets. Select one of them.

- 4) Select the curves for projection.

When one or more curves are selected, **Project** button is enabled. The selected curves are the mother curves for projection.

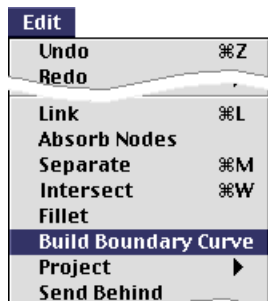
- 5) Click **Project** button.


The the selected curves are projected on the target object.

Nodes as well as control points on the new curves are also formed by projection of corresponding nodes and control points on the mother curves.

### ■ Building boundary curves of a surface primitive

A closed surface such as a sphere or a torus has no boundary curves. But, such an open surface as a cylinder or a cone has boundary curves. These boundary curves may be used in defining the surface region for mesh generation, and should be built for use.



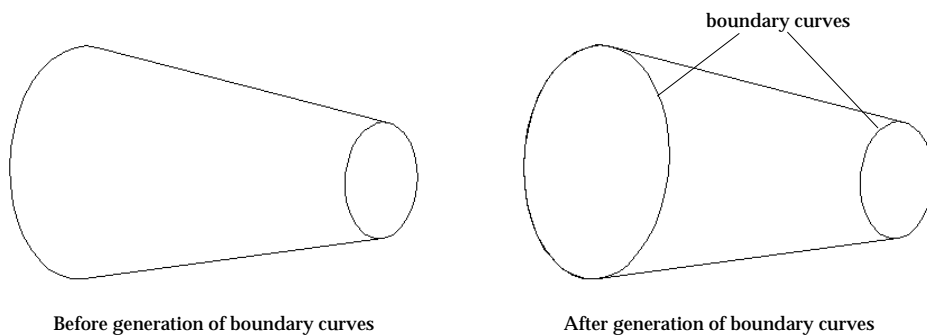
- 1) Activate surface primitive selection tool  if it is not in action.
- 2) Select surface primitives, the boundary curves of which are to be generated.

When one or more surface primitives are selected, "Build Boundary Curve" item is enabled.

- 3) Choose "Build Boundary Curve" from **Edit** menu.

This command is effective only for open surface primitive. Boundary curves are created on all the effective surface primitives.

Once the boundary curves are created, they are independent objects which do not belong to the mother primitive surfaces any longer.

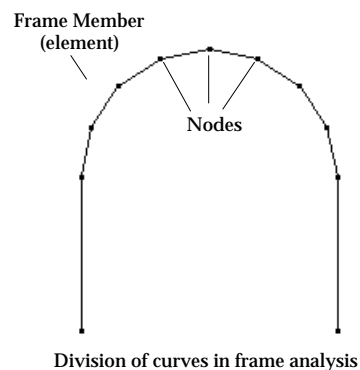
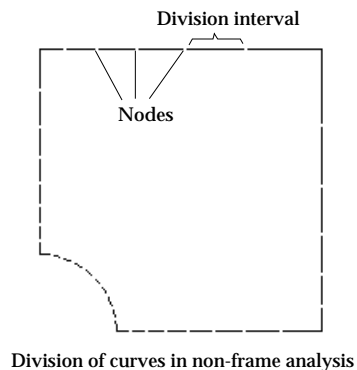
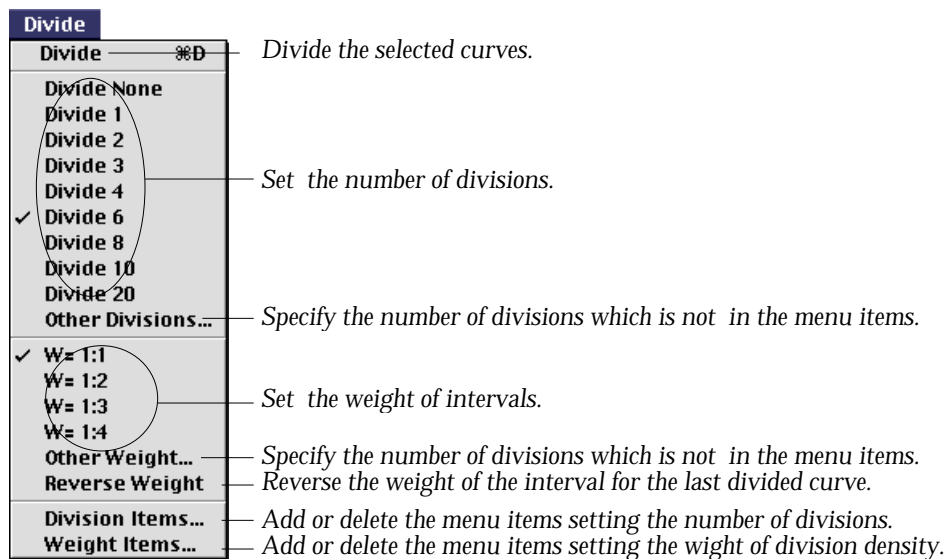


< Building boundary curves of surface primitives >

## Curve Division

Many mesh generation schemes in VisualFEA are based on curves, which may be boundary curves or control curves within or at the boundary of the mesh region. These curves should be divided before being used in mesh generation. The mesh density is determined primarily by the density of the division.

Dividing a curve is, in fact, the same as assigning nodes on the curve with specified intervals. In case of frame structure analysis, the curve segments between adjacent nodes are used as members (elements). But, for other cases, these segments are used only for boundary edges of surface mesh.



< Division of curves >


## Dividing curves

Selected curves can be divided simply by selecting a **Divide** menu item. They can be divided individually or as a whole, with specification of the following two factors:

- Number of divisions : specifies how many segments the curve is to be divided into. This factor is set by the menu items such as "Divide 1", "Divide 2", ... , and "Other Divisions..." The number of divisions is represented by  $n$  in the figure below.
- Weight of division density : specifies the ratio between the segment lengths at both ends of the curve. It is the ratio between  $L_1$  and  $L_n$  in the figure below. This factor is set by the menu items such as "W=1: 1", "W=1: 2", ... , and



< Number of divisions and weight of division density >

Previously divided curves can be divided again with new division factors. The new division always overrides the old division. However, it is not allowed to divide curves which have already been involved in mesh generation and thus have become a part of the mesh. The curve selection tool  should be activated in order to divide curves.



### ■ Dividing curves individually

A group of selected curves as well as a single curve may be divided at once, with individual application of the specified number of divisions and weight of division density. This can be achieved simply in the following steps.

- 1) Set the number of divisions and the weight of division density by selecting the corresponding items of **Divide** menu, if necessary.  
The menu items corresponding to the number of divisions and the weight of division density are marked.
- 2) Select curve(s) to divide.  
Do not select any meshed curve which is already used for mesh generation. In order to divide a meshed curve, the mesh associated with it should first be removed,
- 3) Select one of the following **Divide** menu items.

- "Divide" item

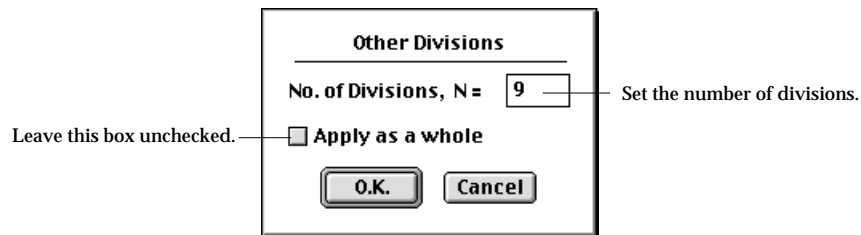
The number of divisions and the weight of division density with marks are applied in dividing the selected curve(s).

- An item for the number of divisions

The selected curves are divided by the number of divisions represented by the menu item.

- "Other Divisions..." item

"Other Divisions" dialog box appears. This dialog is useful for dividing the curve(s) by a number of divisions not shown as a menu item. Set **No. of Divisions, N =**  item, and click **O.K.** button. Be sure ☐ **Apply as a whole** item is not checked.

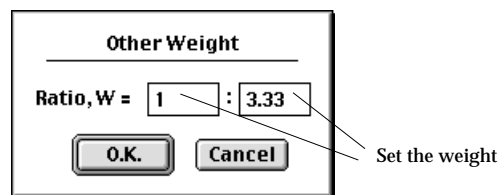


- An item for the weight of division density

The selected curves are divided with the weight represented by the menu item.

- "Other Weight..." item

This dialog is useful for dividing the curve(s) with weight of division density not shown as a menu item. Set **Ratio, W =**  :  items, and click **O.K.** button.



4) Select "Reverse Weight" item, if necessary.

If the weight is applied in the direction contrary to your intention, select "Reverse Weight" menu item. Then, the direction is reversed.



## ■ Dividing curves as a whole

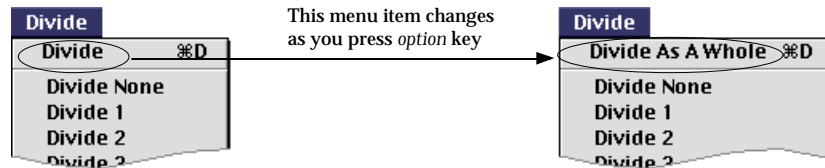
A group of selected curves may be divided as a whole. In this case, the number of divisions is applied as the total number of divisions in all the selected curves. This can be achieved by the following steps.

- 1) Set the number of divisions and the weight of division density by selecting the corresponding items of **Divide** menu, if necessary.

- 2) Select curve(s) to divide.
- 3) Select one of the following **Divide** menu items.

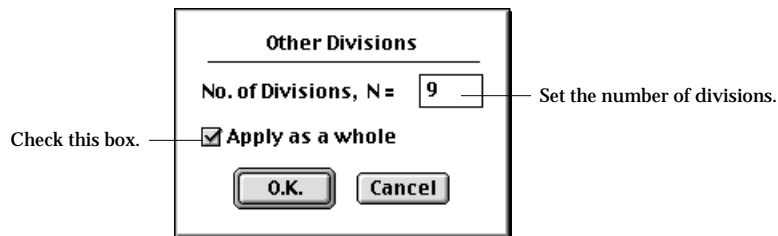
- "Divide As a Whole" item

If you pull-down **Divide** menu with  key (Windows :  key) pressed, you will find the first menu item is turned into "Divide As A Whole."

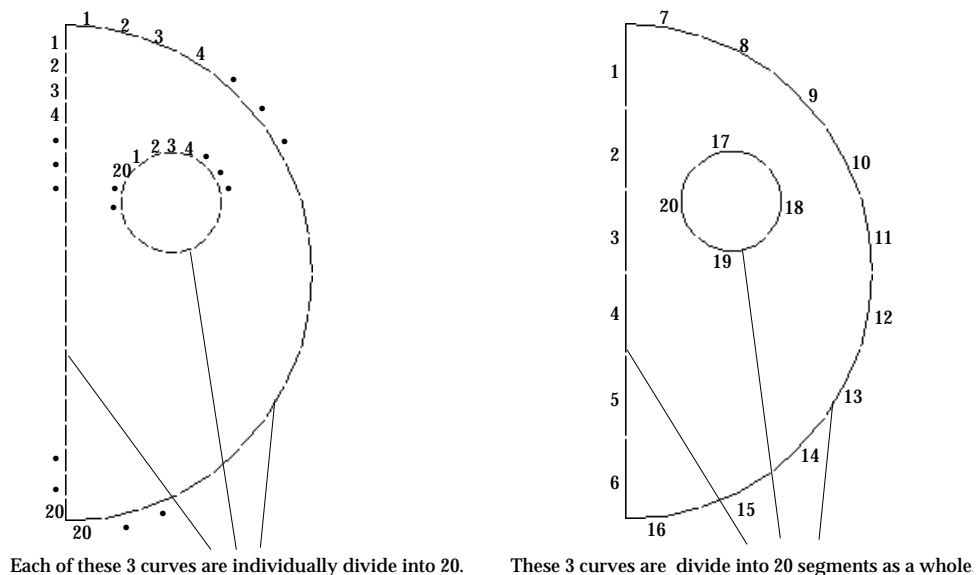


- "Other Divisions..." item

"Other Divisions" dialog box appears. This dialog is useful for dividing the curve(s) by a number of divisions not shown as a menu item. Set No. of Divisions, N =  item, and click  button. Be sure that ☒ **Apply as a whole** item is checked.



The weight of division density is not applied in case of dividing curves as a whole. Therefore, there is no need to reverse the weight.



< Comparison of dividing curves individually and as a whole >

### ■ Setting the number of divisions

The currently effective number of divisions is indicated by ☐ mark in front of the items for number of divisions such as "Divide 1", "Divide 2", ... , and "Other Divisions..." Select one of these items to set the number of divisions effective for future use. The newly selected item is ☒ marked, and selected curve(s) is(are) divided by the specified number. The corresponding number of divisions will be applied for future curve division by "Divide" item.




### ■ Setting the weight of division density

The currently effective weight of division density is indicated by ☐ mark in front of the items for weight of division density such as "W=1:1", "W=1:2", ... , and "Other Weight..." Select one of these items to set the weight of division density effective for future use. The newly selected item is ☒ marked, and selected curve(s) is(are) divided with the specified weight. The corresponding weight of division density will be applied for future curve division by "Divide" item.

### ■ Removing divisions from divided curves

It is possible to remove divisions from divided curves. Select "Divide None" item from **Divide** menu. Then, all the selected curves will be restored to intact state with no division. But, in the case of frame analysis, a straight line is considered as one element even if it is not divided. Therefore, "Divide None" applied to a straight line has the same effect as "Divide 1" in frame analysis.

### ■ Dividing curves using F keys

A single curve or a group of selected curves may be divided using F keys(F1 - F24). The curve division is achieved simply by pressing one of the F keys after selecting curve(s). The F number corresponds to the number of divisions. For example, pressing  will divide the selected curve(s) into 3. If  key (Windows :  key) is pressed at the same time, "Apply as a whole" option applies.

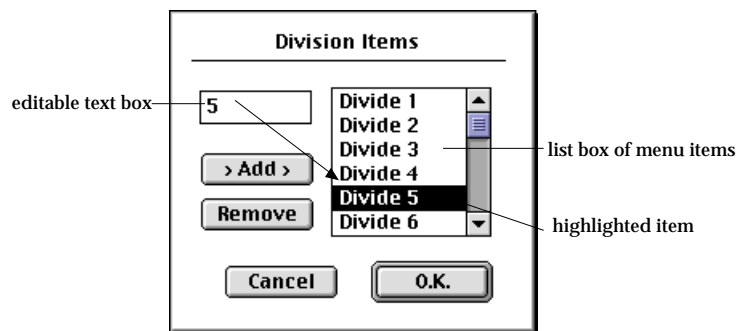


## Changing **Divide** Menu Items

You may want to divide the selected curves using either a number of divisions or a weight of division density not on the menu items. In this situation, you may use either of two methods. The first is to select "Other Divisions..." or "Other Weight..." item and set the dialog that follows. It is rather tedious to repeat this procedure each time you are working with a certain number of divisions or weight of division density. So, in such a case, another way of doing this more conveniently is to change the menu items as explained in the following.

### ■ Modifying the menu items for number of divisions

You may add, delete or change the menu items for number of divisions. First, select "Other Divisions..." item from **Divide** menu. Then, "Other Divisions" dialog appears as shown below.



You can modify the menu items using the dialog as explained below.

In order to change a menu item,

- 1) Click the item shown in the list box containing menu items.  
If there are too many menu items, only part of the menu items are displayed in the list box. Scroll the list to make the desired item visible, if it is not displayed in the box. The clicked item is highlighted, and the text of the item is echoed in the text box.
- 2) Edit the text box.  
As you edit the text, the highlighted item in the list box is changed accordingly.

In order to add a menu item,

- 1) Click **> Add >** button.  
The highlighted item in the list box is duplicated and the text of the item is echoed in the text box.
- 2) Edit the text box.  
As you edit the text, the highlighted item in the list box is changed accordingly.

In order to delete a menu item,

- 1) Click the item in the list box.

The clicked item is highlighted, and the text of the item is echoed in the text box.

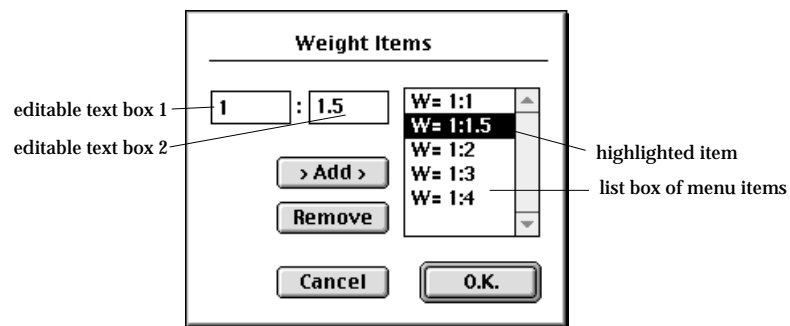
- 2) Click **Remove** button.

The highlighted item is removed from the list, and another item, one above or below, is highlighted.

After you complete adding, deleting or changing the menu items, click **O.K.** button. Then, "Other Divisions" dialog is closed, and **Divide** menu is updated in accordance with what you have done with "Other Divisions" dialog.

### ■ Changing the menu items for weight of division density

You may add, delete or change the menu items for weight of division density. First, select "Other Weight..." item from **Divide** menu. Then, "Other Weight" dialog appears as shown below.



You can modify the menu items using the dialog in the same way as explained above for modifying the menu items for number of divisions. One difference from the previous explanation is that there are two editable text items in "Other Weight" dialog. When you click an item in the list box, the text of the item in the form of m:n is echoed in two text boxes  : . You can edit each of these texts separately.

# **Chapter 4**

## **Mesh Generation**




## Chapter 4 Mesh Generation

The finite element method requires dividing the analysis region into many sub-regions. These small regions are the elements, which are connected with adjacent elements at their nodes. Mesh generation is a procedure of generating the geometric data of the elements and their nodes, and involves computing the coordinates of nodes, defining their connectivity and thus constructing the elements. Here, mesh designates aggregates of elements, nodes and lines representing their connectivity.

Capability and convenience of modeling the analysis domain are dominated by the mesh generation procedure. The geometric characteristics of generated elements affect the overall performance and accuracy of the finite element analysis. Therefore, mesh generation is one of the most important procedures in finite element modeling.

### General procedure of mesh generation

Various methods of mesh generation are provided in VisualFEA. However, the general procedure of mesh generation is almost the same whichever method is used, as can be summarized by the following few steps:

- 1) Activate one of the mesh generation commands.  
Choose a mesh generation command from **Mesh** menu. A corresponding mesh generation dialog box appears. For some mesh generation commands, the proper object selection tool is automatically activated. For example, curve selection tool  is always activated when "4 edges" surface mesh generation command is issued.
- 2) Select curves or surface meshes, which are used as a seed of mesh generation.  
For example, a surface mesh may be generated by revolving a selected curve about a specified axis. In this case, the selected curve is used as a seed of the mesh generation.
- 3) Set the dialog items.  
The dialog has various selectable items to set the shape of element, number of nodes, input methods and so on. Some input text items are provided for defining characteristics of the mesh generation.
- 4) Input axis or path if necessary.  
The axis of rotation or path of sweeping is needed for some mesh generation schemes, and may be determined by selecting an existing curve or creating a new one.
- 5) Press appropriate buttons in the dialog.  
Buttons vary depending on the input stage. When everything is ready for

mesh generation, the button approving mesh generation is enabled. Mesh is generated by clicking the button.

- 6) Repeat step 2) - 5) for next mesh generation.

It is not necessary to issue the same command again to use the same method for further mesh generation.

- 7) In order to terminate the mesh generation command, activate other commands, or click the close box of the dialog.

The mesh generation command is effective as long as the dialog remains on the screen. The command is terminated either by closing the dialog or by activating other command.

The specific procedures are different for each of the mesh generation schemes as detailed in the following sections.

Mesh	
Auto Mesh (Surface)	Unconstrained automatic triangulation on surfaces
Auto Mesh (Volume)	Automatic tetrahedronization in volumes
Auto Mesh (On Primitive)	Constrained automatic triangulation on surface primitives
2 Edges	Mapping using 2 edges
3 Edges	Mapping using 3 edges
4 Edges	Mapping using 4 edges
Extrude (Surface)	Surface mesh generation by extrusion
Extrude to Curve	Surface mesh generation by extrusion bounded to a curve
Translate (Surface)	Surface mesh generation by sweeping
Revolve (Surface)	Surface mesh generation by revolution
Twist (Surface)	Surface mesh generation by twisting
Box Edges (Volume)	Mapping using 12 edges of box shape
Prism Edges (Volume)	Mapping using 9 edges of prism shape
Tetra Edges (Volume)	Mapping using 6 edges of tetrahedron
Extrude (Volume)	Volume mesh generation by extrusion
Extrude to Surface Prim.	Volume mesh generation by extrusion bounded to a curve
Extrude to Mesh	Volume mesh generation by extrusion bounded to a mesh
Translate (Volume)	Volume mesh generation by sweeping
Revolve (Volume)	Volume mesh generation by revolution
Twist (Volume)	Volume mesh generation by twisting
Special Treatment	Other mesh treatments

## Surface Mesh Generation

A surface mesh is a mesh generated on a planar or a curved surface region, and is used for

- modeling a 2-dimensional domain such as a plane or an axisymmetric solid.
- modeling a curved surface domain such as shell.
- a seed mesh for generating a volume mesh.

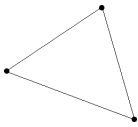
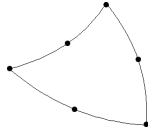
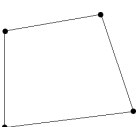
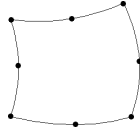
A surface mesh consists of planar or curved surface elements which are either triangular or quadrilateral. There are many methods for generating a surface mesh. They can be classified into the following 5 categories:

- automatic triangulation
- mapping
- sweeping
- duplication
- projection

There are a few methods in each category, as described in the following sections.

VisualFEA supports 4 types of surface elements, which may be used in surface meshes. Each of these element types may be defined on planes or on curved surfaces. The types of surface elements are shown in the following table. You may select one of these types to be adopted in mesh generation using the corresponding dialog box.

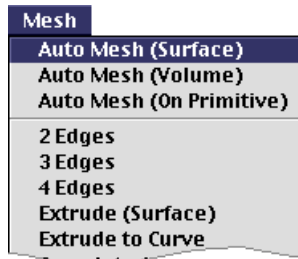
< Element types in surface meshes >

<p>3 node triangle (T3)</p> 	<p>6 node triangle (T6)</p> 
<p>4 node quadrilateral (Q4)</p> 	<p>8 node quadrilateral (Q8)</p> 

## Automatic triangulation

Automatic triangulation is the most convenient and powerful method of generating surface meshes. There are two types of automatic triangulation for surface mesh generation, i.e., unconstrained and constrained. Unconstrained automatic triangulation is to generate a mesh using the selected curves only. In this case, there are no other constraints for mesh generation than the curves inside and at the boundary of the region. This type of automatic triangulation is chiefly used for generating a plane mesh. Constrained automatic triangulation is to generate a mesh on a predefined surface primitive. This type of automatic triangulation is useful for generating a mesh on a trimmed surface.


### ■ Generating mesh on a plane by automatic triangulation

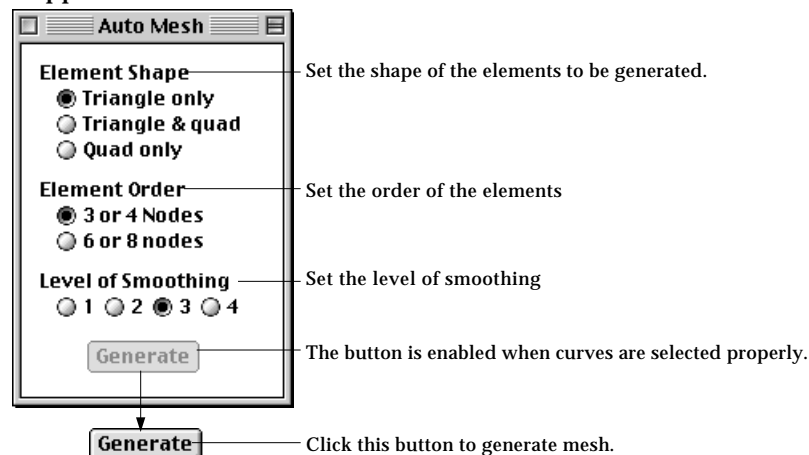


The easiest way of generating mesh on a plane is “unconstrained” automatic triangulation. An arbitrarily shaped surface mesh is generated simply by designating the curves of the mesh boundary and issuing a mesh generation command. A curved surface as well as a plane may be meshed by this method. However, it is advisable to limit the usage of this method to a plane mesh, and for a curved surface mesh to apply “constrained” automatic triangulation as explained in the later section.

You can easily fill an arbitrarily shaped surface region with triangular or quadrilateral elements using automatic triangulation as described below.

- 1) Choose “Auto Mesh” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Auto Tri” dialog box appears.



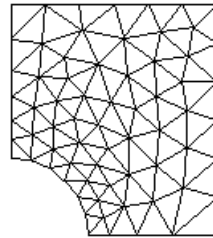
- 2) Set the element shape using “Auto Tri” dialog.

Click one of the buttons for element shape. There are following 3 options.

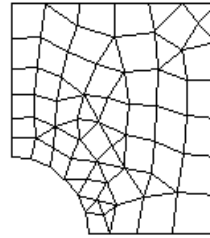
- “Triangle only” : Only triangular elements are generated.



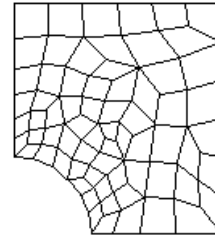
- “Triangle and quad” : The generated mesh is filled with mixture of triangular elements and quadrilateral elements.
- “Quad only” : Only quadrilateral elements are generated. In order to use this option, the total number of divisions in the selected curves should be even.



“Triangle only”



“Triangle and quad”

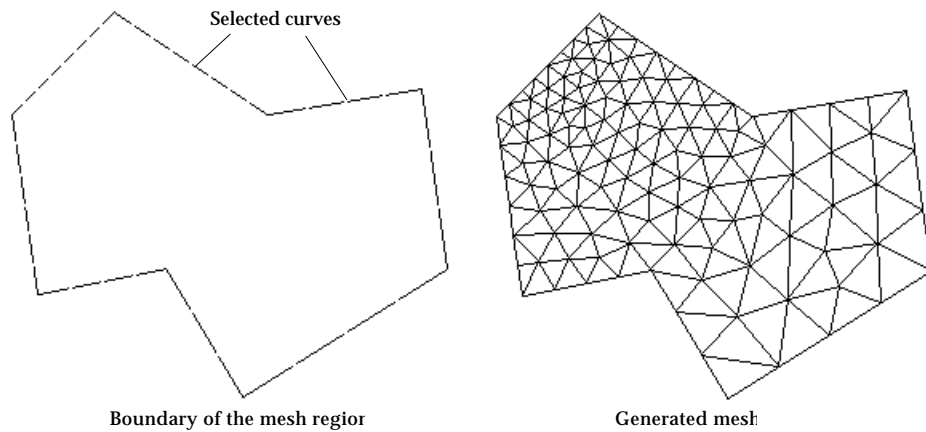


“Quad only”

< Mesh generated using different options of element type >

- 3) Set the element order.  
Click one of the buttons for the order of elements. Either linear or quadratic order can be selected.
  - “3 or 4 nodes” : Linear elements (3 node triangle or 4 node quadrangle) are generated.
  - “6 or 8 nodes” : Quadratic elements (6 node triangle or 8 node quadrangle) are generated.
- 4) Set the level of smoothing using “Auto Tri” dialog.  
The shape of the individual element is polished through smoothing process. The level of smoothing is defined in four grades, 1, 2, 3 and 4. Grade 4 takes longest time, but produces the best shaped elements.
- 5) Select curves.  
All the selected curves should be divided. Some of the selected curves are used in defining the boundary of the mesh region. Others may be needed to control the mesh density and element boundaries inside the region.
- 6) Click **Generate** button.  
A surface mesh is generated by automatic triangulation, if the selected curves are compatible for automatic triangulation as described in the next section. Otherwise, the action is ignored with a message notifying that automatic triangulation cannot not be executed with the selected curves.

In the above procedure, the order of step 2), 3), 4) and 5) can be interchanged. You may repeat the above procedure of generating mesh by automatic triangulation without issuing the command again, while “Auto Tri” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< Example of automatic triangulation >

### ■ Setting a compatible region for automatic triangulation

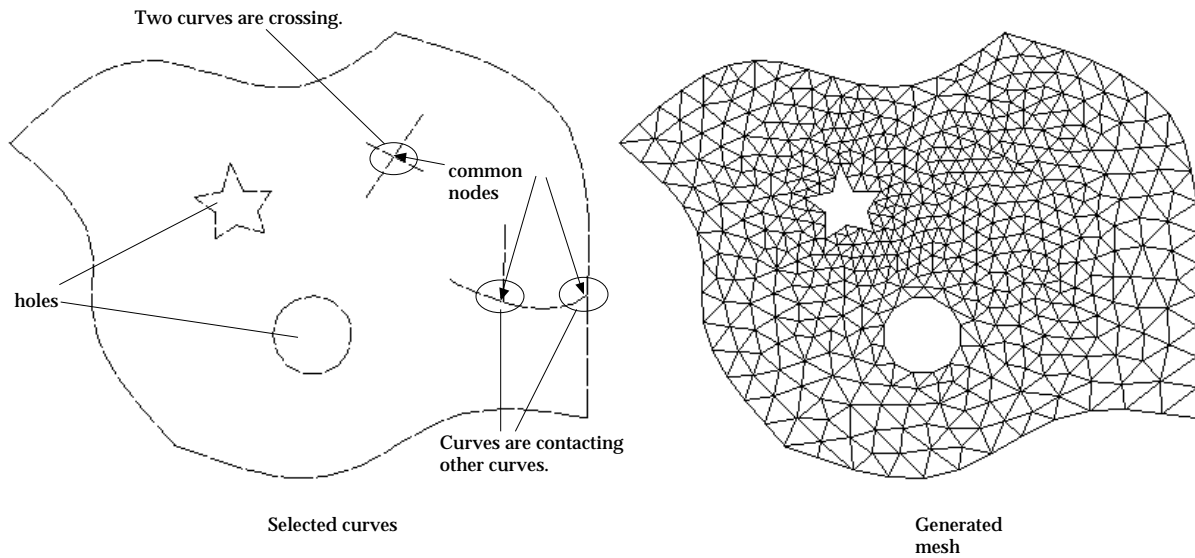
For successful automatic triangulation, curves should be selected so that they may form a region compatible for mesh generation as described below.

- The outer boundary of the region should be enclosed by serially connected curves.
- All the selected curves should be in one united region.
- The region boundary may be either convex or concave.
- The region may contain holes or control curves inside the boundary.
- Curves inside the region may cross or contact with each other, but they must have a common node at each crossing or contacting point.

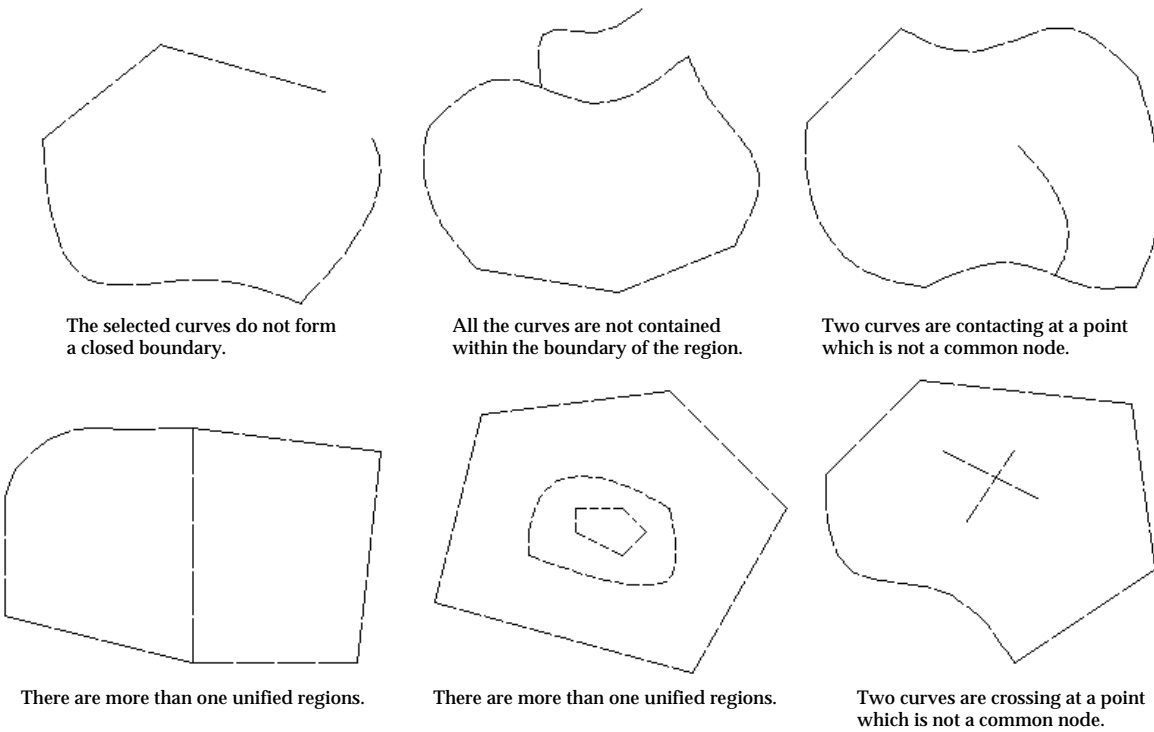
If you issue the automatic triangulation command with improper curve selection, you may receive one of the responses shown from VisualFEA depending on the state of the curve selection.

- The command is ignored with a warning message. It is the error message expected in most cases.
- The mesh is generated improperly. Undo the mesh generation, or delete the mesh, and retry with newly selected curves.
- You may get unpredictable results, and sometimes system crash. VisualFEA avoids this situation as much as possible. Such a situation rarely happens. But there may be still special cases which have not been excluded.

The following examples illustrate mesh regions compatible and incompatible for automatic triangulation.



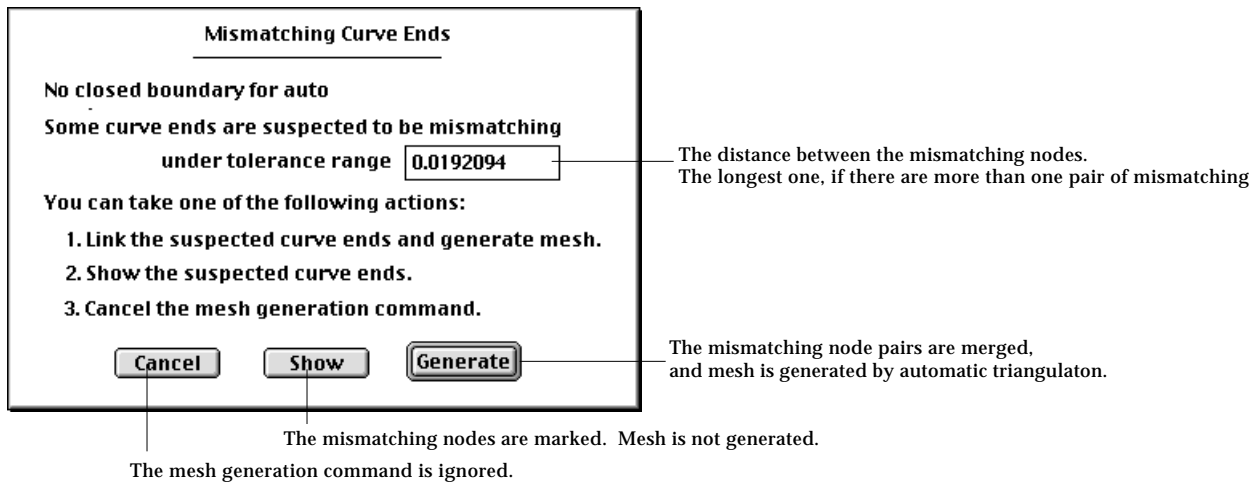
< An example of compatible region for automatic triangulation >



< Examples of incompatible region for automatic triangulation >

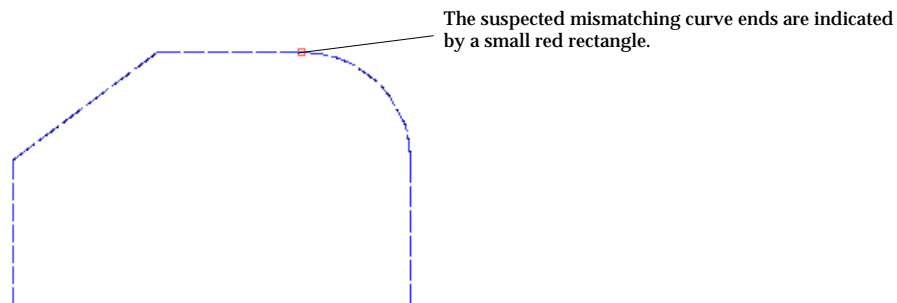
### ■ Handling the incompatibility due to mismatching curve ends

Although you selected curves compatible for automatic triangulation, you may not get a successful mesh generation. It is the case in which there exists visually unidentifiable incompatibility in the formation of boundary curves. The major source of such incompatibility is the numerical errors in various operations related to curves including curve-curve intersection. You may get a notice dialog box as shown below.



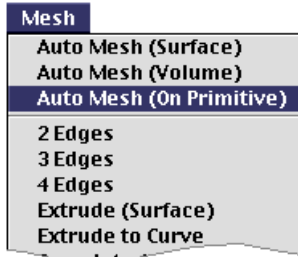
The dialog box informs that the boundary curves are not closed probably because some curve ends are mismatching. You may choose one of the 3 buttons provided by this dialog box.

- **Cancel** : cancel the mesh generation command.
- **Show** : display the suspected mismatching curve ends.
- **Generate** : Close the gap between the mismatching curve ends so that the boundary curves are close. You may set the tolerance range of the gap to be closed.




< A example of showing mismatching curve ends >

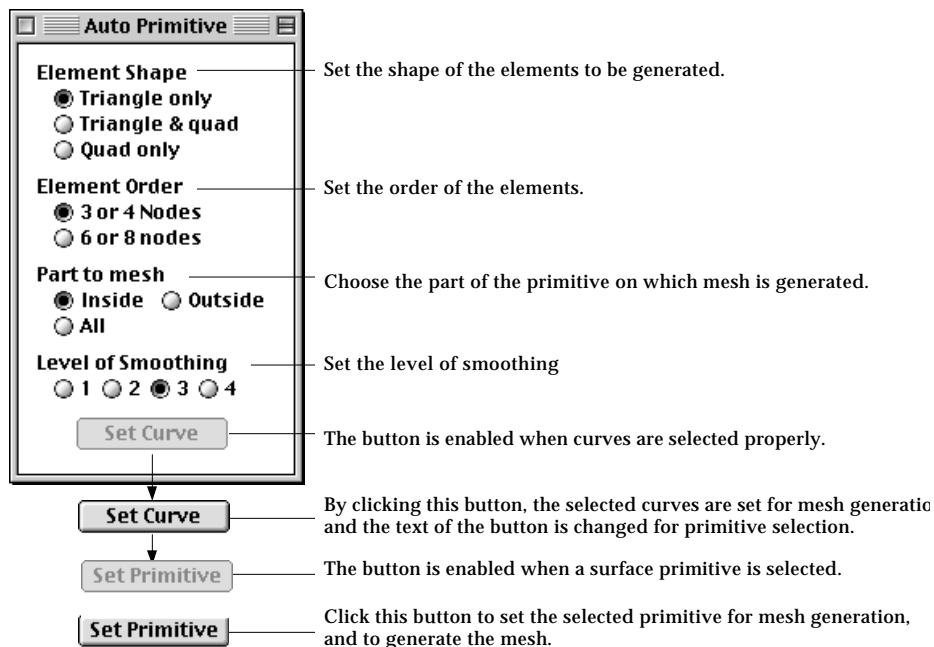
### ■ Generating mesh on a surface primitive by automatic triangulation



You may create surface primitives such as spheres, cylinders, torus, Bezier surface and so on (refer to “Creating surface primitives”). A mesh can be generated using selected curves and a surface primitive. The mesh is bounded and controlled by the selected curves, and at the same time, is confined on the surface primitive.

- 1) Choose “Auto Mesh on Primitive” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Auto Primitive” dialog box appears.



- 2) Set the element shape.


Click one of the buttons for element shape. There are 3 options.

- “Triangle only” : Only triangular elements are generated.
- “Triangle and quad” : The generated mesh is filled with mixture of triangular and/or quadrilateral elements.
- “Quad only” : Only quadrilateral elements are generated. In order to use this option, the total number of divisions in the selected curves should be even.

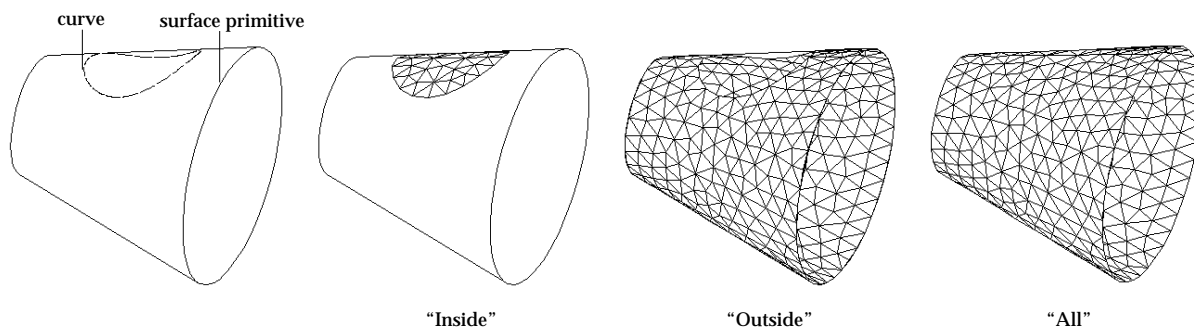
- 3) Set the element order.

Click one of the buttons for the order of elements. Either linear or quadratic order can be selected.

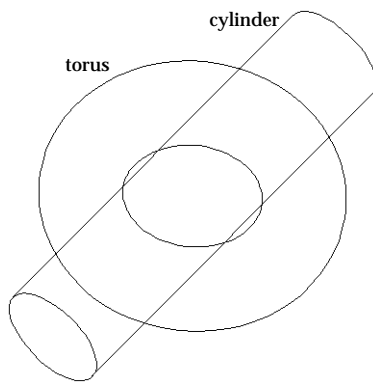
- “3 or 4 nodes” : Linear elements (3 node triangle or 4 node quadrangle) are generated.
- “6 or 8 nodes” : Quadratic elements (6 node triangle or 8 node quadrangle) are generated.

- 4) Choose the part of the primitive on which mesh is to be generated.  
A mesh may be generated inside, outside or both sides of the region boundary of the surface primitive. Choose the part by turning on one of the radio buttons “Inside”, “Outside” and “All”.
- 5) Set the level of smoothing using “Auto Primitive” dialog.  
The shape of the individual element is polished through smoothing process. The level of smoothing is defined in four grades, 1, 2, 3 and 4. Grade 4 takes longest time, but produces the best shaped elements.
- 6) Select curves.  
All the selected curves should be divided. Some of the selected curves are used in defining the boundary of the mesh region. Others may be needed to control the mesh density and element boundaries inside the region. The initially dimmed **Set Curve** button is enabled, when one or more curves are selected.
- 7) Click **Set Curve** button.  
The selected curves are reserved for mesh generation, and the button changes into **Set Primitive** indicating that selecting surface primitive is expected in the next step. The primitive selection tool  is automatically activated.
- 8) Select a surface primitive.  
The selected surface primitive is highlighted in bright red color. Only one surface primitive should be selected. The dimmed **Set Primitive** button is enabled when a surface primitive is selected.
- 9) Click **Set Primitive** button.  
A mesh is generated on the selected surface primitive. **Set Primitive** button is restored to **Set Curve** button. It is now ready for inputting another surface mesh.

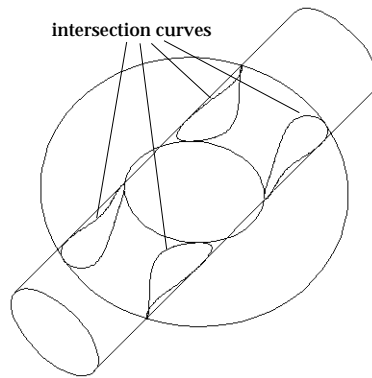
You may repeat the above procedure of generating mesh on a surface primitive, without issuing the command again, while “Auto Primitive” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



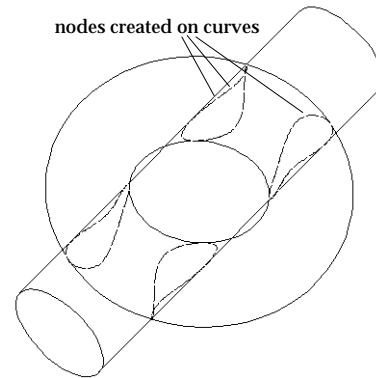
< Comparison of meshes generated using different options of part to mesh >



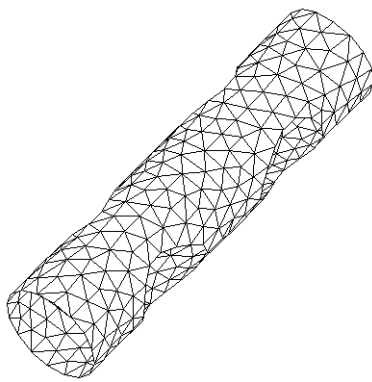
Create surface primitives.



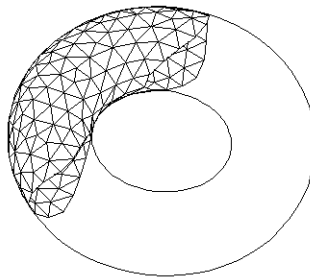
Obtain intersection curves of the two surface primitives.



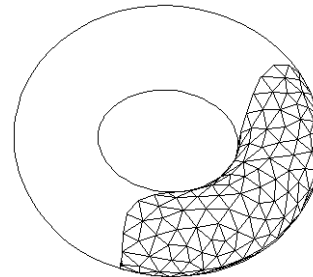
Divide the curves.



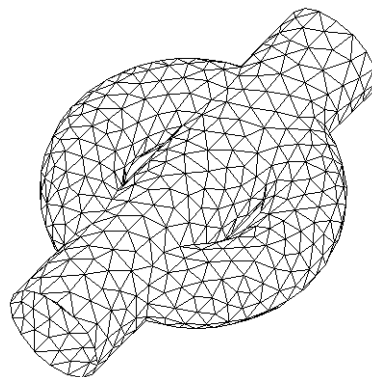
Generate mesh on the cylinder.



Generate mesh on one part of the torus.



Generate mesh on the other part of the torus.



Completed mesh

< Example of mesh generation using surface primitives and their intersection curves >

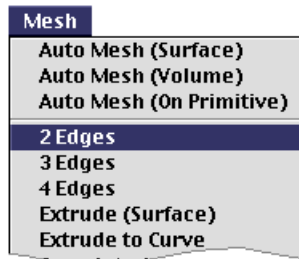
## Surface mesh generation by mapping

If 2, 3 or 4 edges are given, a mesh region is formed between or within the edges. VisualFEA supports 3 types of edge formation as follows:

- 2 edges
- 3 edges
- 4 edges


The nodal points of the generated mesh are computed by mapping techniques called “lofting”, “triangular mapping” or “transfinite mapping” respectively for each of the above 3 types. These techniques are not explained in detail here. The commands generating mesh by mapping are provided as menu items in **Mesh** menu.

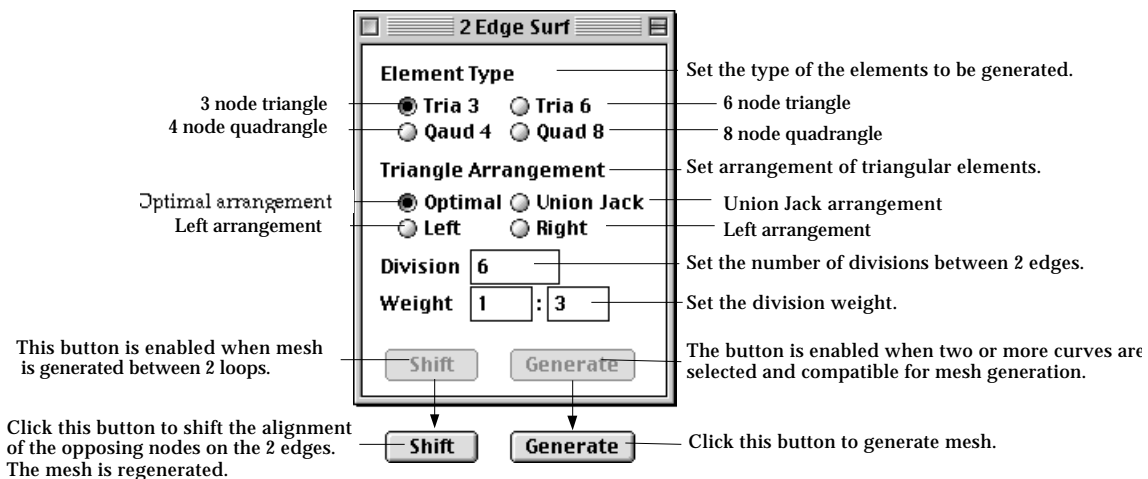
### ■ Generating mesh using 2 edges



A surface mesh can be generated using two opposite edges formed by a series of curves. This is the simplest and most frequently used method of mesh generation. The space between the two edges is filled with triangular or quadrilateral elements depending on your choice. The nodes on the mesh are created along the lines connecting two opposite nodes on the edges.

- 1) Choose “2 Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “2 Edge Surf” dialog box appears.



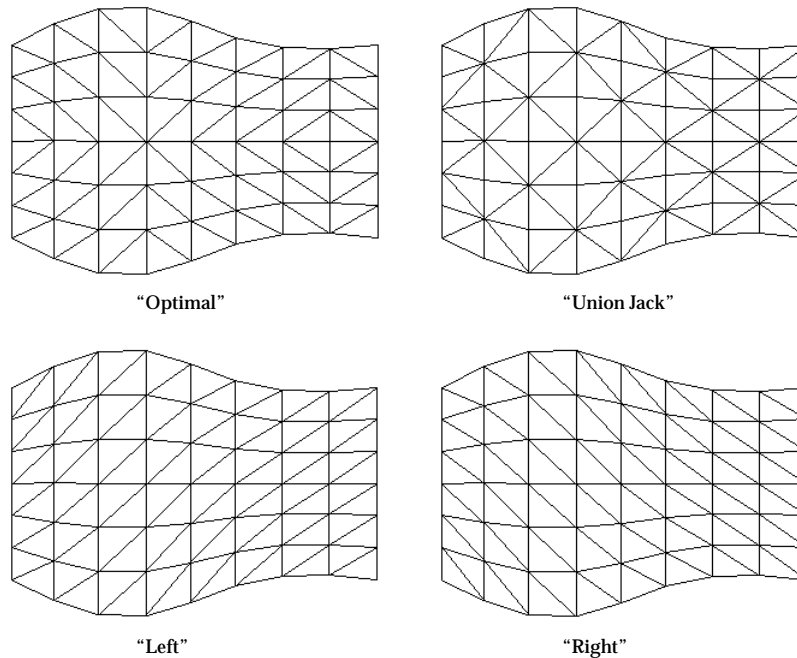
- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog. “Tria 3”, “Tria 6”, “Qaud 4” and “Quad 8” represent respectively 3 node triangle, 6 node triangle, 4 node quadrangle and 8 node quadrangle elements.

- 3) Select the type of arrangement for triangular elements.



The radio buttons for triangle arrangement are enabled if the element type is set as “Tri 6” or “Tri 3”, and disabled otherwise. These options are not effective for quadrilateral elements. The triangular elements can be generated in the 4 different types of arrangement as shown below.



<Arrangement of triangular elements>

- 4) Set the number of divisions between two edges.

It is the same as designating how many rows of elements to be generated between two edges. The number of divisions is denoted by  $n$  in the following example.

- 5) Set the weight of division density.

The weight of division density is entered in the form of  $w_1:w_n$ , which is the ratio between the length of the first division  $l_1$  and that of the last  $l_n$  as indicated in the figure below.

- 6) Select curves forming two edges.

All the selected curves should be divided, and form two edges, each of which consists of one curve or serially connected curves. The total divisions on each edge should be equal. **Generate** button is enabled when two edges are formed.

*The order of selecting curves does not make any difference in the resulting mesh generation. It is not necessary to designate the direction of the edges, although most other software asks to match the directions of the two edges. VisualFEA does it automatically.*

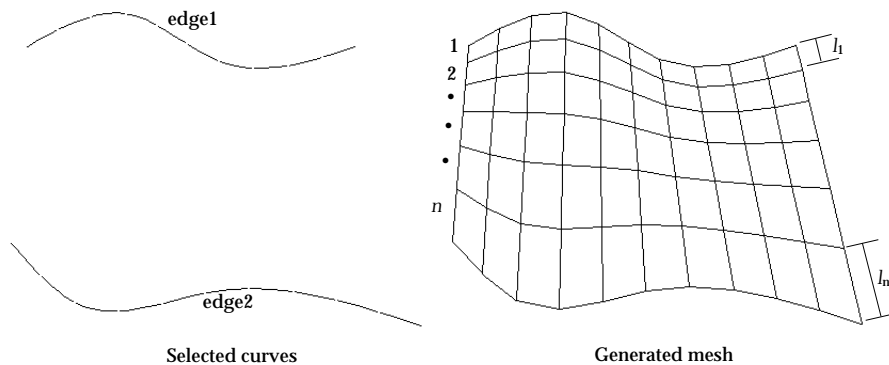
*The division is directed from the first edge to the second. Which one is the first is*

determined by the input order of curves forming the edges.

- 7) Click **Generate** button.

A surface mesh is generated between two edges, if the selected curves are compatible for mesh generation. Otherwise, the action is ignored after a message “Incompatible curve selection for 2 edge surface.”

In the above procedure, the order of step 2), 3), 4), 5) and 6) can be interchanged. You may repeat the above procedure of generating mesh using 2 edges without issuing the command again, while “2 Edge Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



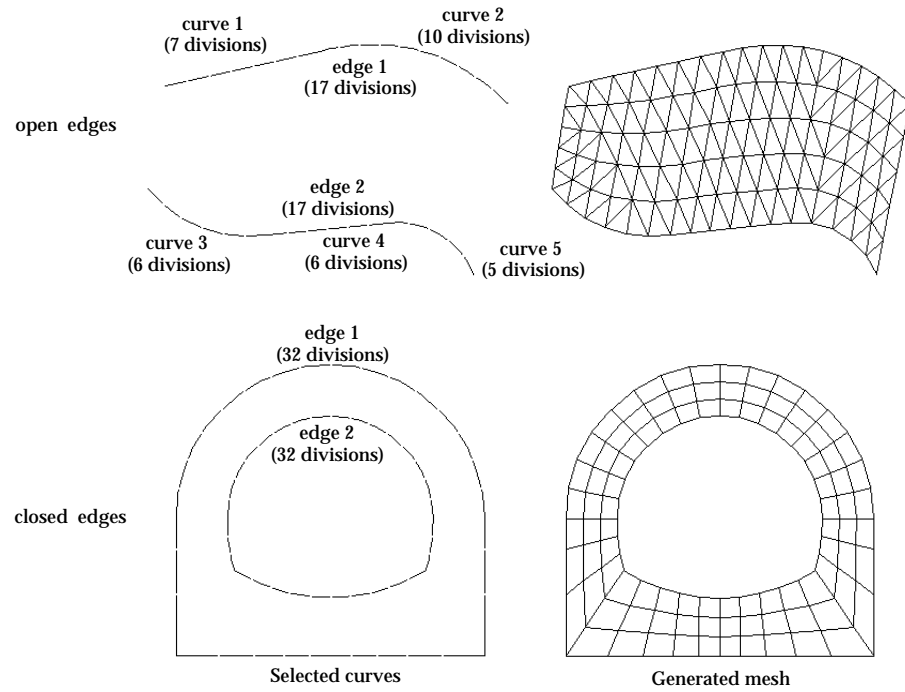
< Example of surface mesh generation between 2 edges >

### ■ Setting 2 edges compatible for mesh generation

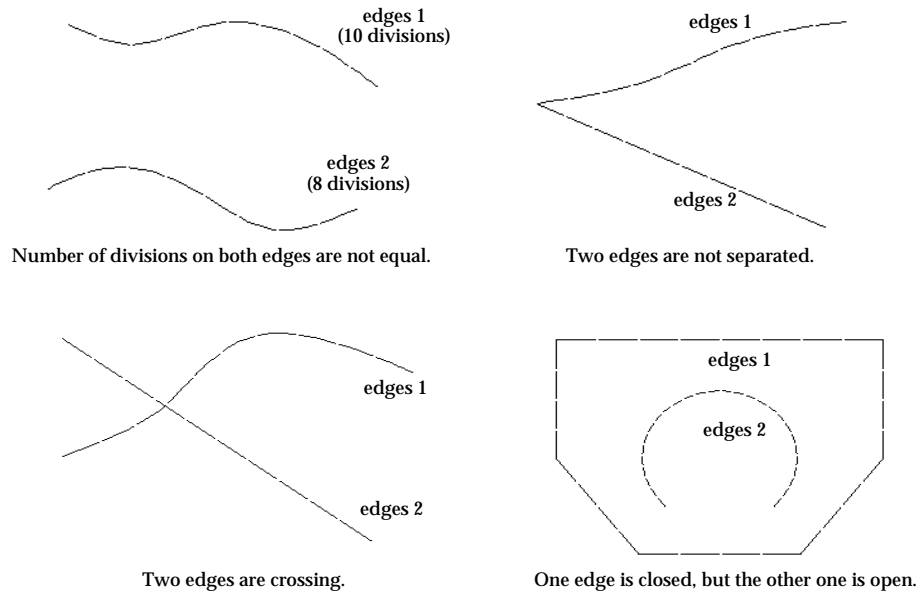
For successful 2 edge surface mesh generation, curves should be selected so that they may form 2 edges compatible for mesh generation as described below.

- The selected curves should be connected serially in two groups. This grouping for optimal mesh generation is automatically done by VisualFEA. Each of the two groups forms an edge. Each edge may consist of one or more curves.
- The two edges must be separated. They should not contact or cross with each other.
- Edges may be open or closed. If one edge is closed, the other edge must also be closed.
- Each edge should have an equal number of nodes on it.

Examples on the following page illustrate mesh regions compatible and incompatible for 2 edge mesh generation.



<Examples of 2 edge formation compatible for mesh generation>

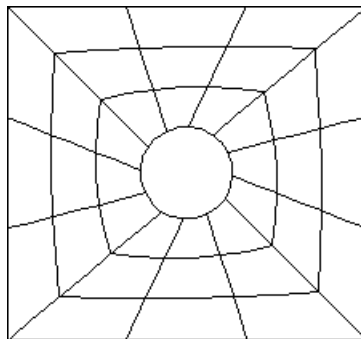


<Examples of 2 edge formation incompatible for mesh generation>

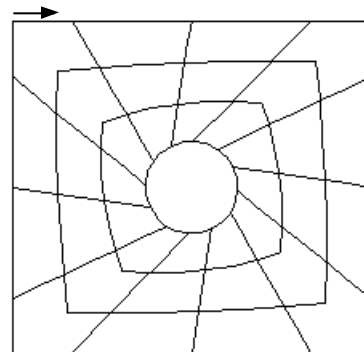
### ■ Shifting alignment of node pairing on 2 edges

If the two edges form closed loops respectively, the one-to-one pairing of the nodes on the edges are optimally determined by the software. However, it is rare but possible that the automatic pairing results in generation of elements with distorted or improper shape. The mesh shape can be improved by manually adjusting the alignment of the node pairing. This is achieved by pressing **Shift** button, in “2 Edge Surf” dialog, which is enabled when the mesh is generated. **Shift** button is valid only in case both edges form closed loops respectively. There are 7 different alignments, each of which is obtained one after another as the button is pressed consecutively. The sequence of the alignment is as follows:

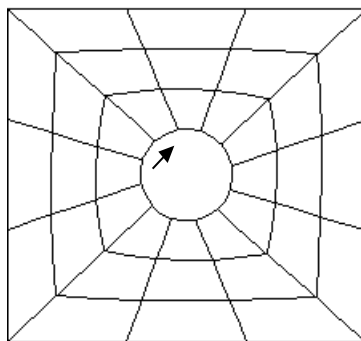
- 1) The outer loop shifts clockwise.
- 2) The inner loop shifts clockwise.
- 3) Both the inner loop and the outer loop shift clockwise.
- 4) The outer loop shifts counter-clockwise.
- 5) The inner loop shifts counter-clockwise.
- 6) Both the inner loop and the outer loop shift counter-clockwise.
- 7) The initial alignment is obtained again.



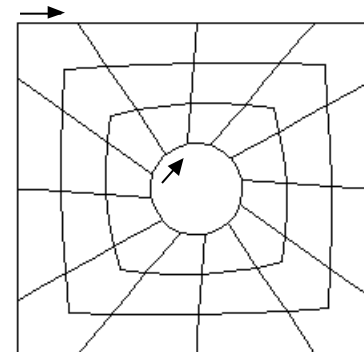
Initially generated mesh.



The outer loop shifts clockwise.  
("Shift" button is clicked once)



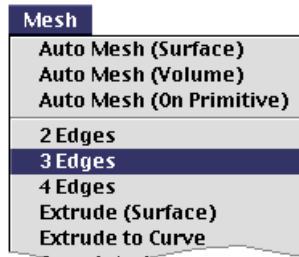
The inner loop shifts clockwise.  
("Shift" button is clicked twice)



Both the inner and the outer loop shift clockwise.  
("Shift" button is clicked three times)


< Example of alignment shifting in looped 2 edges >

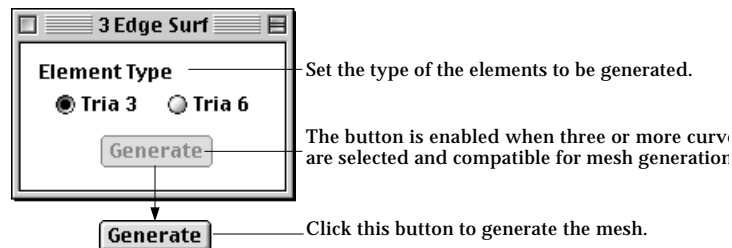
### ■ Generating mesh using 3 edges



A surface mesh can be generated using 3 edges formed by a series of curves. The region enclosed by the 3 edges is filled with triangular elements. The coordinates of the nodes on the mesh are determined by interpolating the nodal points on the 3 edges using triangular mapping technique. The region is mapped with an equilateral triangle. Therefore, the 3 edges should be formed in the configuration of a triangle.

- 1) Choose “3 Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “3 Edge Surf” dialog box appears.



- 2) Set the element type.

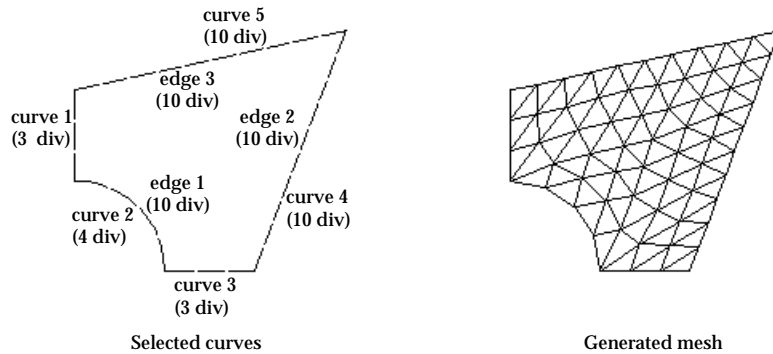
Only triangular elements can be generated using this method. Thus, the radio buttons for quadrilateral shape are not included in this dialog. “Tria 3” for 3 node triangle or “Tria 6” for 6 node triangle can be chosen as the element type.

- 3) Select curves forming 3 edges.

At least 3 curves are necessary to form 3 edges. They must be closed. If the selected curves meet these conditions, **Generate** button is enabled. In addition, the number of nodes should be the same for all the 3 edges. This condition is checked in the next step.

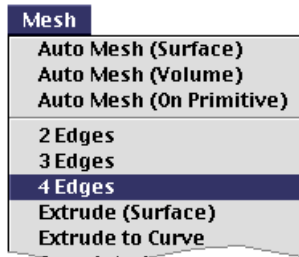
- 4) Click **Generate** button.

A surface mesh is generated in the region surrounded by the 3 edges, if the selected curves are compatible for mesh generation. When you click the button, the selected curves are automatically grouped into 3 edges compatible for mesh generation. Thus, all the 3 edges have equal number of nodes on them. If such formation of 3 edges is not possible, the action is ignored after a message “Incompatible curve selection for 3 edge surface.”




< Example of surface mesh generation enclosed by 3 edges >

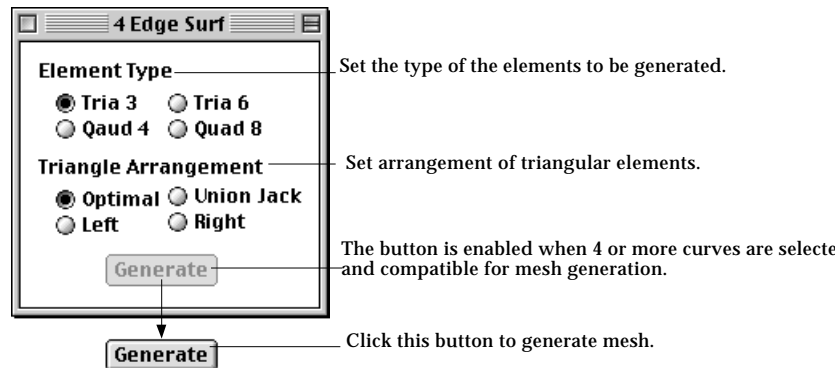
### ■ Generating mesh using 4 edges



A surface mesh can be generated using 4 edges formed by a series of curves. The region enclosed by the 4 edges is filled with triangular or quadrilateral elements depending on your choice. The coordinates of the nodes on the mesh are determined by interpolating the nodal points on the 4 edges using transfinite mapping technique. The region is mapped with a square. Therefore, the 4 edges should be formed in the configuration of a quadrangle.

- 1) Choose “4 Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “4 Edge Surf” dialog box appears.



- 2) Set the element type.

You may choose one of 3 node triangle, 6 node triangle, 4 node quadrangle, and 8 node quadrangle element as the element type.

- 3) Select the type of arrangement for triangular elements.

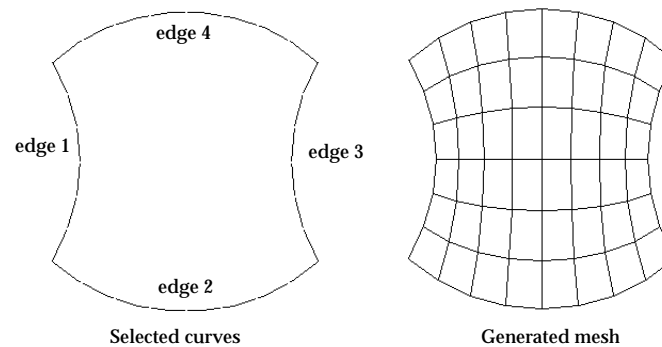
The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Select curves forming 4 edges.

At least 4 curves are necessary to form 4 edges. They must be closed. If the selected curves meet these conditions, **Generate** button is enabled. In addition, there must be the same number of nodes on each pair of opposite edges. This condition is checked in the next step.

- 5) Click **Generate** button.

A surface mesh is generated in the region surrounded by the 4 edges, if the selected curves are compatible for mesh generation. When you click the button, the selected curves are automatically grouped into 4 edges compatible for mesh generation.



< Example of surface mesh generation enclosed by 4 edges >

### ■ Setting 4 edges compatible for mesh generation

For successful 4 edge surface mesh generation, curves should be selected so that they may form 4 edges compatible for mesh generation as described below.

- The selected curves should be connected serially to form a closed loop with 4 edges.

*The 4 edges may be regarded as boundary lines of a quadrilateral region with two pairs of edges facing opposite to each other.*

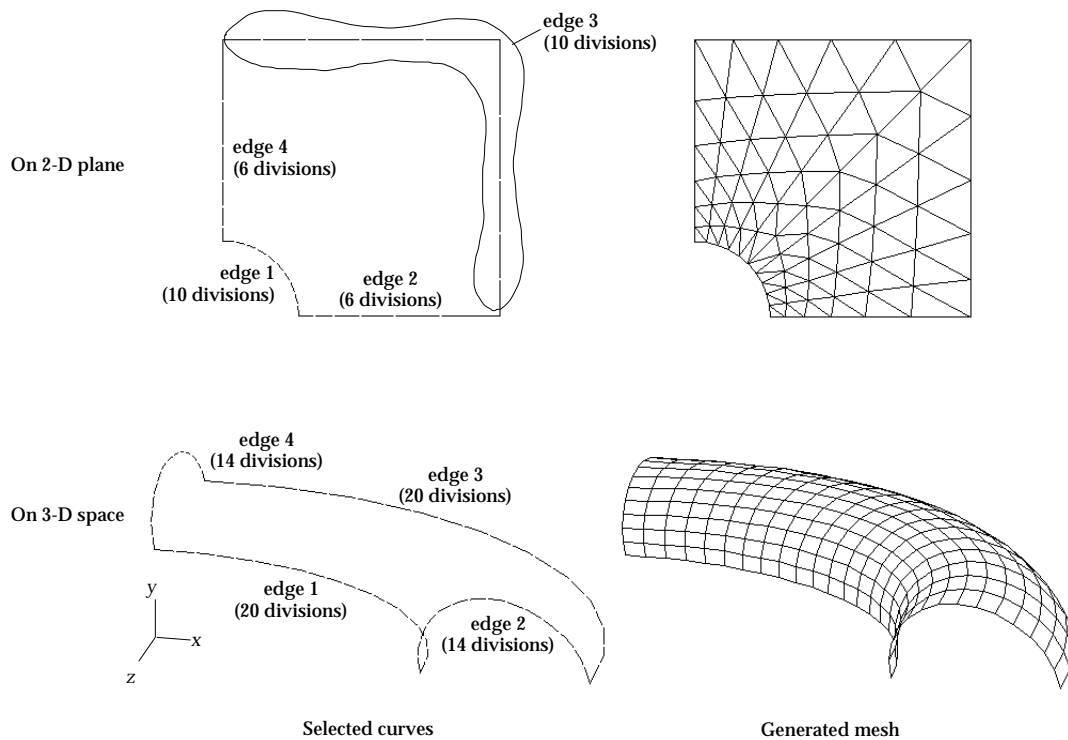
- Each pair of edges should have an equal number of nodes on them.

*Edges are constructed automatically by VisualFEA so that each pair of edges have equal number of nodes on them. If such construction is not possible, mesh generation is aborted after a message "Incompatible curve selection for 4 edge surface."*

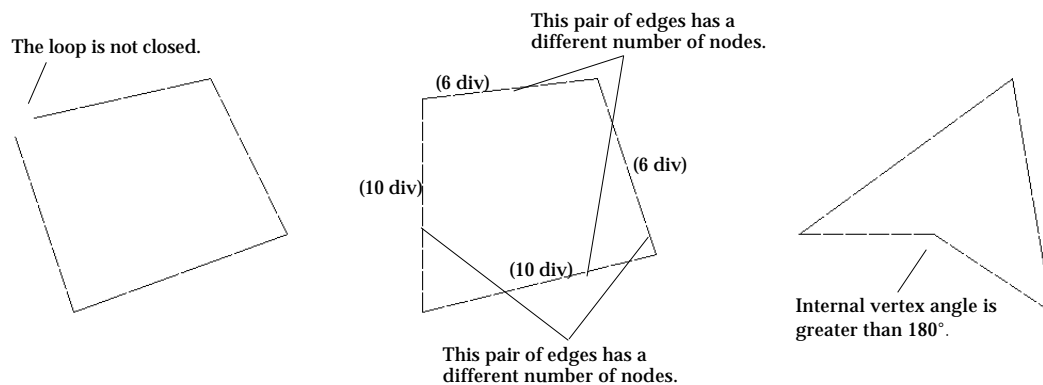
- All the internal angles of vertexes made by two adjacent edges should be less than  $180^\circ$ .

*Even if this condition is not met, a mesh may be generated. But the generated mesh will have improper shape and will trespass outside of the mesh region*

The following examples illustrate compatible and incompatible selection of curves for 4 edge mesh generation.



<Examples of 4 edge formation compatible for mesh generation>



<Examples of 4 edge formation incompatible for mesh generation>



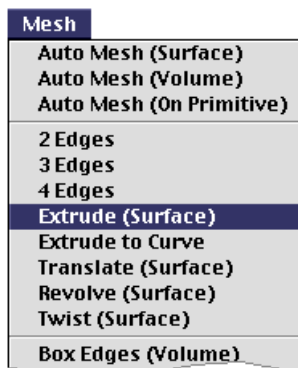
## Surface mesh generation by sweeping operations

VisualFEA supports the following 4 types of sweeping operations which may be used for surface mesh generation:

- Extrusion
- Translation
- Revolution
- Twisting


Sweeping creates a surface mesh by traversing the seed curves along a path defined in space. The sweeping operations, i.e., extrusion, translation, revolution, and twisting are distinguished by the characteristics of their sweeping path. The sweeping path for extrusion is a single straight line. Any continuous curve passing through, or meeting at the seed curve, may be used as the path for translation. The paths for revolution are circles with their centers along the axis of revolution. Twisting uses helical path for mesh generation. The commands for surface mesh generation by sweeping are provided as menu items in **Mesh** menu.

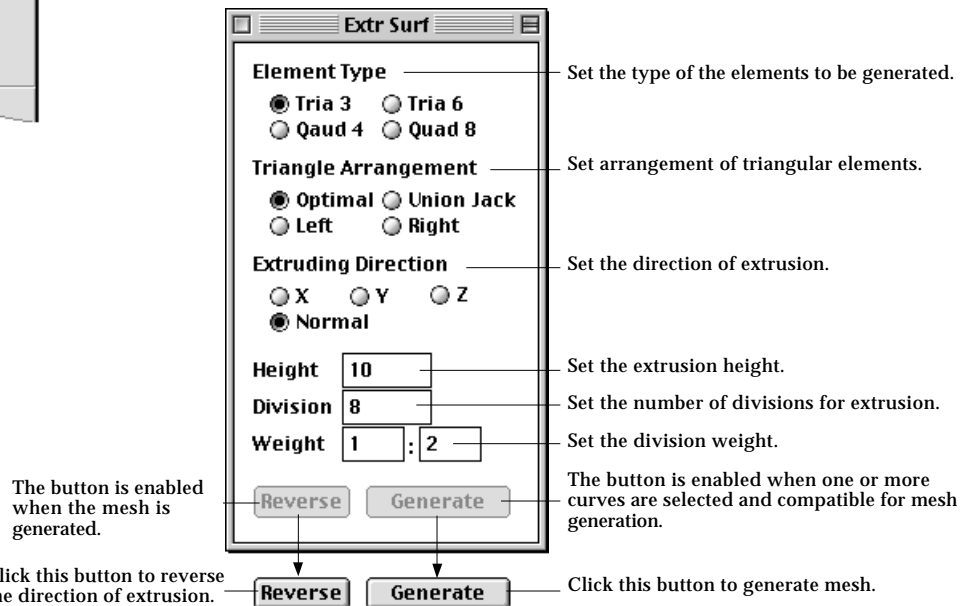
### ■ Generating mesh by extrusion



A surface mesh can be generated by extruding selected curves up to the specified height and in the specified direction. The height of the extrusion is entered using the “Extr Surf” dialog.

- 1) Choose “Extrude(Surface)” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Extr Surf” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog.

- 3) Select the type of arrangement for triangular elements.

The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Set the direction of extrusion.

The mesh may be extruded either in the direction of a coordinate axis, or in the direction normal to the seed curve. The normal direction is determined independently at each of the extruding nodes on the seed curve.

*The normal direction is defined on the plane of the seed curves. If the curves were created at a certain grid planes, the line of normal direction at every point of the curve lies on that grid plane. But, there are some 3-dimensional cases in which the normal direction cannot be determined uniquely. In such cases, the plane of normal direction is guessed by VisualFEA, and may not be what you are expecting. Then, you had better try other options.*

- 5) Enter the height of extrusion.

Extrusion height is the distance from the seed curve to the extent of the mesh generation.

- 6) Enter the number of divisions for extrusion.

Specify how many rows of elements to be generated by extrusion.

- 7) Enter the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of extrusion.

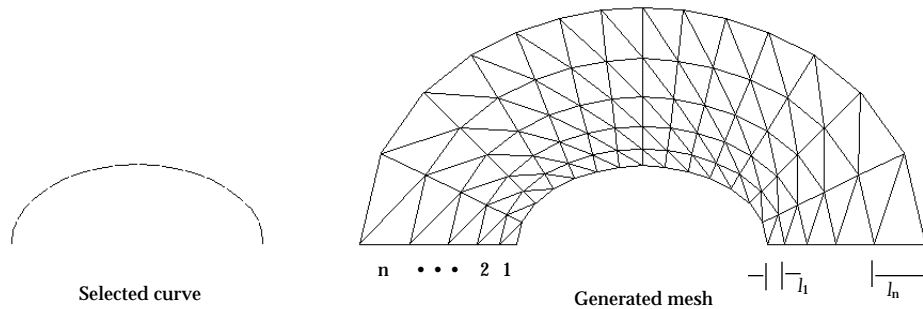
- 8) Select curves forming seed curves for extrusion.

All the selected curves should be divided, and form an edge, which may be one curve or serially connected curves. The edge may be either open or closed. **Generate** button is enabled when an edge is formed properly for mesh generation.

- 9) Click **Generate** button.

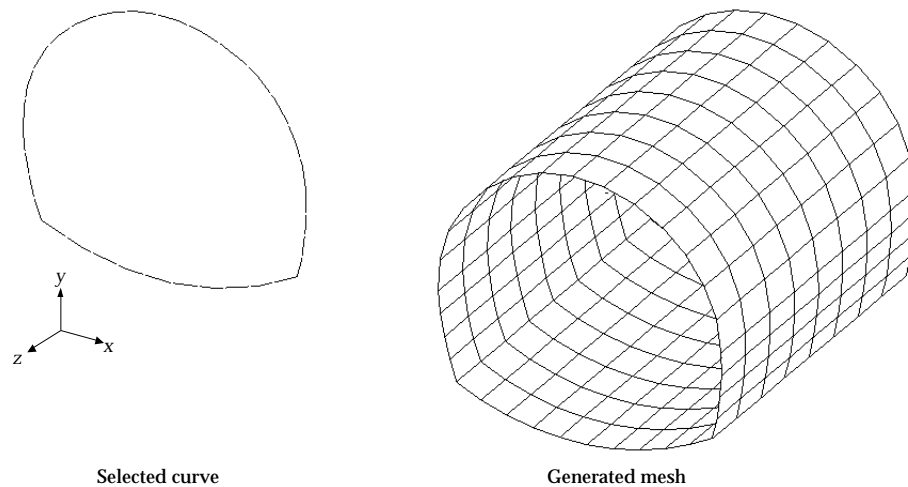
A surface mesh is generated by extruding the seed curve, if the selected curves are compatible for mesh generation. Otherwise, the action is ignored after a message “Incompatible curve selection for extrusion.”

At this stage, even if the cycle ended with failure, **Generate** button is disabled as it should be at step 2). **Reverse** button is enabled, only when mesh generation is successful. In case the mesh is generated opposite to the desired direction, click **Reverse** button to revert the direction of extrusion. Then, mesh will be regenerated with the reverse direction. You may repeat the above procedure of generating mesh by extrusion without issuing the command again, while “Extr Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< A surface mesh generated by extrusion in normal direction >

The direction of extrusion is not necessarily in the same plane as the seed curve. Extrusion can be used effectively in creating 3-dimensional surface mesh by extruding the seed curve out of its plane as shown in the example below.




< A 3-D surface mesh generated by extrusion in an out-of-plane direction >

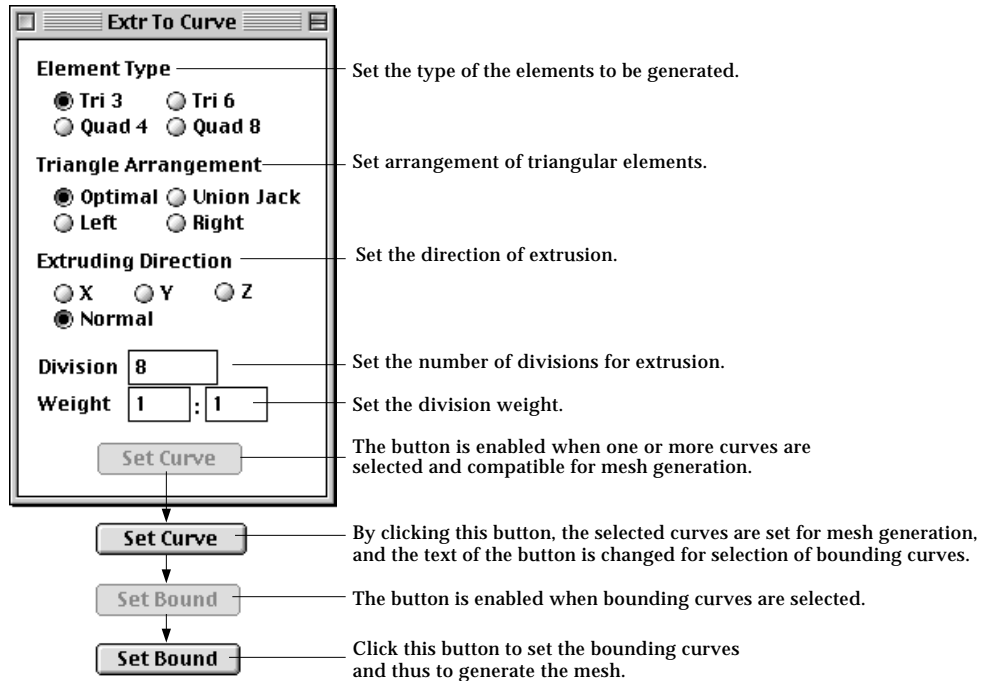
#### ■ Generating mesh by extrusion up to bounding curves

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)
2 Edges
3 Edges
4 Edges
Extrude (Surface)
<b>Extrude to Curve</b>
Translate (Surface)
Revolve (Surface)
Twist (Surface)
Box Edges (Volume)

Instead of specifying the height of extrusion, you may define the bound of the extrusion by selected curves. These curves are termed here as “bounding curve.” This method of mesh generation is the same as the above described extrusion method except that the extent of extrusion is determined not by its height but by the bounding curves. The advantage of this method is that the direction of extrusion as well as the boundary of the mesh can be controlled.

- 1) Choose “Extrude to Curve” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Extr to Curve” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog.

- 3) Select the type of arrangement for triangular elements.

The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Set the direction of extrusion.

The mesh may be extruded either in the direction of a coordinate axis, or in the direction normal to the seed curve. The direction is always signed toward the bounding curves.

*If both the seed curves and the bounding curves are closed, normal direction is the only option that can generate valid mesh by extrusion.*

- 5) Set the number of divisions for extrusion.

Specify how many rows of elements are to be generated by extrusion.

- 6) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of extrusion.

- 7) Select curves forming seed curves for extrusion.

All the selected curves should be divided, and form an edge, which may be one curve or serially connected curves.  button is enabled when an edge is formed properly for mesh generation.

- 8) Click  button.

The selected curves are reserved as the seed curves for mesh generation, and the button changes into  indicating that selection of bounding curves is expected in the next step.

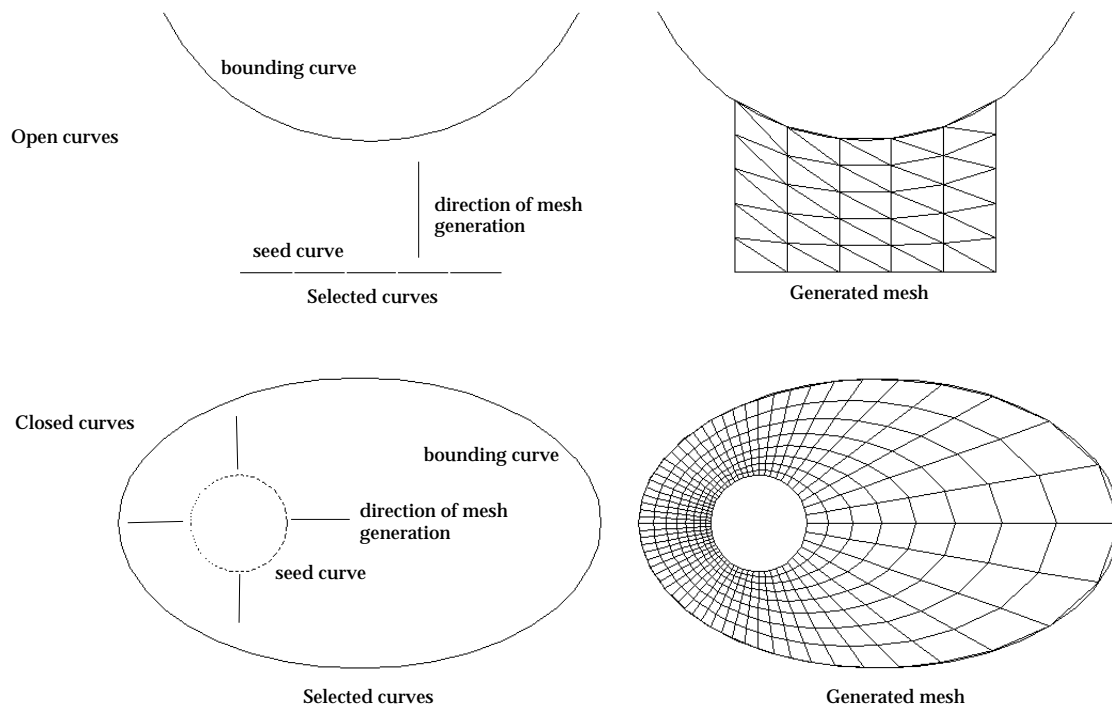
- 9) Select the bounding curves.

The selected bounding curves are highlighted in bright red color. The dimmed  button is enabled when bounding curves are selected.

- 10) Click  button.

A mesh is generated by extruding the seed curves up to the bounding curves.  button is restored to  button. It is now ready for generating another surface mesh.

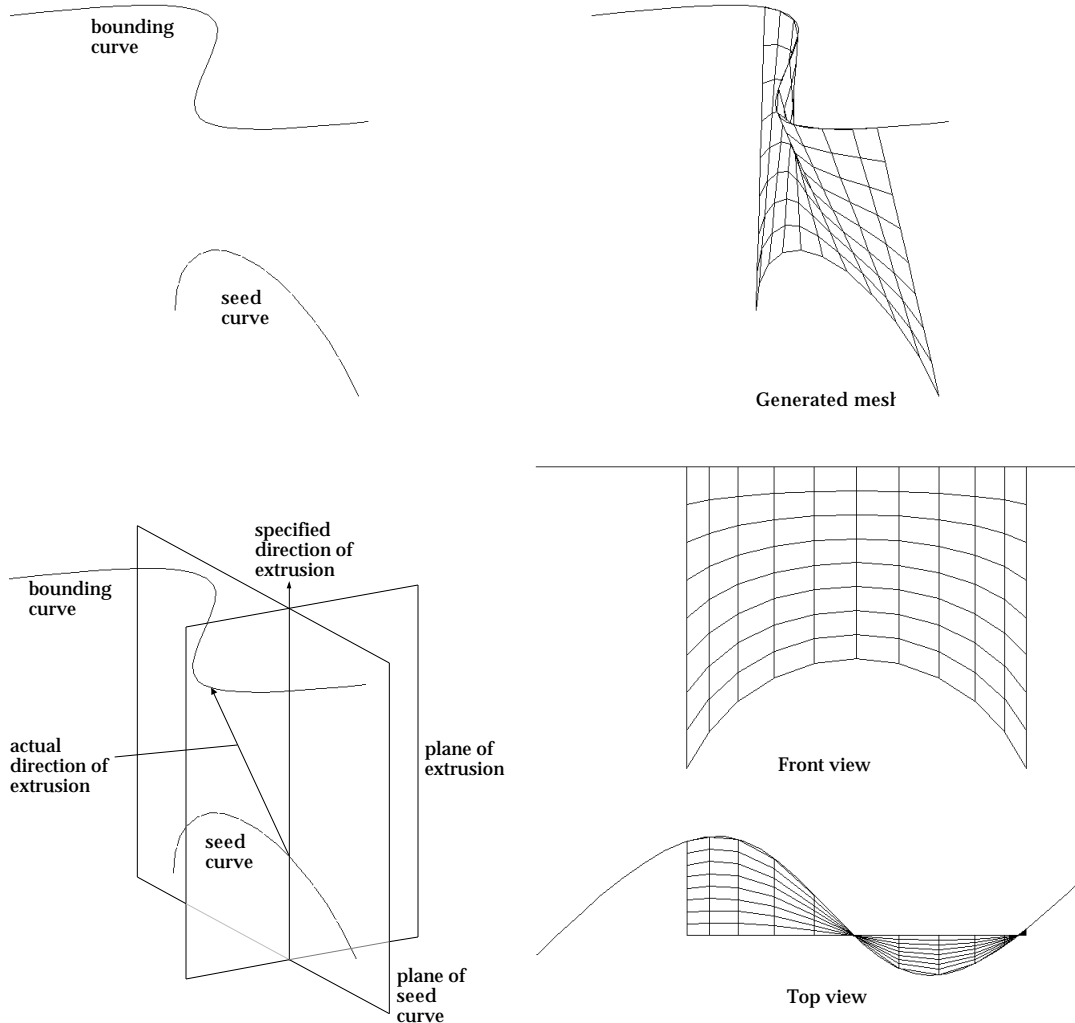
*The bounding curve should be closed if the seed curve is closed, and should be open otherwise. If the bounding curves are open, they must be long enough to cover the whole range of extrusion. Otherwise, the mesh generation will be aborted with a message, "Insufficient coverage of the bounding curve."*



<Mesh generated by extrusion up to bounding curves>

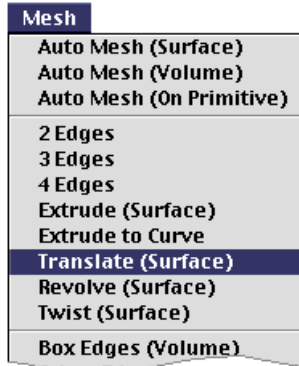
You may repeat the above procedure of mesh generation without issuing the command again, while “Extr to Curve” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.

The seed curves and the bounding curves are not necessarily in the same plane. If they are not, the generated mesh will form a curved surface bounded by them. In general, the specified direction of extrusion cannot be realized exactly in this case. However, the actual direction of extrusion is determined so that its projection on the plane of the seed curve agrees with the specified direction as shown in the figure below. The end point of extrusion is a point on the bounding curve intersecting the plane of extrusion which is normal to the plane of seed curve and includes the specified direction of extrusion.




< Determination of extruding direction in case the seed curve and the bounding curve are not on the same plane >

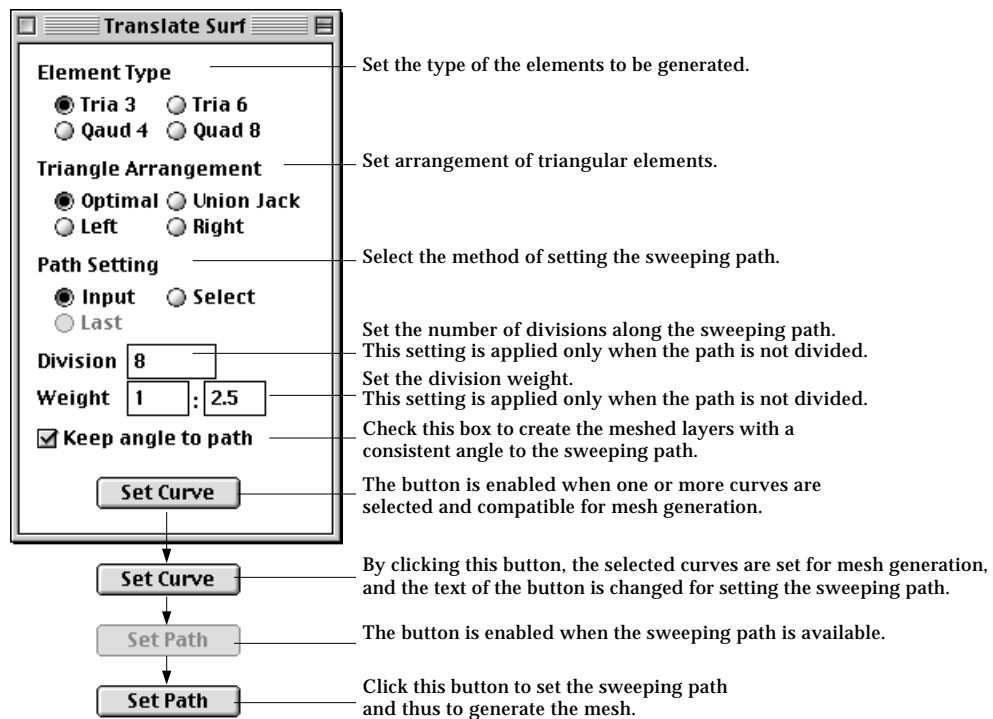
## ■ Generating mesh by translation



A surface mesh can be generated by translating the selected seed curves along the specified sweeping path. The boundaries of the generated elements are defined by a set of curves parallel to the seed curves and the other set parallel to the sweeping path. The previously described method of mesh generation by extrusion may be regarded as a special case of this operation in which the sweeping path is a straight line drawn in the specified direction.

- 1) Choose “Translate(Surface)” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Translate Surf ” dialog box appears.



- 2) Set the element type.


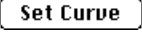




Choose one of the 4 element types given as radio buttons in the dialog.

- 3) Select the type of arrangement for triangular elements.

Triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Select the option for path setting.

Choose one of the 3 options setting the sweeping path, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is enabled only when this method of mesh generation was applied at least once since the dialog box appeared.

- 5) Set the number of divisions for translation.  
Specify how many rows of elements are to be generated by translation. This setting is applied only when the sweeping path is not divided. If divided curves are selected as the sweeping path, their divisions will be applied regardless of this setting.
- 6) Check or uncheck “Keep angle to path” check box.  
If this check box is checked, the meshed layers are created so that they have a constant angle with the sweeping path. Otherwise, all the layers are made parallel.
- 7) Set the weight of division density.  
Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of translation. This setting is also applied only when the sweeping path is not divided.
- 8) Select curves forming seed curves for translation.  
All the selected curves should be divided, and form an edge, which may be one curve or serially connected curves. The seed curves may be open or closed.  button is enabled when an edge is formed properly for mesh generation.
- 9) Click  button.  
The selected curves are reserved as the seed curves for mesh generation, and the button changes into  indicating that setting the sweeping path is expected in the next step.
- 10) Set the sweeping path.  
Set the sweeping path by the method selected at step 4).
  - “Input “ : If the method is set as “Input”, one of the curve input tool is activated, and the cursor changes into  shape. And, it is now ready for creating a sweeping path by inputting a new curve. At this step, line tool button  is activated so that straight lines may be entered. Any type of curves may be used as the sweeping path. In order to input desired types of curves, click the corresponding curve tool button.
  - “Select” : If the method is set as “Select”, the curve selection tool is activated, and thus cursor changes into  shape. Select a curve which will be used as the sweeping path. Either divided or undivided curves are acceptable.
  - “Last” : The sweeping path for last mesh generation is applied again. So, it is not necessary to input or select the sweeping path. This option can be used only when this method of mesh generation was applied at least once since the dialog box appeared.


*The sweeping path should be open, and its one end point should meet with a node on the seed curve. The sweeping path may be one curve or serially connected curves. If*



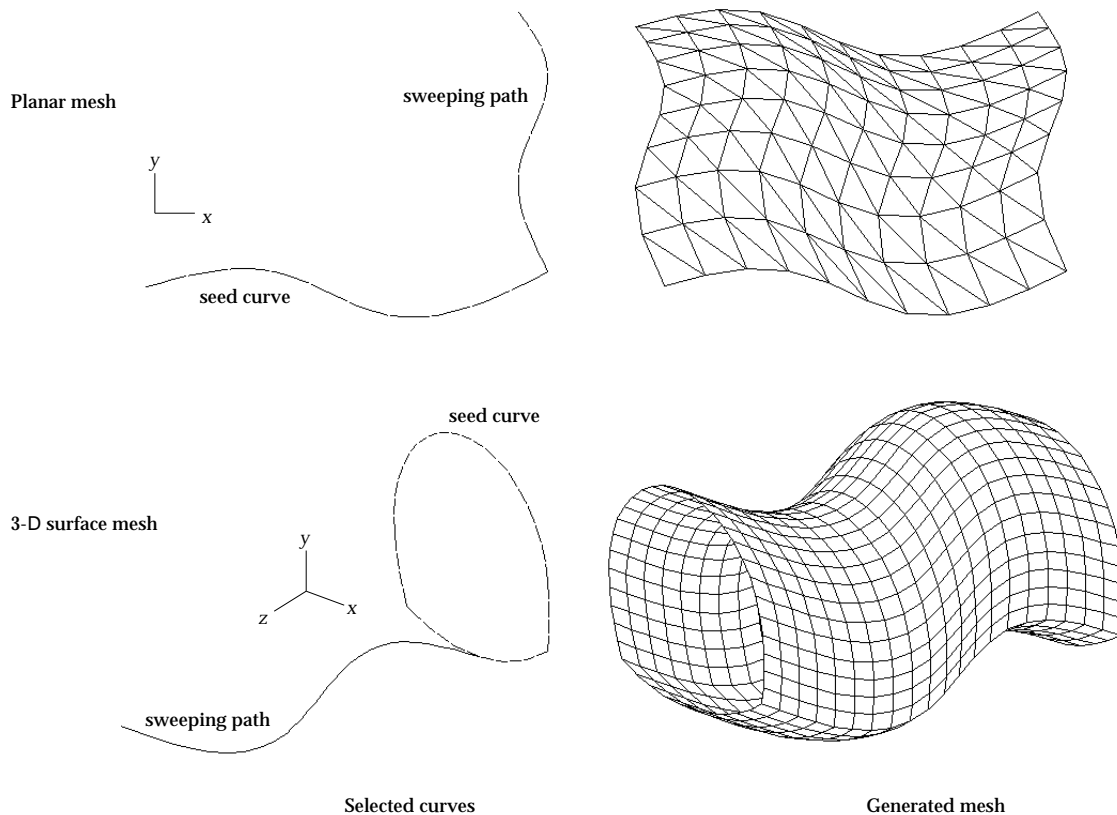
*the sweeping path consists of more than one curve, they must be either all divided or all undivided. Mixed use of divided and undivided curves for sweeping path is not allowed.*

- 11) Click  button.

A mesh is generated by translating the seed curves along the sweeping path.

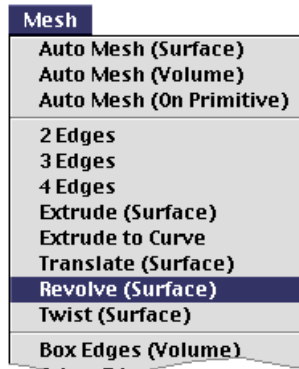
button is restored to  button. It is now ready for generating another surface mesh. The curve selection tool  is automatically activated, if it is not in action.

You may repeat the above procedure of mesh generation without issuing the command again, while “Translate Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.




< Example of mesh generation by translation on plane and in 3-D space >

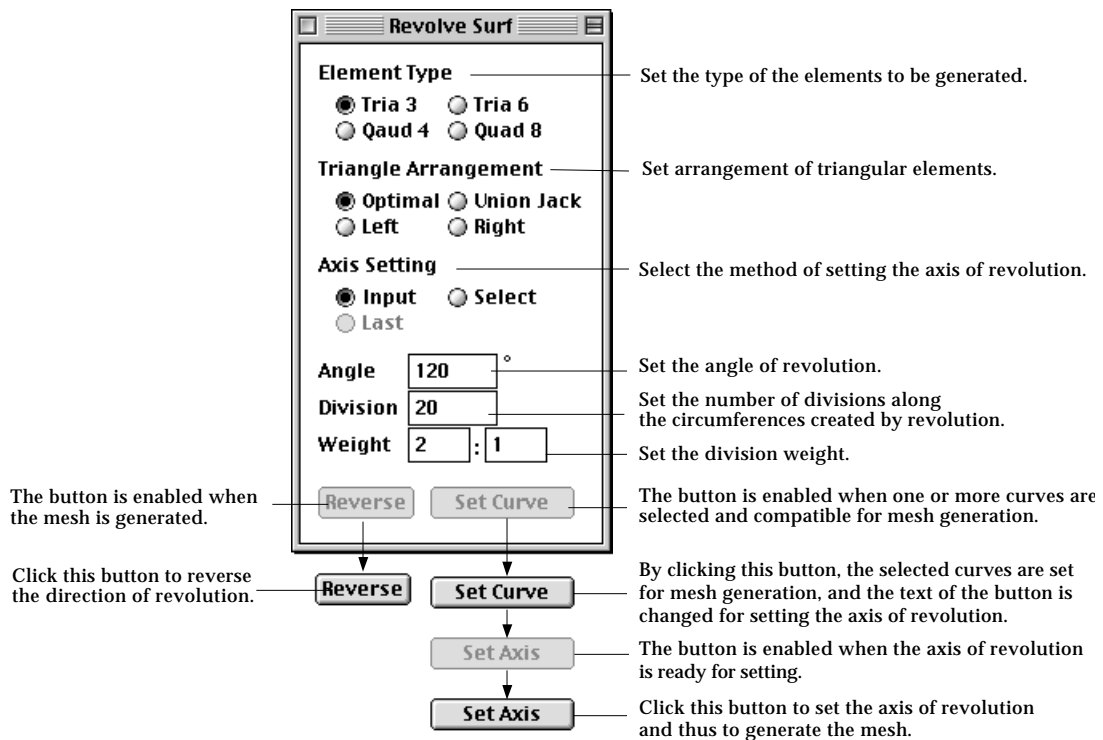
### ■ Generating mesh by revolution



Curved surface elements are generated on the surface of revolution created by revolving seed curves about the specified axis. The latitude and the longitude of the surface form the boundaries of the generated elements. The axis of revolution may be set interactively in any desired direction. Either a partial or a full surface of revolution is created depending on the specified angle. The direction of revolution can be reversed if necessary.

- 1) Choose “Revolve (Surface)” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Revolve Surf” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog.

- 3) Select the type of arrangement for triangular elements.

The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Select the option for setting the axis of revolution.

Choose one of the 3 options setting the axis of revolution, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is

enabled only when this method of mesh generation was applied at least once since the program started.

- 5) Set the angle of revolution.

Insert the angle of revolution in the dialog box. The angle should be greater than or equal to  $-360^\circ$  and less than equal to  $360^\circ$ . Both  $-360^\circ$  and  $360^\circ$  makes full surface of revolution. The negative sign reverses the direction of revolution.

- 6) Set the number of divisions for revolution.

Specify how many rows of elements are to be generated by revolution in circumferential direction.

- 7) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of revolution.

- 8) Select curves forming seed curves for revolution.



All the selected curves should be divided, and form an edge, which may be one curve or serially connected curves. The seed curves may be open or closed.  button is enabled when an edge is formed properly for mesh generation.

- 9) Click  button.

The selected curves are reserved as the seed curves for mesh generation, and the button changes into  indicating that setting the axis of revolution is expected in the next step.

- 10) Set the axis of revolution.


Set the axis of revolution by the method selected at step 4).

- “Input” : If the method is set as “Input”, the line tool button  is automatically activated, and the cursor changes into  $+$  shape. Input the axis of revolution following the same procedure as that of creating a straight line.
- “Select” : If the method is set as “Select”, the curve selection tool is activated, and thus cursor changes into  shape. Select a straight line which will be used as the axis of revolution.
- “Last” : The axis of revolution for last mesh generation is applied again. So, it is not necessary to input or select the axis. This option can be used only when this method of mesh generation was applied at least once since the program started.

*The axis of revolution may meet one end of the seed curve, but should not interfere with the seed curve. Otherwise, a self intruding mesh may be generated. However, in case the axis meets one end of the seed curve, a continuous surface mesh is constructed near the axis. Even if the element type is set as quadrilateral, triangular elements will be generated around the axis*

11) Click **Set Axis** button.

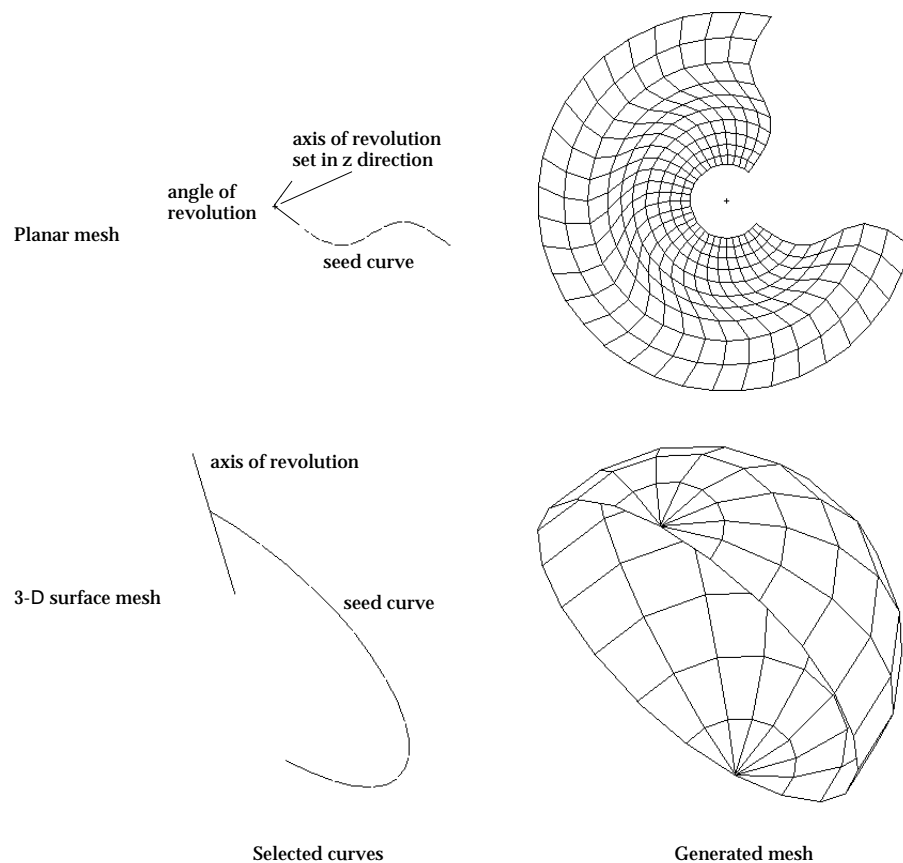
A mesh is generated by revolving the seed curves about the axis of revolution.

**Set Axis** button is restored to **Set Curve** button. The program is now ready for generating another surface mesh. The curve selection tool  is automatically activated, if it is not in action.

**Reverse** button is enabled, only when mesh generation is successful.

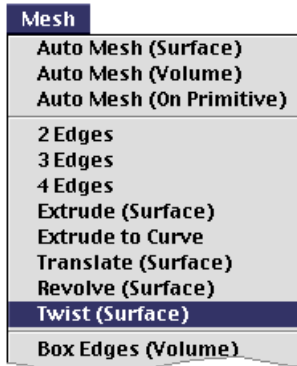
Clicking **Reverse** button reverses the sign of the revolution angle and regenerates the mesh.

You may repeat the above procedure of mesh generation without issuing the command again, while “Revolve Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< Example of mesh generation by revolution in a plane and in 3-D space >

## ■ Generating mesh by twisting




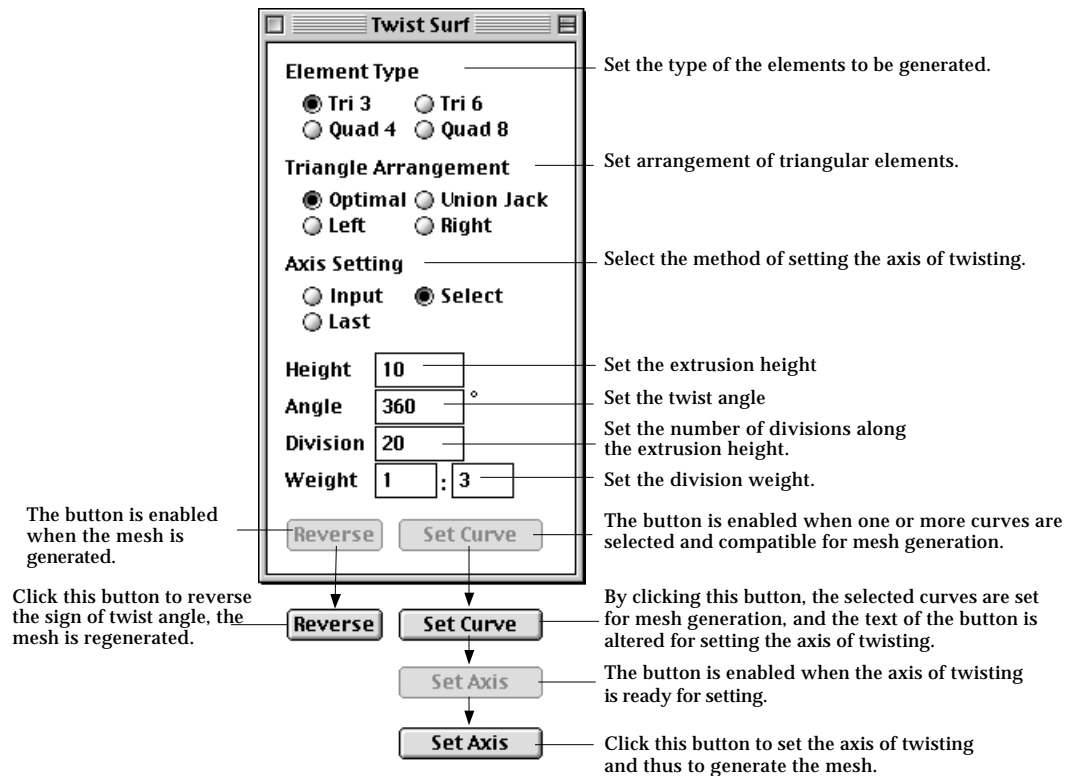
Twisting is a method of mesh generation by extrusion combined with revolution. So, it may be termed as “twisted extrusion.” Elements are generated on the surface which is formed by extruding the selected seed curves along the specified axis while twisting the direction of extrusion about the axis by the specified angle. Along this twisted extrusion, the trace of a node on the seed curve makes a helix on which new nodes are created.

If the twist angle is set to zero, the resulting mesh generation is identical to that of extrusion in the direction of the twist axis. Likewise, if the extrusion height is set to zero, a surface mesh of revolution is obtained.

This method of mesh generation is useful in modeling curved surfaces of spiral shape.

- 1) Choose “Twist (Surface)” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Twist Surf” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog.

- 3) Select the type of arrangement for triangular elements.

Triangular elements can be generated in the 4 different types of arrangement as explained in the previous section “Generating mesh using 2 edges”.

- 4) Select the option for setting the twist axis.

Choose one of the 3 options setting the twist axis, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is enabled only when this method of mesh generation was applied at least once since the program started.

- 5) Set the extrusion height.

Insert the extrusion height in the dialog box. The direction of extrusion is given by that of the twist axis which is directed from the first input point to the second.

- 6) Set the twist angle.

Insert the twist angle in the dialog box. There is no limit in the acceptable range of twist angle. The negative sign reverses the direction of twist.

- 7) Set the number of divisions.

Specify how many rows of elements are to be generated in the direction of the twisted extrusion.

- 8) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of twisting.

- 9) Select curves forming seed curves for twisting.



All the selected curves should be divided, and form an edge, which may be one curve or serially connected curves. The seed curves may be open or closed.  button is enabled when an edge is formed properly for mesh generation.

- 10) Click  button.

The selected curves are reserved as the seed curves for mesh generation, and the button changes into  indicating that setting the twist axis is expected in the next step.


- 11) Set the twist axis.

Set the twist axis by the method selected at step 4).

- “Input “ : If the method is set as “Input”, the line tool button  is automatically activated, and the cursor changes into + shape. Input the twist axis following the same procedure as that of creating a straight line.
- “Select” : If the method is set as “Select”, the curve selection tool is activated, and thus cursor changes into  shape. Select a straight line which will be used as the twist axis.
- “Last” : The twist axis for last mesh generation is applied again. So, it is not necessary to input or select the axis. This option can be used only when this method of mesh generation was applied at least once since the

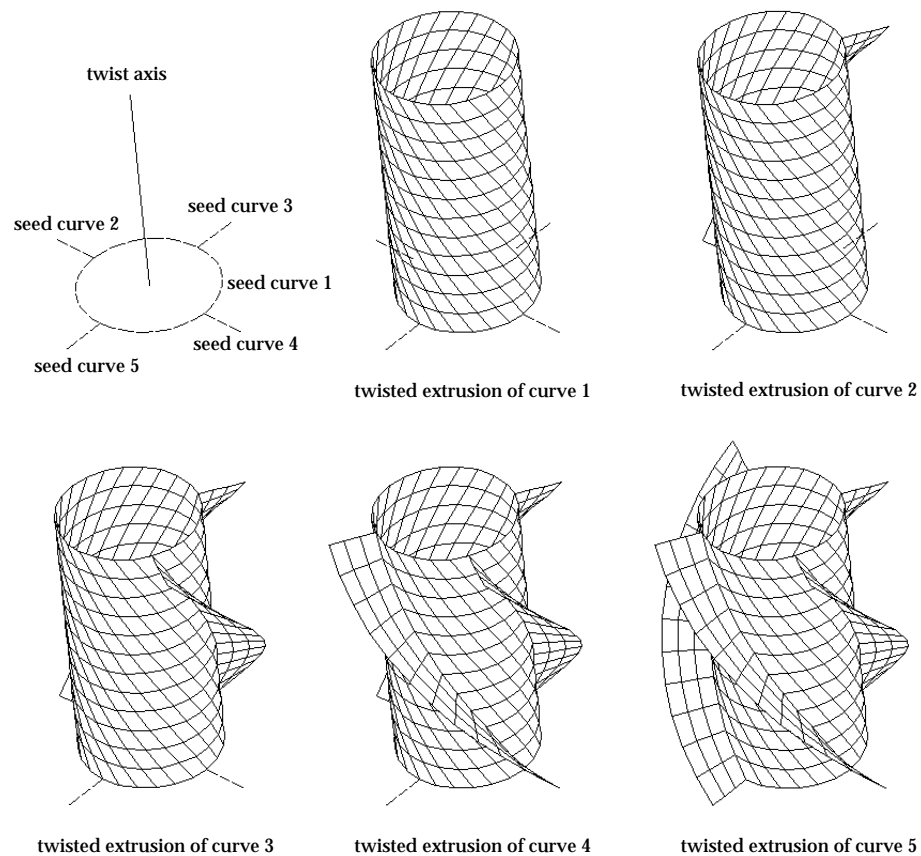
program started.

- 12) Click  button.

A mesh is generated by twisted extrusion of the seed curve.  button is restored to  button. The program is now ready for generating another surface mesh. The curve selection tool  is automatically activated, if it is not yet activated.

button is enabled, only when mesh generation is successful. Clicking  button reverts sign of the twist angle and regenerates the mesh.


You may repeat the above procedure of mesh generation without issuing the command again, while “Twist Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



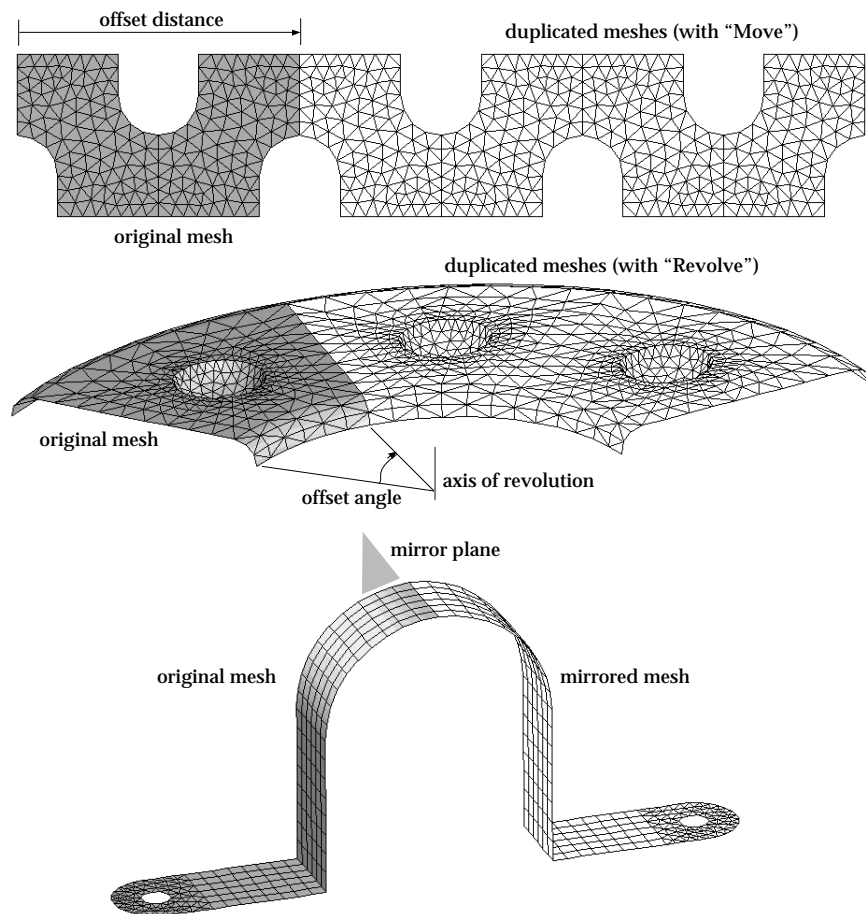
< Example of modeling 3-D surfaces by twisting >

## Duplicating surface meshes

New surface meshes may be created by duplicating existing meshes. The position and orientation of the duplicated meshes are determined by moving, revolving or mirroring coordinates of original meshes with specified offsets. The curves belonging to the surface mesh are automatically duplicated, and annexed to the host surface mesh.

The procedure of duplicating surface meshes is the same as that of duplicating curves. In order to start duplicating surface meshes, click surface selection tool  if it is not in action. Then, select one of the submenu items for duplications: “Move”, “Revolve” or “Mirror”. Further details of duplication procedure are explained in Chapter 3, and are not repeated here.

There is no limit in the number of duplicates which can be made at once. You can specify the number of duplications using the dialog box for the corresponding duplication method. The following figure shows examples of multiplying surface meshes by duplicating existing ones with moving, revolving or mirroring.




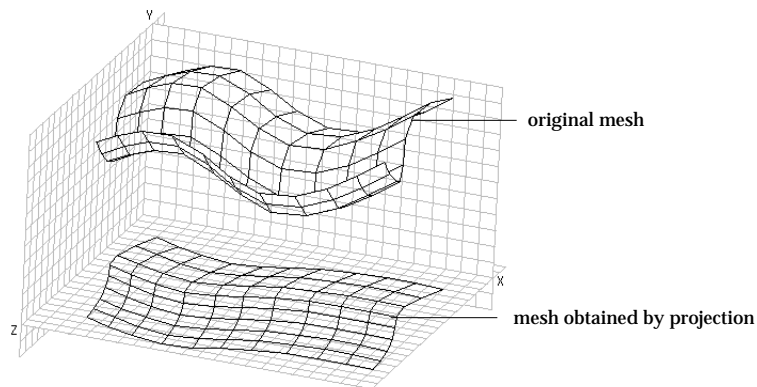
< Examples of duplicated surface meshes >



## Projecting surface meshes

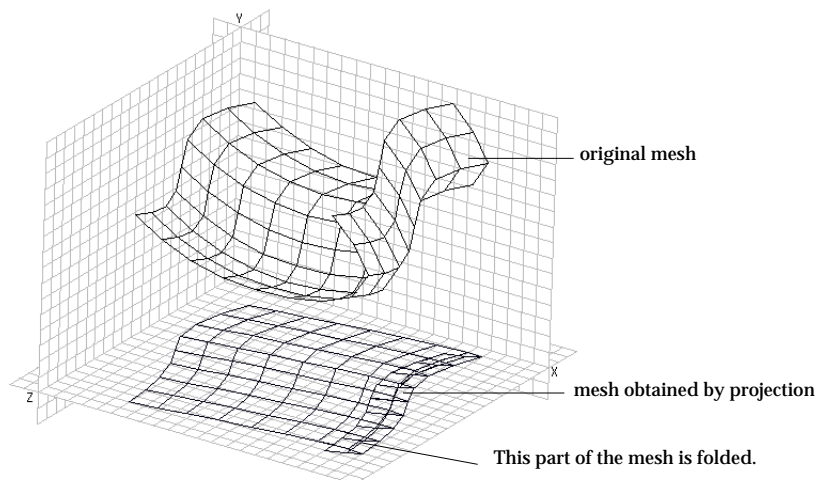
New surface meshes may be created by projecting existing meshes on the specified grid plane.

The procedure of projecting surface meshes is the same as that of projecting curves. In order to start projecting surface meshes, click surface selection tool  if it is not in action. Then, select one of the **Project** submenu items: "On XY", "On YZ" and "On ZX". Prior to issuing this projection command, the grid plane should be moved to the desired position. Further details of projection procedure are explained in Chapter 3, and are not repeated here. The following figure shows an example of a surface mesh projected on a grid plane.



< Example of surface mesh projection >

You should be careful not to generate folded surface meshes which may be obtained by faulty projection. The original meshes for projection should be uniquely mapped with the projection plane. Otherwise, folded surface meshes will be created. The following example shows an example of a faulty projection.



< Example of a faulty projection >

## Volume Mesh Generation

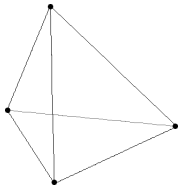
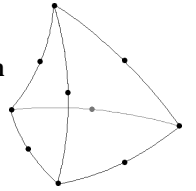
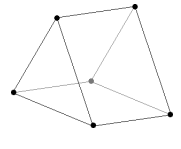

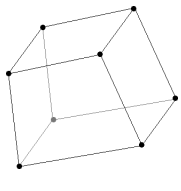
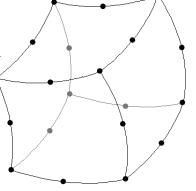
For 3-dimensional solid structural or heat conduction models, the analysis regions are modeled by volume meshes. The solid elements may be tetrahedra, prisms or hexahedra. VisualFEA provides a number of methods for generating volume meshes. They can be classified into the following 3 categories:

- automatic tetrahedronization
- mapping
- sweeping
- duplication

There are a few methods in each category, as described in the following sections. The generated volume meshes are always represented only by their surfaces, and their internal meshes are not displayed. The intent is to relieve the complexity of the wireframe representing the mesh. But, the geometry of each element can be displayed individually by a number-related function, which is explained in “Object Selection” section of Chapter 2.

VisualFEA supports 6 types of solid elements, which may be used in volume meshes. In case of surface mesh generation, the element type is always selectable from all the available types. For a volume mesh, however, selectable types are limited depending on the method of mesh generation. The types of solid elements are shown in the following table.

< Element types in volume meshes >

<p>4 node tetrahedron</p> 	<p>10 node tetrahedron</p> 
<p>6 node prism</p> 	<p>15 node prism</p> 
<p>8 node hexahedron</p> 	<p>20 node hexahedron</p> 

## Volume mesh generation by automatic tetrahedronization

Automatic tetrahedronization is one of the most advanced technique generating 3-dimensional volume meshes. This method is analogous to automatic triangulation for surface mesh generation. Automatic tetrahedronization fills a volume surrounded by boundary surface meshes, while automatic triangulation fills an area surrounded by boundary curves. The boundary surface meshes should form a closed surface enclosing a 3-dimensional space. A volume mesh is generated by an algorithm subdividing the volume space recursively along the surface element boundaries. A volume element is created at each end of the recursion.

This method is very powerful and useful, because a volume mesh can be generated, with minimum user interaction, in a relatively arbitrarily shaped space. There is no requirement for convexity of the space.


Only 4 node and 10 node tetrahedron elements can be generated by tetrahedronization in the current version of VisualFEA.

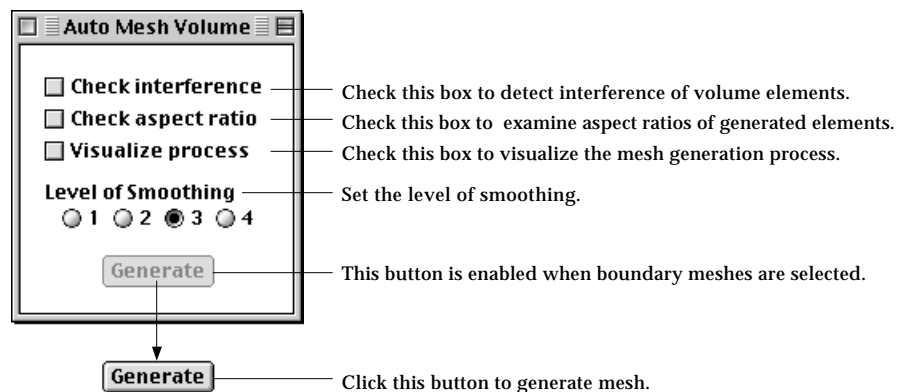
### ■ Generating mesh by automatic tetrahedronization



A volume mesh can be generated by automatic tetrahedronization technique in a 3-dimensional space surrounded by closed surface meshes. The element shape generated by automatic tetrahedronization is limited to tetrahedron in the current version of VisualFEA. Either 4 or 10 node elements can be generated. If elements within the boundary meshes are 3-node triangles, 4-node tetrahedral elements will be generated. If elements within the boundary meshes are 6-node triangles, 10-node tetrahedral elements will be generated.

- 1) Choose “Auto Mesh (Volume)” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Auto Mesh Volume” dialog box appears.



- 2) Set options for mesh generation process.
  - “Check interference” : If this option is checked, interference between elements is detected during mesh generation.
  - “Check aspect ratio” : The aspect ratios of generated elements are examined, and reported if they are not acceptable.
  - “Visualize process” : The recursive process of mesh generation is graphically visualized.

- 3) Set the level of smoothing.

The shape of the individual element is polished through smoothing process. The level of smoothing is defined in four grades, 1, 2, 3 and 4. Grade 4 takes longest time, but produces the best shaped elements.

- 4) Select surface meshes forming closed boundary.

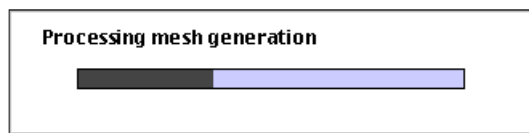
The selected meshes should form a closed surface boundaries. All the elements in the meshes should be triangular and have the same number of nodes. That is, they should consists of 3-node triangular elements only, or 6-node triangular elements only.  button is enabled when acceptable boundary meshes are formed.

- 5) Click  button.

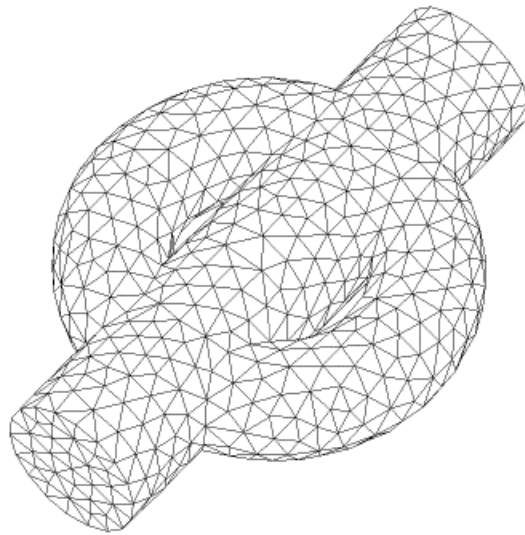
A volume mesh is generated by automatic tetrahedronization, if the selected surface meshes are acceptable for automatic tetrahedronization as described in the next section. Otherwise, the action is ignored with a message notifying that automatic tetrahedronization cannot not be completed.

In the above procedure, the order of step 2), 3) and 4) can be interchanged. You may repeat the above procedure of generating a mesh by automatic tetrahedronization without issuing the command again, while “Auto Mesh Volume” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.

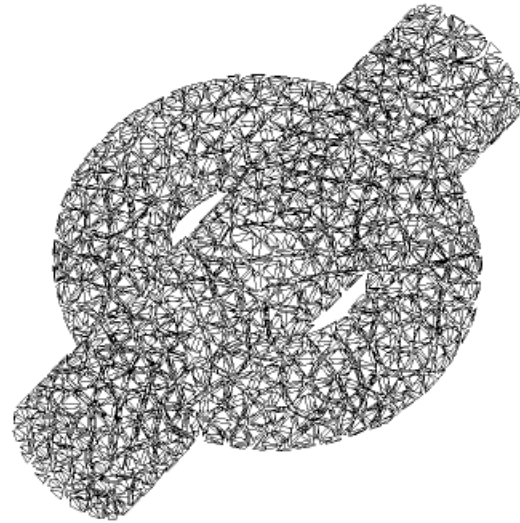
Automatic tetrahedronization usually takes a long time. While mesh generation is progressing, its status is indicated by a progress bar.



The progress bar is not to the actual time scale. The display indicates only the progress in the recursion stages. If “Visualize process” box is checked, the recursive process of mesh generation is also graphically visualized.



Boundary surface meshes



Volume mesh generated by automatic tetrahedronization (exploded view)

< Example of volume mesh generation by automatic tetrahedronization >

### ■ Forming boundary meshes for automatic tetrahedronization

For successful automatic tetrahedronization, surface meshes should be selected so that they form a boundary surfaces acceptable for mesh generation as described below.

- The selected meshes should form a closed boundary surface enclosing the 3-dimensional region in which new volume mesh is to be generated.
- The region may be either convex or concave, but cannot contain another volume (pocket).
- All the elements of the boundary meshes should be triangular.
- All the elements of the boundary meshes should have equal number of nodes. That is, the number of nodes in every element should be uniformly either 3 or 4.

The automatic tetrahedronization will be aborted if the above conditions are not satisfied. The mesh generation may also fail if the 3-dimensional space is too complex. In this case, the space should be broken into two or more sub-regions which are simpler, and the automatic tetrahedronization should be applied for each of these sub-regions.

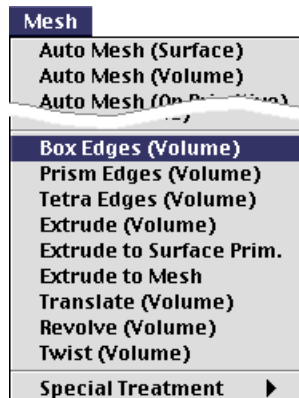
## Volume mesh generation by mapping

An analysis region may be represented by its boundary edges. Volume meshes are generated to fill the region formed by the boundary edges. VisualFEA supports following 3 types of edge formation for volume mesh generation:

- box edges : 12 edges forming the boundaries of a box shaped region
- prism edges : 9 edges forming the boundaries of a prism shaped region
- tetra edges : 6 edges forming the boundaries of a tetrahedral region


The “triangular mapping” and “transfinite mapping” techniques applied to surface mesh generation are also extended for volume mesh generation in 3-dimensional regions defined by the above 3 types of edge formation. The new nodal points in the generated mesh are computed by these mapping techniques. The mesh generation commands are provided as menu items in **Mesh** menu.

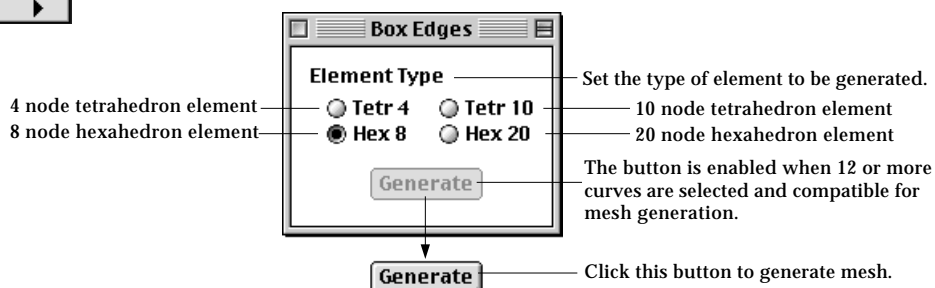
### ■ Generating mesh using box edges



A volume mesh can be generated using transfinite mapping technique in a box-shaped region bounded by 12 edges, each of which consists of a curve or serially connected curves. The shape of the elements within the mesh may be chosen from tetrahedron and hexahedron. Prism elements are not included because they may cause incompatibility between adjacent mesh regions.

- 1) Choose “Box Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Box Edges” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog. “Tetra 4”, “Tetra 10”, “Hex 8” and “Hex 20” represent elements of 4 node tetrahedron, 10 node tetrahedron, 8 node hexahedron and 20 node hexahedron respectively.

- 3) Select curves forming box edges.

All the selected curves should be divided, and form compatible box edges.

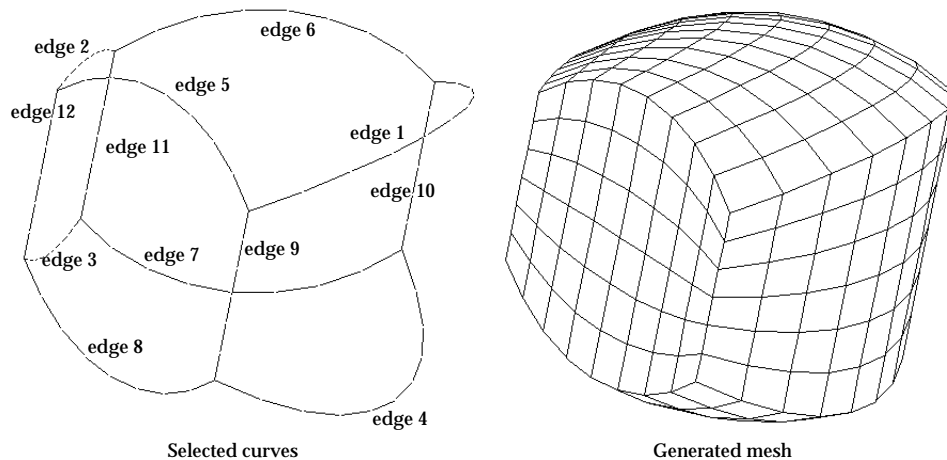
**Generate** button is enabled when more than equal to 12 separate curves are

selected and box edges are formed by them. The number of divisions in each set of parallel edges should be equal.

- 4) Click **Generate** button.

A volume mesh is generated within the box edge region, if the selected curves are compatible for mesh generation. Otherwise, the action is ignored after a message “Incompatible curve selection for box edges.”

You may repeat the above procedure of generating mesh using 12 edges without issuing the command again, while “Box Edges” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< Example of mesh generation in a box edge region >

#### ■ Setting box edges compatible for mesh generation

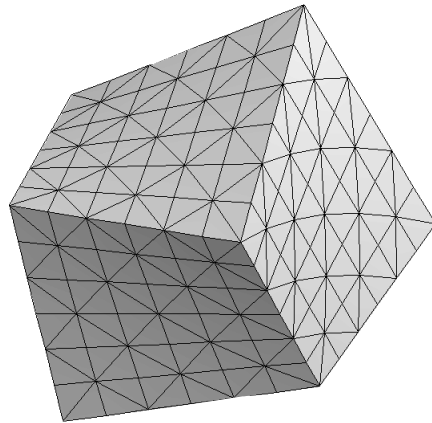
For successful mesh generation by this method, curves should be selected so that they may form 12 edges compatible for box edge mesh generation as described below. When **Generate** button is clicked at step 4 in the above procedure, VisualFEA checks if selected curves fulfill these conditions. When the curves selections are acceptable, the mesh generation procedure is processed.

- The selected curves should be connected serially in 12 groups. This grouping is automatically done by VisualFEA. Each of the 12 groups forms an edge. Each edge may consist of one or more curves.
- The 12 edges must have configurations equivalent to edges of a box. For this requirement, 3 edges must meet at each of the 8 corner points of the box. There should be no edges with open ends. In other words, each edge should be connected to other edges at both ends.
- The number of nodes on each group of parallel edges must be equal. Edges 1, 2, 3 and 4 are parallel in the above example. Likewise, edges 5, 6, 7 and 8,

and edges 9, 10, 11 and 12 are parallel edges respectively.

- For element types “Tetra 4” and “Hexa 20”, each edge should be divided in multiples of 2. And for “Tetra 10”, each edge should be divided in multiples of 4.

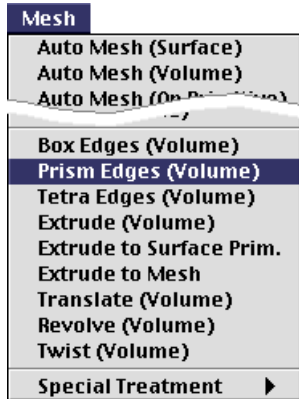
*For compatibility of element boundaries between adjacent mesh regions, the tetrahedron elements of a box edge region are generated so that triangular faces in the outer surfaces should have Union Jack formation as shown in the figure below. Accordingly, each of the 12 edges should be divided into multiples of 2 in order to generate “Tetra 4”, and into multiples of 4 in order to generate “Tetra 10”. If this condition does not meet when you try generating these types of elements, the action will not be processed.*



<Formation of Union Jack on surfaces of a box edge volume mesh>




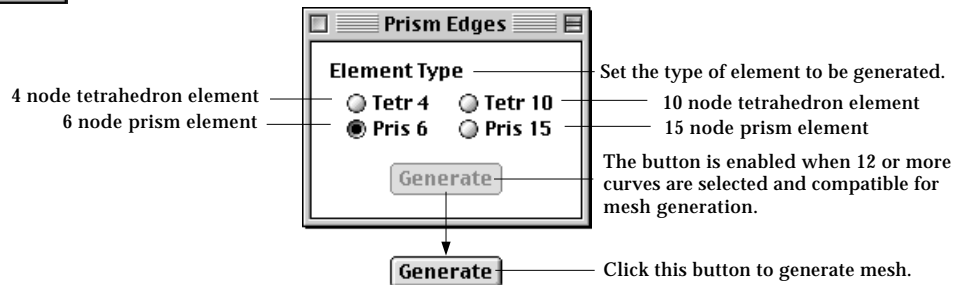
### ■ Generating mesh using prism edges



A volume mesh can be generated in a prism-shaped region bounded by 9 edges, each of which consists of a curve or serially connected curves. The shape of the elements within the mesh may be chosen from tetrahedron and prism. For mesh generation, transfinite mapping is applied in the longitudinal direction of the prism and triangular mapping on the triangular sections.

- 1) Choose “Prism Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Prism Edges” dialog box appears.



- 2) Set the element type.

Choose one of the 4 element types given as radio buttons in the dialog. “Tetra 4”, “Tetra 10”, “Pris 6” and “Pris 15” represent elements of 4 node tetrahedron, 10 node tetrahedron, 6 node prism and 15 node prism respectively.

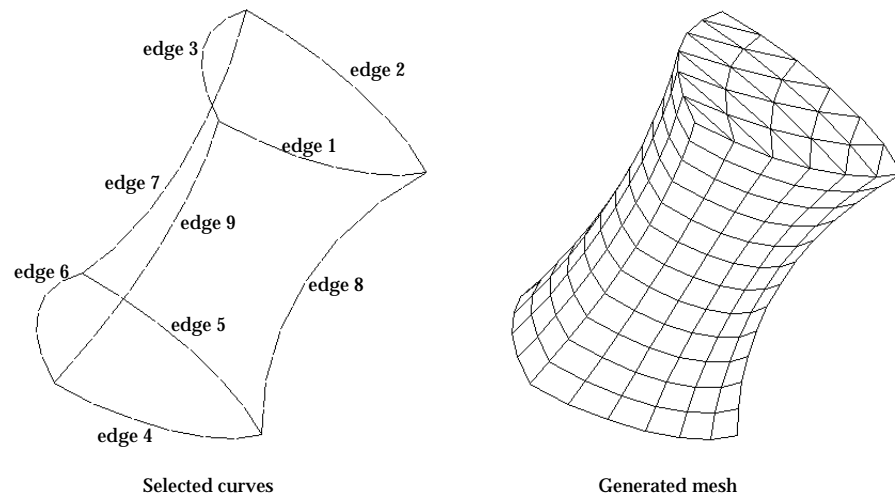
- 3) Select curves forming prism edges.

All the selected curves should be divided, and form compatible prism edges. **Generate** button is enabled when more than or equal to 9 separate curves are selected and a prism edges are formed by them. The number of divisions of should be acceptable for triangular mapping or transfinite mapping.

- 4) Click **Generate** button.

A volume mesh is generated within the prism edge region, if the selected curves are compatible for mesh generation. Otherwise, the action is ignored after a message “Incompatible curve selection for prism edges.”

You may repeat the above procedure of generating mesh using 9 edges without issuing the command again, while “Box Edges” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< Example of mesh generation in a prism edge region >

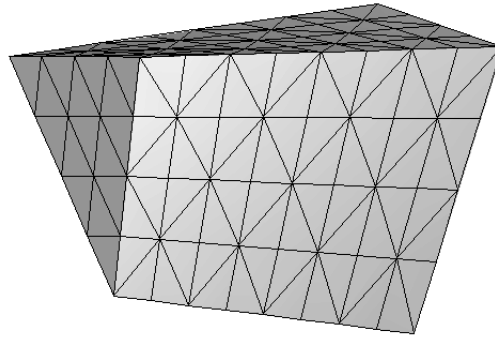
### ■ Setting prism edges compatible for mesh generation

For successful mesh generation by this method, curves should be selected so that they may form 9 edges compatible for prism edge mesh generation as described below. When **Generate** button is clicked at step 4 in the above procedure, VisualFEA checks if selected curves fulfill these conditions. Only when the curves selections are acceptable, the mesh generation procedure is processed.

- The selected curves should be connected serially in 9 groups. This grouping is automatically done by VisualFEA. Each of the 9 groups forms an edge. Each edge may consist of one or more curves.
- The 9 edges must have configurations equivalent to edges of a prism. For this requirement, 3 edges must meet at each of the 6 corner points of the box. There should be no edges with open ends.
- The number of nodes on the edges forming triangular faces must be equal. Edge 1, 2 and 3 forms a triangular face, and edge 4, 5 and 6 form another. The number of nodes on the edges forming longitudinal faces should also be equal. That is, the number of nodes on edge 7, 8 and 9 must not be different.
- For element types “Tetra 4” and “Prism 15”, each edge should be divided in multiples of 2. And for “Tetra 10”, each edge should be divided in multiples of 4.

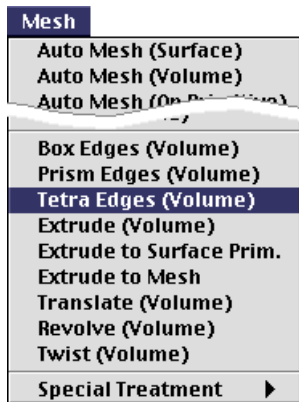
*For compatibility of element boundaries between adjacent mesh regions, the tetrahedron elements of a prism edge region are generated so that triangular faces in the longitudinal surfaces should have Union Jack formation as shown in the figure below. Accordingly, each of the 9 edges should be divided into multiples of 2 in order to generate “Tetra 4”, and into multiples of 4 in order to generate “Tetra 10.”*

If this condition does not meet when you try generating these types of elements, the action will not be processed.




<Formation of Union Jack on surfaces of a prism edge mesh>

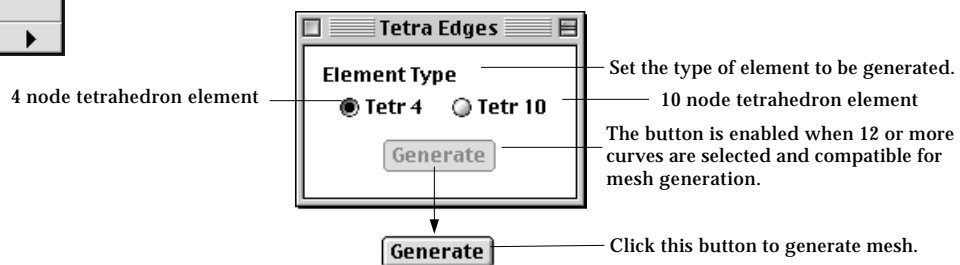
### ■ Generating mesh using tetrahedron edges



A volume mesh can be generated in a tetrahedron-shaped region bounded by 6 edges, each of which consists of a curve or serially connected curves. The shape of the elements generated by this method is always tetrahedron. For mesh generation, triangular mapping is applied on every triangular section of the tetrahedron.

- 1) Choose “Tetra Edges” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Tetra Edges” dialog box appears.



- 2) Set the element type.

Choose one of the 2 element types given as radio buttons in the dialog. “Tetra 4” and “Tetra 10” represent element shape that are 4 and 10 node tetrahedron, respectively.

- 3) Select curves forming tetra edges.

All the selected curves should be divided, and form compatible tetra edges.

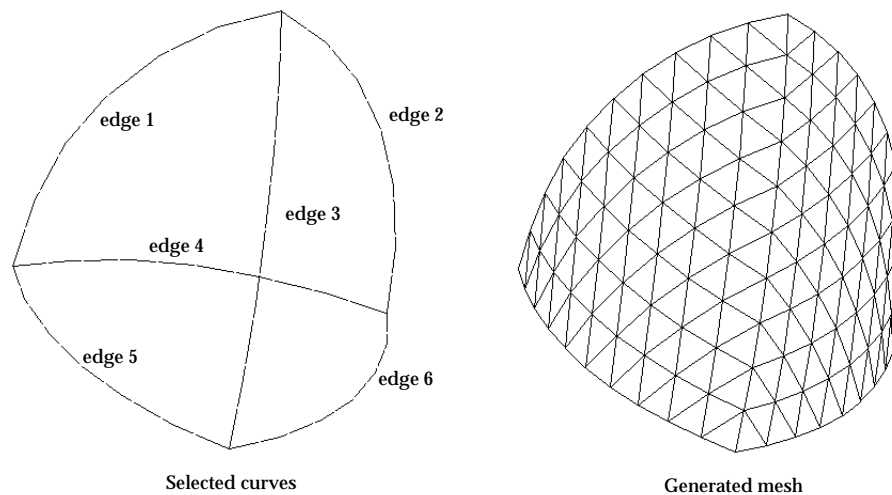
**Generate** button is enabled when more than equal to 6 separate curves are selected and a tetra edges are form by them. The number of divisions on all

the edges should be equal.

- 4) Click **Generate** button.

A volume mesh is generated within the tetra edge region, if the selected curves are compatible for mesh generation. Otherwise, the action is ignored after a message “Incompatible curve selection for tetra edges.”

You may repeat the above procedure of generating a mesh using 6 edges without issuing the command again, while “Tetra Edges” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.



< Example of mesh generation in a tetra edge region >

### ■ Setting tetra edges compatible for mesh generation

For successful mesh generation by this method, curves should be selected so that they may form 9 edges compatible for tetra edge mesh generation as described below. When **Generate** button is clicked at step 4 in the above procedure, VisualFEA checks if selected curves fulfill these conditions. Only when the curves selections are acceptable is the mesh generation procedure processed.

- The selected curves should be connected serially in 6 groups. This grouping is automatically done by VisualFEA. Each of the 6 groups forms an edge. Each edge may consist of one or more curves.
- The 6 edges must have configurations equivalent to edges of a tetrahedron. For this requirement, 3 edges must meet at each of the 4 corner points of the box. There should be no edges with open ends.
- The number of nodes on all the edges forming triangular faces must be equal.
- For element types “Tetra 10”, each edge should be divided in multiples of 2.

## Volume mesh generation by sweeping operations

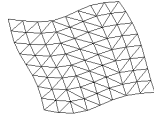
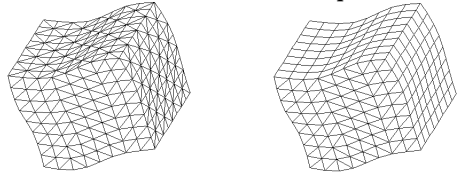
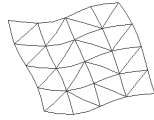
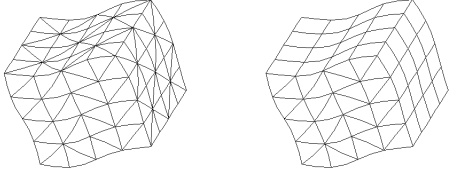
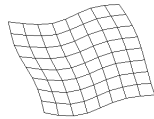
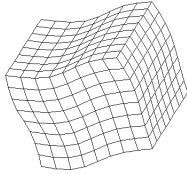
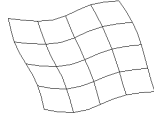
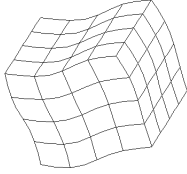
VisualFEA supports the following 4 types of sweeping operations for volume mesh generation:

- Extrusion
- Translation
- Revolution
- Twisting

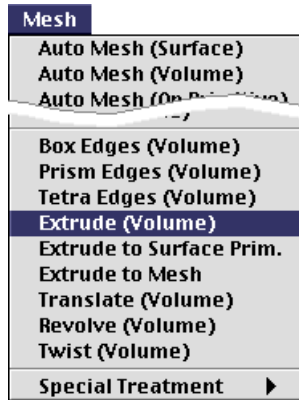
Sweeping operations for volume mesh generation are analogous to those for surface mesh generation explained in the previous section. Refer to “Surface mesh generation by sweeping operations.” For volume mesh generation, sweeping operations are based on selected surfaces, i.e. seed surfaces while surface mesh generation is based on seed curves. The commands for volume mesh generation by sweeping are provided as menu items in **Mesh** menu.

In case of volume mesh generation by sweeping, the type of element is determined by the type of element on the seed surfaces as shown below. Therefore, it is not necessary to set the type of the element.

< Element types in the seed mesh and in the generated volume mesh >


Seed surface mesh	Generated mesh
<b>3 node triangle</b> 	<b>4 node tetrahedron or 6 node prism</b> 
<b>6 node triangle</b> 	<b>10 node tetrahedron or 15 node prism</b> 
<b>4 node quadrilateral</b> 	<b>8 node hexahedron</b> 
<b>8 node quadrilateral</b> 	<b>20 node hexahedron</b> 

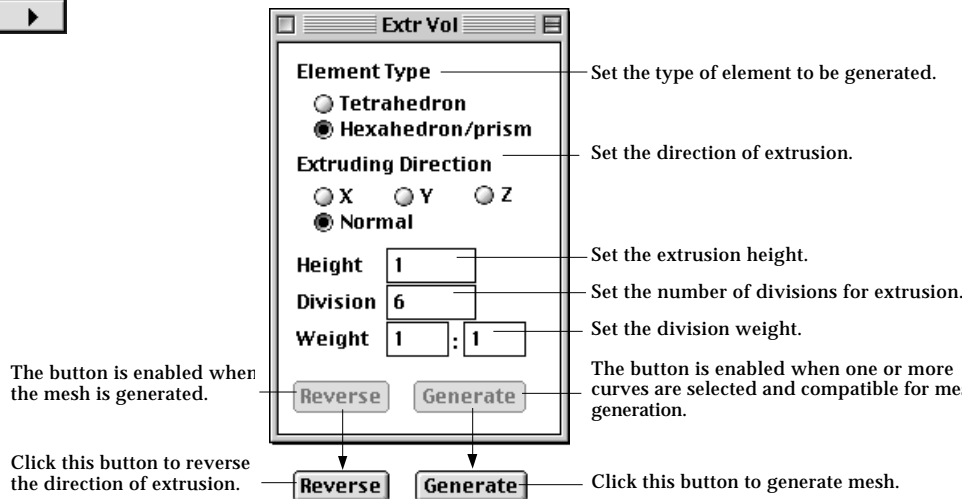
### ■ Generating mesh by extrusion



A volume mesh can be generated by extruding the seed surfaces up to the specified height and in the specified direction. The height of the extrusion is entered using the “Extrude” dialog. This is the simplest but most frequently used method of generating volume meshes. There are three methods generating a volume mesh by extruding seed surfaces. The other 2 methods are explained in the following sections.

- 1) Choose “Extrude(Volume)” from **Mesh** menu.

The curve selection tool  is automatically activated, and “Extr Vol” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Set the direction of extrusion.

The mesh may be extruded either in the direction of a coordinate axis, or in the direction normal to the seed surface. The normal direction is determined independently at each of the extruding nodes on the seed surface. If more than one surfaces are joined at a node and their slopes are not continuous, there exists more than one normals at the node. In this case, the normal direction is evaluated by averaging the multiple normal directions.

- 4) Enter the height of extrusion.

Extrusion height is the distance from the seed surface to the extent of the generated volume mesh. If extrusion direction is set as “normal”, the extrusion height designates the uniform thickness of the generated volume mesh.

- 5) Enter the number of divisions for extrusion.

Specify how many layers of elements to be generated by extrusion. If the element type is 15 node prism or 20 node hexahedron, the number of element layers is half the number of divisions, because 2 divisions are required to make one layer of the elements.

- 6) Enter the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between thickness of elements at the starting part,  $l_1$  and at the ending part of extrusion,  $l_n$ .

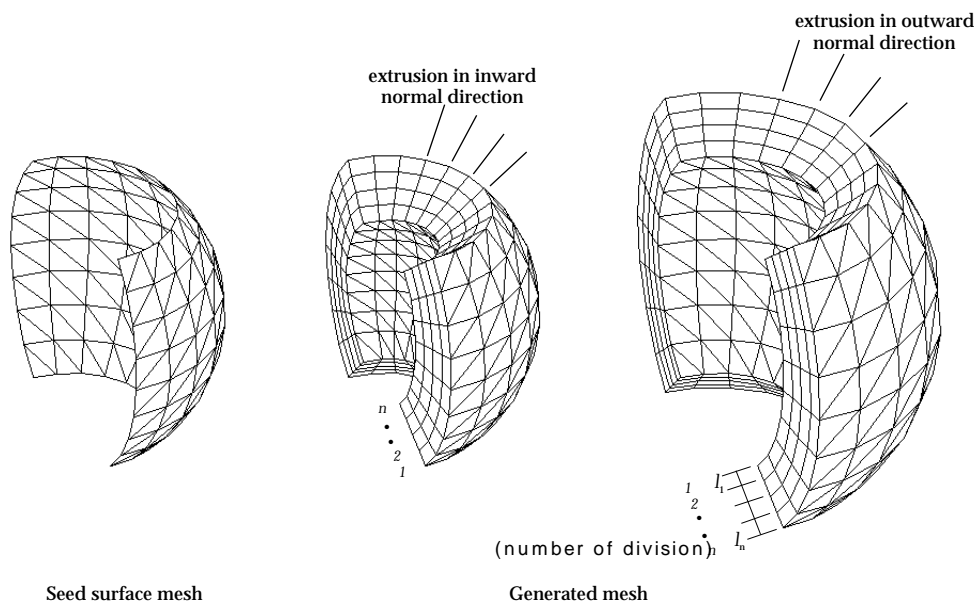
- 7) Select the seed surfaces for extrusion.

Select surface meshes forming seed surfaces for mesh generation. **Generate** button is enabled when more than one or more surfaces are selected. More than one surface may be used as the seed surfaces. In this case, the seed surfaces should be connected so that their extrusion may generate a continuous volume.

- 8) Click **Generate** button.

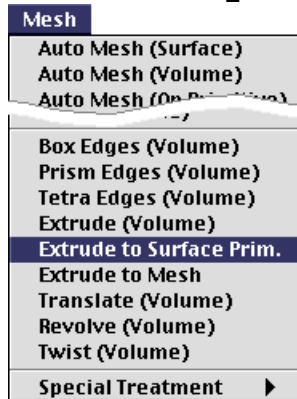
A volume mesh is generated by extruding the seed surfaces. If mesh generation is successful, **Reverse** button is enabled. In case the mesh is generated opposite to the desired direction, click **Reverse** button to reverse the direction of extrusion. Then, mesh will be regenerated with the reverse direction.

You may repeat the above procedure of generating mesh by extrusion without issuing the command again, while “Extr Surf” dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.




< A volume mesh generated by extrusion in normal direction >

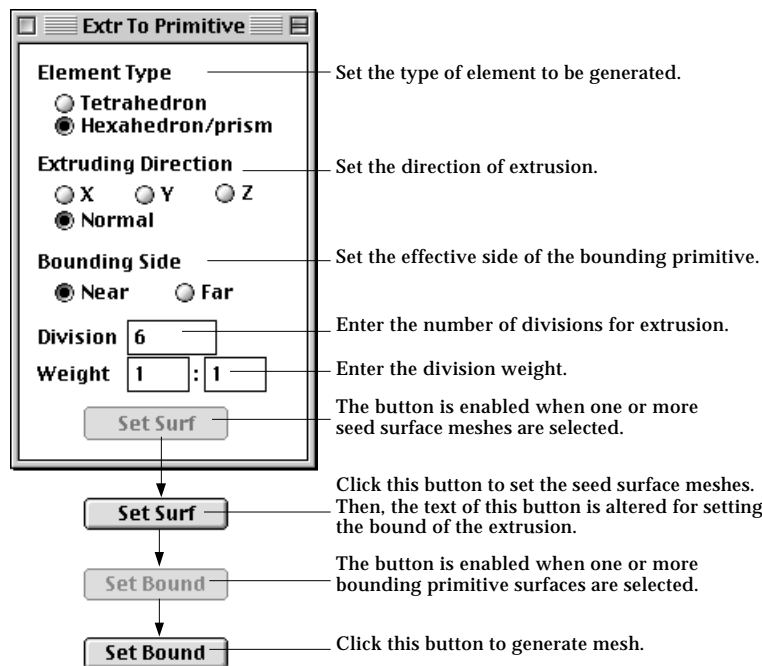
### ■ Generating mesh by extrusion up to bounding surface primitives



The bound of the extrusion may be specified by selected surface primitives instead of specifying the height of extrusion. These surface primitives are termed here as “bounding primitives.” This method of mesh generation is the same as the above described extrusion method except that the extent of extrusion is determined not by its height but by the bounding primitives. In generating a volume mesh by this method, you can control the direction of extrusion as well as the boundary of the mesh. Thus, a volume mesh of desired shape can be constructed with relative ease.

- 1) Choose “Extrude to Surface Prim.” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Extr to Primitive” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Set the direction of extrusion.

The mesh may be extruded either in the direction of a coordinate axis, or in the direction normal to the seed surface. The direction is always signed toward the bounding surface primitive.

- 4) Set the side of the bound.

There are usually more than two sides of the bounding surfaces applicable as



the boundary of the generated mesh. Therefore, it is necessary to specify which side of the bounding primitives should be applied.

- 5) Set the number of divisions for extrusion.

Specify how many layers of elements to be generated by extrusion.


- 6) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between thickness of elements at the starting part,  $l_1$  and at the ending part of extrusion,  $l_n$ .

- 7) Select surface meshes forming seed surfaces for extrusion.

button is enabled when one or more surface meshes are selected.


- 8) Click  button.

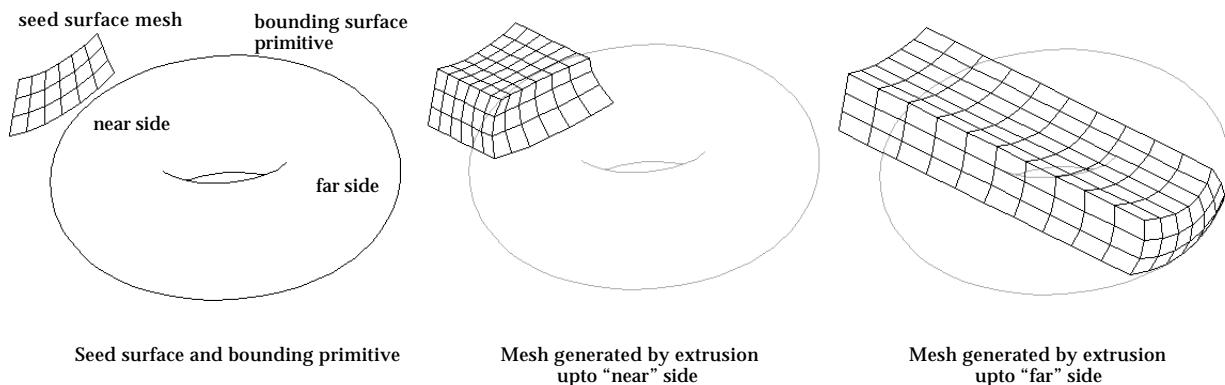
The selected surface meshes are reserved as the seed surfaces for mesh generation, and the button changes into  indicating that selection of bounding surface primitives is expected in the next step. At the same time, the primitive selection tool  is automatically activated.

- 9) Select the bounding surface primitives.

The selected bounding primitives are highlighted in bright red color. The dimmed  button is enabled when bounding primitives are selected.

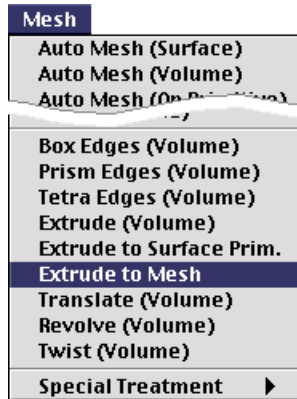
- 10) Click  button.

A mesh is generated by extruding the seed meshes up to the bounding primitives.  button is restored to  button, and the surface selection tool  is activated. The program is now ready for generating another volume mesh. The bounding primitives must be wide enough to cover the whole front of the extrusion. Otherwise, the mesh generation process will be aborted with a message "The bounding primitives cannot cover the range of the extrusion."




< Example of mesh generation by extrusion to a surface primitive >

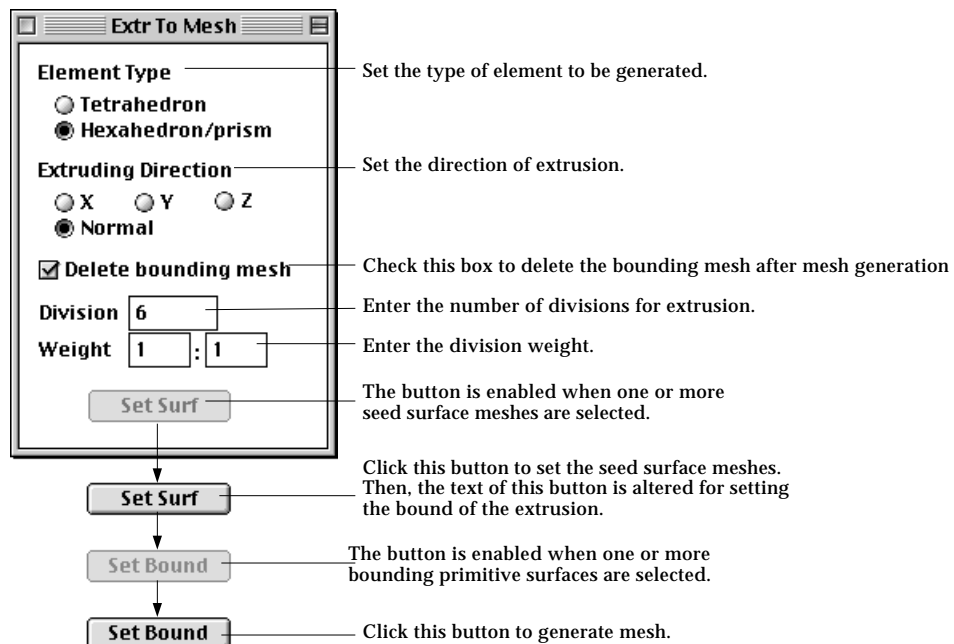
### ■ Generating mesh by extrusion up to surface meshes



The bound of the extrusion may be specified by surface meshes. This method of mesh generation is useful when the bound of extrusion is a free-form surface which cannot be expressed by surface primitives supported in VisualFEA. These surface meshes are termed here as “bounding mesh.” This method of mesh generation is the same as the above described method of “extrusion up to surface primitives” except that surface meshes instead of bounding primitives are used as the bounding limit. In order to apply this method, the bounding surface mesh should be generated before starting the process of mesh generation.

- 1) Choose “Extrude to Mesh” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Extr to Mesh” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Set the direction of extrusion.

The mesh may be extruded either in the direction of a coordinate axis, or in the direction normal to the seed surface. The direction is always signed toward the bounding surface meshes.

- 4) Specify whether to delete the bounding mesh automatically after mesh generation.

In most cases, the only usage of the bounding mesh is to generate the volume mesh by this method. Then it is desirable to delete the bounding meshes after the mesh generation. If “Delete bounding mesh” box is checked, the bounding mesh will be deleted automatically when the mesh is generated. Otherwise, the bounding meshes remain undeleted after mesh generation.

- 5) Set the number of divisions for extrusion.

Specify how many layers of elements to be generated by extrusion.


- 6) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between thickness of elements at the starting part,  $l_1$  and at the ending part of extrusion,  $l_n$ .

- 7) Select surface meshes forming seed surfaces for extrusion.

button is enabled when one or more surface meshes are selected.

- 8) Click  button.


The selected surface meshes are reserved as the seed surfaces for mesh generation, and the button changes into  indicating that selection of bounding surface meshes is expected in the next step. At the same time, the mesh selection tool  is automatically activated.

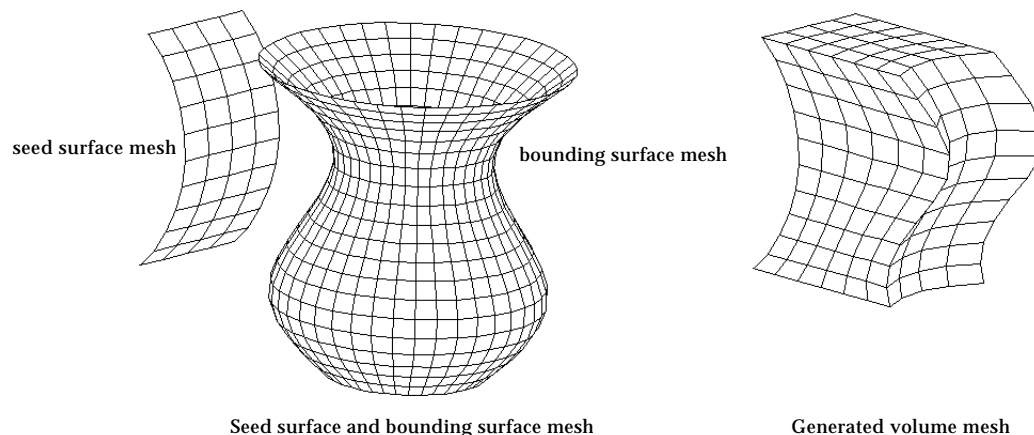
- 9) Select the bounding surface meshes.

The selected bounding meshes are highlighted in bright red color. The dimmed  button is enabled when bounding meshes are selected.

If the bounding surface is flat, its density does not affect the result of the mesh generation. But, in case the bounding surface is curved, the smoothness of generated mesh may be affected by its density. Higher density is required in order to represent curved surfaces more accurately.

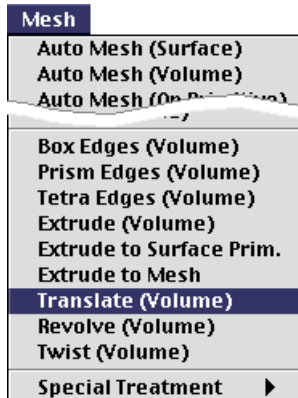
- 10) Click  button.

A volume mesh is generated by extruding the seed meshes up to the bounding meshes.  button is restored to  button, and the surface selection tool  is activated.



< Example of mesh generation by extrusion to a surface mesh >


### ■ Generating volume mesh by translation

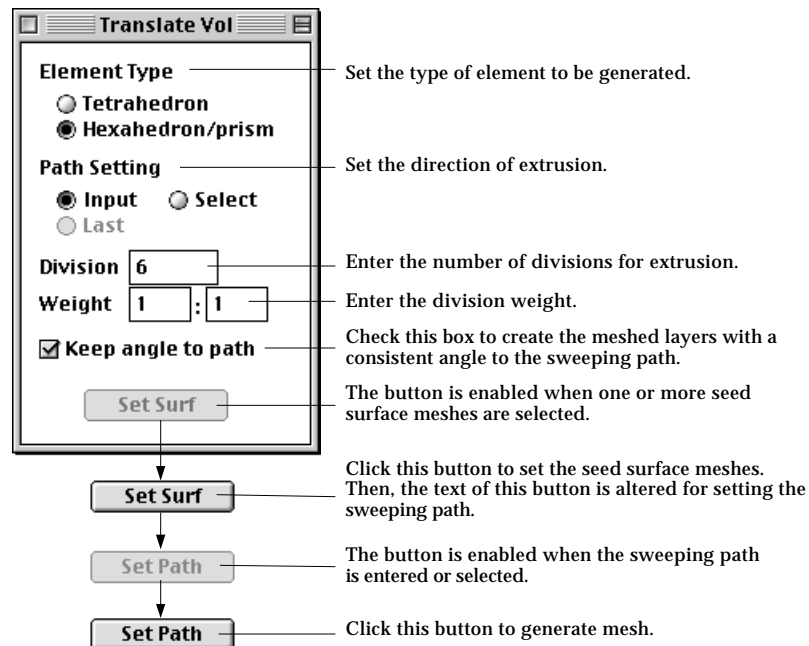


A volume mesh can be generated by translating the selected seed meshes along the specified sweeping path. Every node on the seed meshes makes a trace parallel to the sweeping path. New nodes are created along these traces, and all of them are contained in the newly generated mesh.

This method is useful in modeling curved structural members with uniform section. In applying this method, some care should be taken not to make a self-interfering mesh.

- 1) Choose “Translate (Volume)” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Translate Vol” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Select the option for path setting.

Choose one of the 3 options setting the sweeping path, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is enabled only when this method of mesh generation was applied at least once since the dialog box appeared.

- 4) Set the number of divisions for translation.

Specify how many rows of elements to be generated by translation. This

setting is applied only when the sweeping path is not divided. If divided curves are selected as the sweeping path, their divisions will be applied regardless of this setting.

- 5) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting and at the ending part of the curve.

- 6) Check or uncheck “Keep angle to path” check box.

If this check box is checked, the meshed layers are created so that they have a constant angle with the sweeping path. Otherwise, all the layers are made parallel.

- 7) Select surface meshes for seed meshes.

button is enabled when one or more surface meshes are selected.


- 8) Click  button.

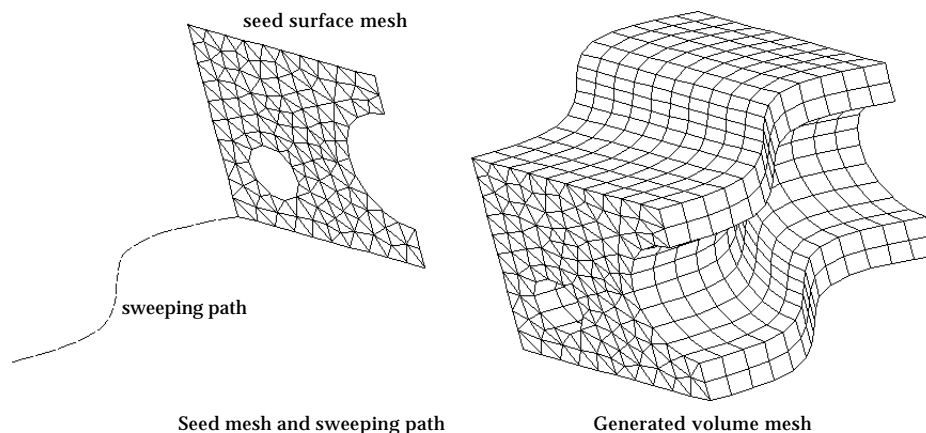
The selected surfaces are reserved as the seed meshes for mesh generation, and the button changes into  indicating that setting the sweeping path is expected in the next step.

- 9) Set the sweeping path.

Set the sweeping path by the method selected at step 3). Tools corresponding to the method are automatically activated, and the cursor changes accordingly. The sweeping path should be open, and its one end point should meet with a node on the seed curve. The sweeping path may be one curve or serially connected curves. If the sweeping path consists of more than one curve, they must be either all divided or all undivided. Mixed use of divided and undivided curves for sweeping path is not allowed.

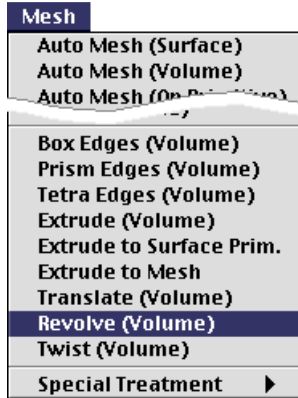
- 10) Click  button.

A mesh is generated by translating the seed meshes along the sweeping path.  button is reverted to  button. The program is now ready for generating another volume mesh. The surface selection tool  is automatically activated, if it is not in action.



< Example of volume mesh generation by translation >


### ■ Generating volume mesh by revolution

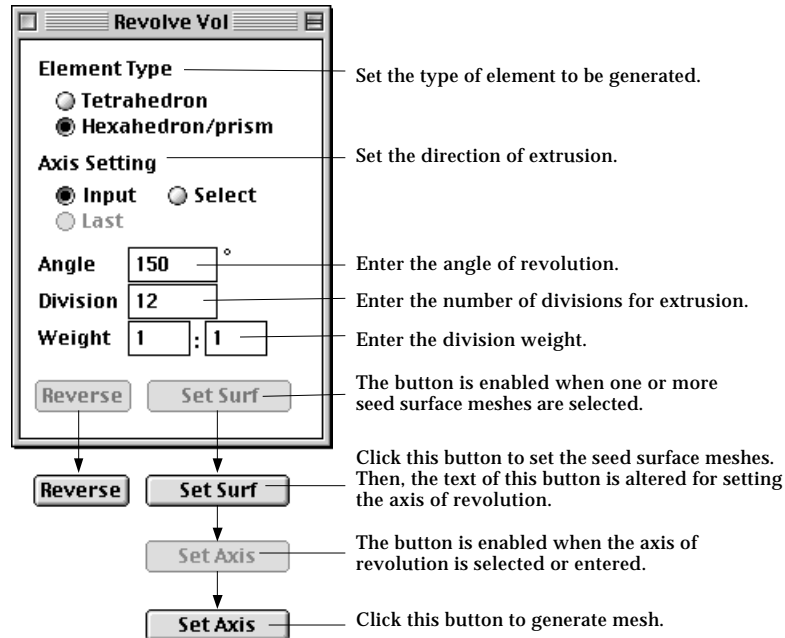


A solid of revolution is generated by revolving seed surfaces about the specified axis. Volume elements are created within a ring formed by revolving each of the surface element on the seed mesh.

The axis of revolution may be set interactively in any desired direction. The revolution may be full or partial depending on the specified angle. The direction of revolution can be reversed if necessary.

- 1) Choose “Revolve (Volume)” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Revolve Vol ” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Select the option for setting the axis of revolution.

Choose one of the 3 options setting the axis of revolution, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is enabled only when this method of mesh generation was applied at least once since the program started.

- 4) Set the angle of revolution.

Insert the angle of revolution in the dialog box. The angle should be greater than or equal to  $-360^\circ$  and less than equal to  $360^\circ$ . Both  $-360^\circ$  and  $360^\circ$  make a full surface of revolution. The negative sign reverses the direction of

revolution.

- 5) Set the number of divisions for revolution.

Specify how many rows of elements to be generated by revolution in circumferential direction.

- 6) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of revolution.

- 7) Select surface meshes forming seed meshes for revolution.

button is enabled when one or more surface meshes are selected.


- 8) Click  button.

The selected surfaces are reserved as the seed surfaces for mesh generation, and the button changes into  indicating that setting the axis of revolution is expected in the next step.

- 9) Set the axis of revolution.

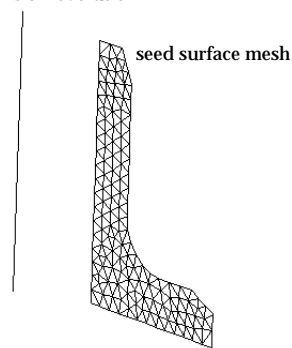
Set the axis of revolution by the method selected at step 3). Tools corresponding to the method is automatically activated, and the cursor changes accordingly.

- 10) Click  button.

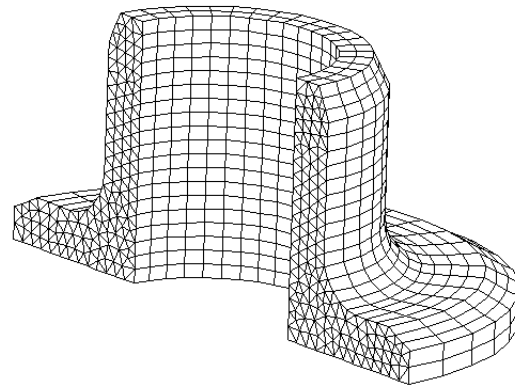
A mesh is generated by revolving the seed meshes about the axis of revolution.  button is reverted to  button. It is now ready for generating another volume mesh. The surface selection tool  is automatically activated, if it is not in action.

button is enabled, only when mesh generation is successful. Clicking  button reverses the sign of the revolution angle and regenerates the mesh.

axis of revolution



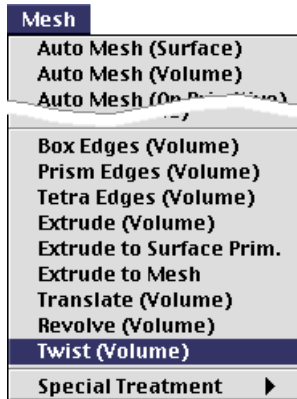
Seed mesh and axis of revolution



Generated volume mesh


< Example of volume mesh generation by revolution >

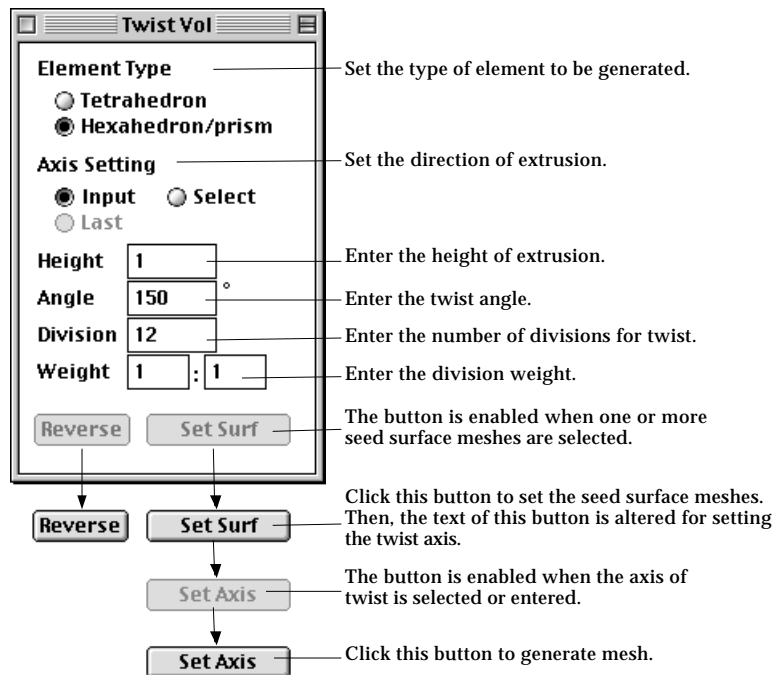
### ■ Generating volume mesh by twisting



Twisting is a method of mesh generation by extrusion combined with revolution. So, it may be termed as “twisted extrusion”. Elements are generated on the surface which is formed by extruding the selected seed curves along the specified axis while twisting the direction of extrusion about the axis by the specified angle. Along this twisted extrusion, the trace of a node on the seed curve makes a helix on which new nodes are created.

- 1) Choose “Twist (Volume)” from **Mesh** menu.

The surface selection tool  is automatically activated, and “Twist Vol” dialog box appears.



- 2) Set the element type.

If the element shape of the seed mesh is triangle, either tetrahedron or prism elements can be generated. Otherwise, only hexahedron elements can be generated, and thus this step is not necessary.

- 3) Select the option for setting the twist axis.

Choose one of the 3 options setting the twist axis, “Input”, “Select” and “Last” by clicking the corresponding radio button. The “Last” button is enabled only when this method of mesh generation was applied at least once since the program started.

- 4) Set the extrusion height.

Insert the extrusion height in the dialog box. The direction of extrusion is given by that of the twist axis which is directed from the first input point to



the second.

- 5) Set the twist angle.

Insert the twist angle in the dialog box. There is no limit in the acceptable range of twist angle. A negative sign reverses the direction of twist.

- 6) Set the number of divisions.

Specify how many rows of elements are to be generated in the direction of the twisted extrusion.

- 7) Set the weight of division density.

Enter the weight of division density in the form of  $w_1:w_n$ , which is the ratio between width of elements at the starting part and at the ending part of twisting.

- 8) Select surface meshes forming seed surfaces for twisting.

button is enabled when one or more surface meshes are selected.

- 9) Click  button.

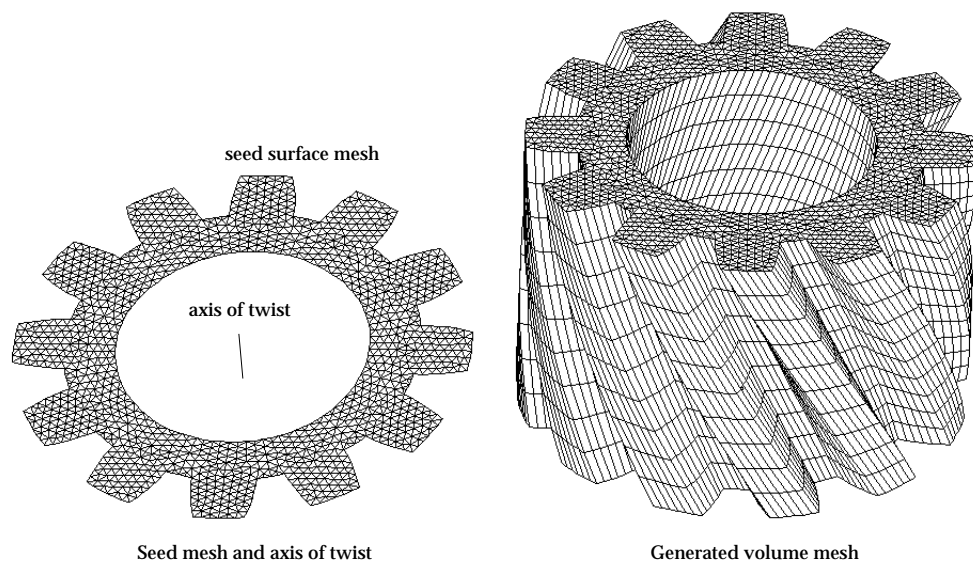
The selected surfaces are reserved as the seed surfaces for mesh generation, and the button changes into  indicating that setting the twist axis is expected in the next step.

- 10) Set the twist axis.

Set the twist axis by the method selected at step 3).

- 11) Click  button.


A mesh is generated by twisted extrusion of the seed surfaces.  button is reverted to  button. The program is now ready for generating another volume mesh.  button is enabled only when mesh generation is successful. Clicking  button reverses the sign of the twist angle and regenerates the mesh.



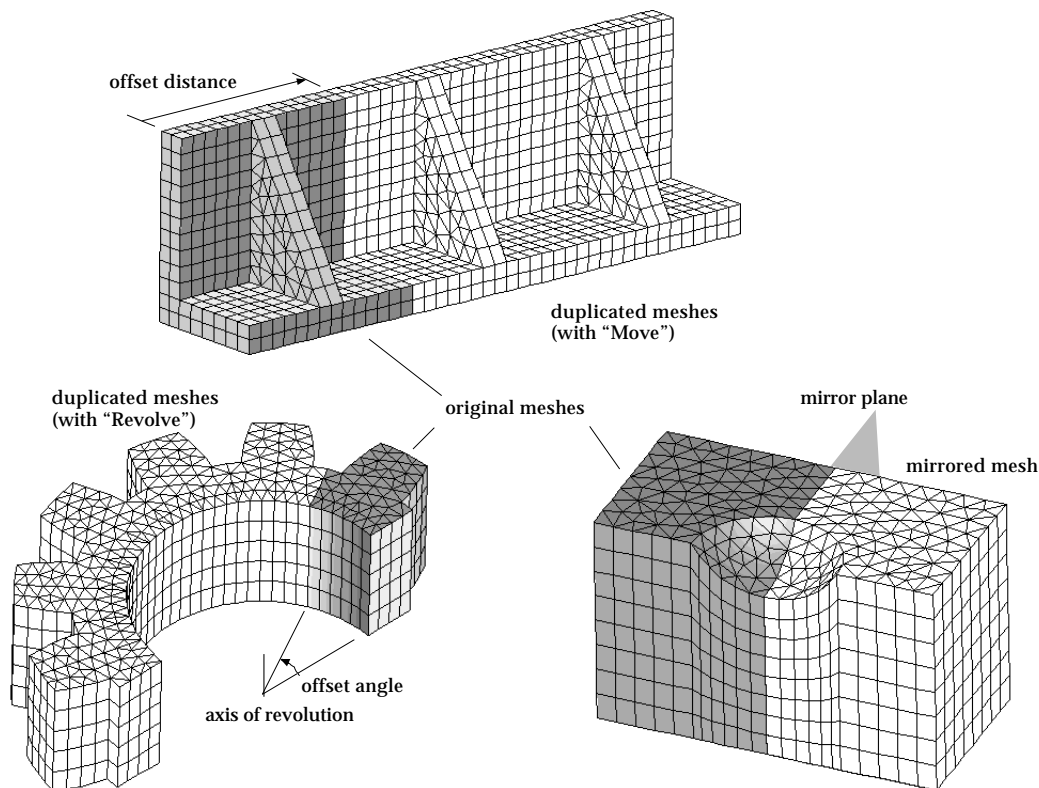
< Example of modeling 3-D solid by twisting >

## Duplicating volume meshes

Volume meshes may be proliferated by duplicating existing ones. The position and orientation of the duplicated meshes are determined by moving, revolving or mirroring coordinates of original objects with specified offsets. Surface meshes and curves are subsidiary objects belonging to a volume mesh. They are duplicated together with their host volume mesh.

The procedure of duplicating volume meshes is the same as that of duplicating curves. In order to start duplicating volume meshes, click volume selection tool  if it is not in action. Then, select one of the submenu items for duplications: “Move”, “Revolve” or “Mirror”. Further details of duplication procedure are explained in Chapter 3, and are not repeated here.

There is no limit in the number of duplicates which can be made at once. You can specify the number of duplication using the dialog box for the corresponding duplication method. VisualFEA does not check the interference between meshes. So, you must be cautious about interference or compatibility of meshes when duplicating volume meshes. The following figure shows examples of multiplying volume meshes by duplicating existing ones with moving, revolving or mirroring.



< Examples of duplicated volume meshes >

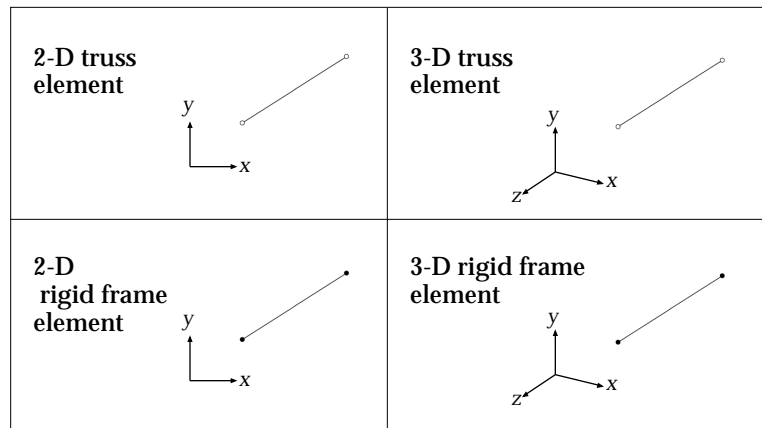
## Frame Element Generation

The only shape of frame elements available in VisualFEA is straight line. All the elements are assumed to be linear, i.e. with a node at both ends. Frame elements may be created by one of the following methods:

- making straight lines : Frame elements are automatically generated on a newly created straight lines.
- dividing curves : Dividing curves assign frame elements on curves by the number of divisions.
- mesh generation : Frame elements may be generated using some of the surface generation techniques.
- duplication : Frame elements may be multiplied by duplication.

There are 4 types of frame elements as shown below, namely 2-D truss, 3-D truss, 2-D frame and 3-D frame. The element type is determined by the problem setup. Identical procedures are applied for frame element generation regardless of the element type.

< Types of frame element >

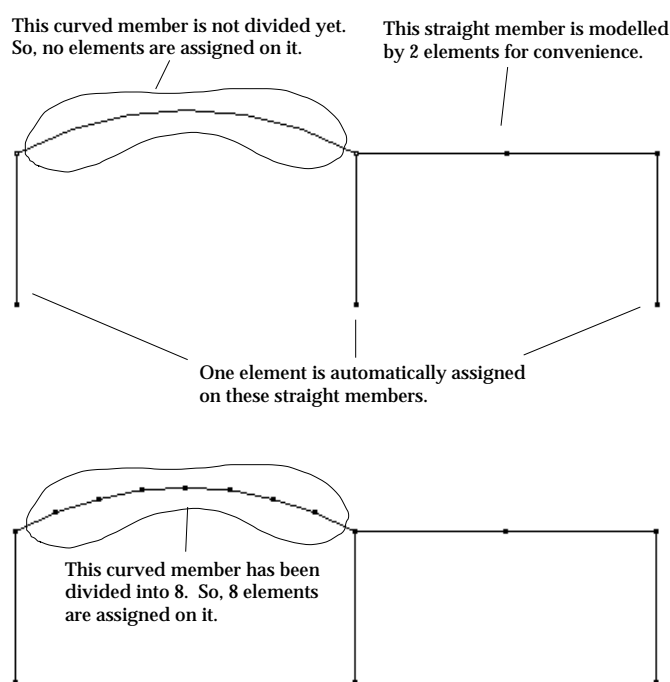


Frame elements are represented by a line segment with small circles at both ends. The representation of rigid frame and truss elements are distinguished by the shape of these circles. The circles are filled with black for rigid frame elements and are hollow for truss elements.

Frame elements may be included in 2-D or 3-D solid analysis. In this case, frame elements are created by assigning element properties. The method is detailed in “Element Properties” section of Chapter 5, and will not be explained here.

### Creating frame elements using straight lines or curves

A frame members which may be modeled by a straight line or by a curve. One frame member may consist of one element or multiple elements. A frame element is automatically generated on a straight line, but elements are not assigned on a curved members, until it is divided. A curved member may be approximated by multiple line elements. Even a straight member sometimes needs to be modeled by more than one element in connection with load assignment, force diagrams and so on as exemplified in the following figure.



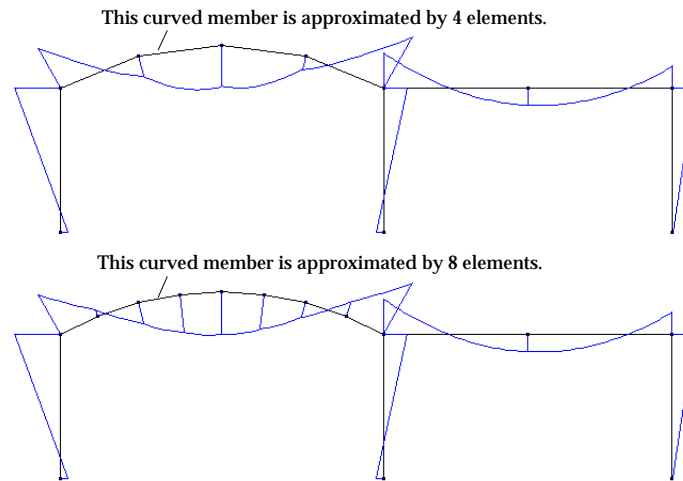
< Example of element generation on frame members >

#### ■ Creating a frame element by inputting a straight line

If the analysis subject is defined as a frame, i.e, 2-D truss, 3-D truss, 2-D frame or 3-D frame, in the problem setup, a frame element is automatically assigned on a newly created straight line. Line elements are usually assigned on a line or a curve by dividing it. So, assigning an element on a line is equivalent to dividing it into one. Dividing a curve into one implies creating a two node on both ends of the selected curves and can be achieved by the menu command “Divide into 1.” But, you don’t have to use this command, because it is automatically done when a straight line is newly created. Automatic element assignment facilitates modeling frame, because most frame members are straight line. This automatic assignment is applied only for a frame.

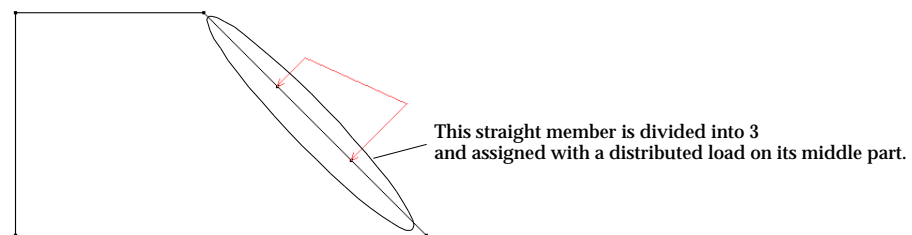
### ■ Generating frame elements by dividing curves

As mentioned previously, a straight line is the only frame element shape available in VisualFEA. Therefore, a curved member should be modeled by multiple pieces of line segments which can be obtained by dividing curves. When a curve is created, initially no elements are assigned on it. Elements are generated on the curves when they are divided. The curves may be divided into as many elements as necessary. More elements will produce better approximation of the curved members but will increase the complexity of the model as exemplified below.



< Bending moment diagrams with different approximation of curved member >

Straight lines are initially assigned with an element, but more than one elements may be generated on it if necessary. Division of a straight line does not affect the analysis results, because straight members can be modeled exactly by one element. However, there are situations under which straight members need be modeled by multiple elements. In such cases, you may create either multiple line segments with one element or a straight line with multiple elements. The following figure shows an example of such a case in which distributed load is applied on some part of a straight member. The inclined straight line member is divided to three parts, because distributed load cannot be applied partially on an element in VisualFEA.



< Example of a straight member partially loaded with trapezoidal force >

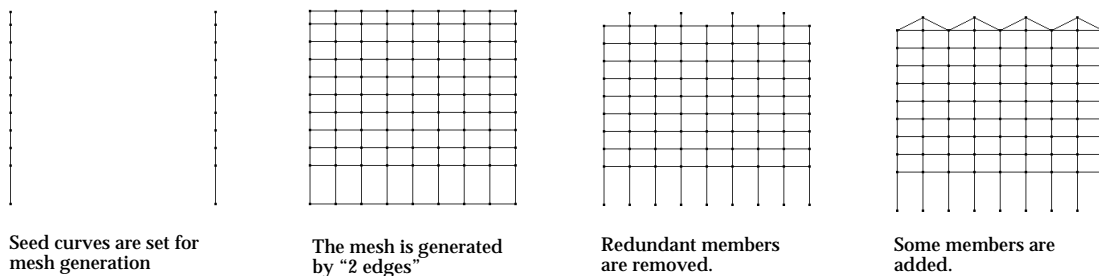
### Creating frame elements using mesh generation functions

Large and complex skeletal structures like buildings consist of large number of frame members. In modeling such a frame structure, it is difficult to create individual members one by one. Such a model can be constructed simply by generating frame elements as a mesh and then editing part of the mesh if necessary. Having identified the structure as a frame, mesh lines are recognized as frame elements. Frame elements can be generated using the following mesh generation functions:

- “2 Edges”
- “3 Edges”
- “4 Edges”
- “Extrude (Surface)”
- “Extrude to Curve”
- “Sweep (Surface)”
- “Revolve (Surface)”
- “Twist (Surface)”

In fact, the above list includes all techniques of surface mesh generation except “Auto Mesh” and “Auto Mesh On Primitive.” These two functions are excluded, because they cannot control the shape of the generated mesh as normally required in modeling frames. Meshes with either triangular or quadrilateral cells (basic formation of frame members corresponding to an element in a surface mesh) may be generated. The arrangement of triangular cells can be controlled by the options “Left”, “Right” and “Union Jack” provided as radio buttons in the dialog boxes for mesh generation. The usage of the setting is the same as is in surface mesh generation. The “Optimal” button is disabled, because regular shape of the mesh is not guaranteed by this option.


The following example shows how efficiently a complex frame model can be constructed by using mesh generation functions. First, a frame mesh is generated using “2 Edges”, and then redundant members are deleted. Lastly, some members are added.



< Example of constructing frame model using mesh generation >

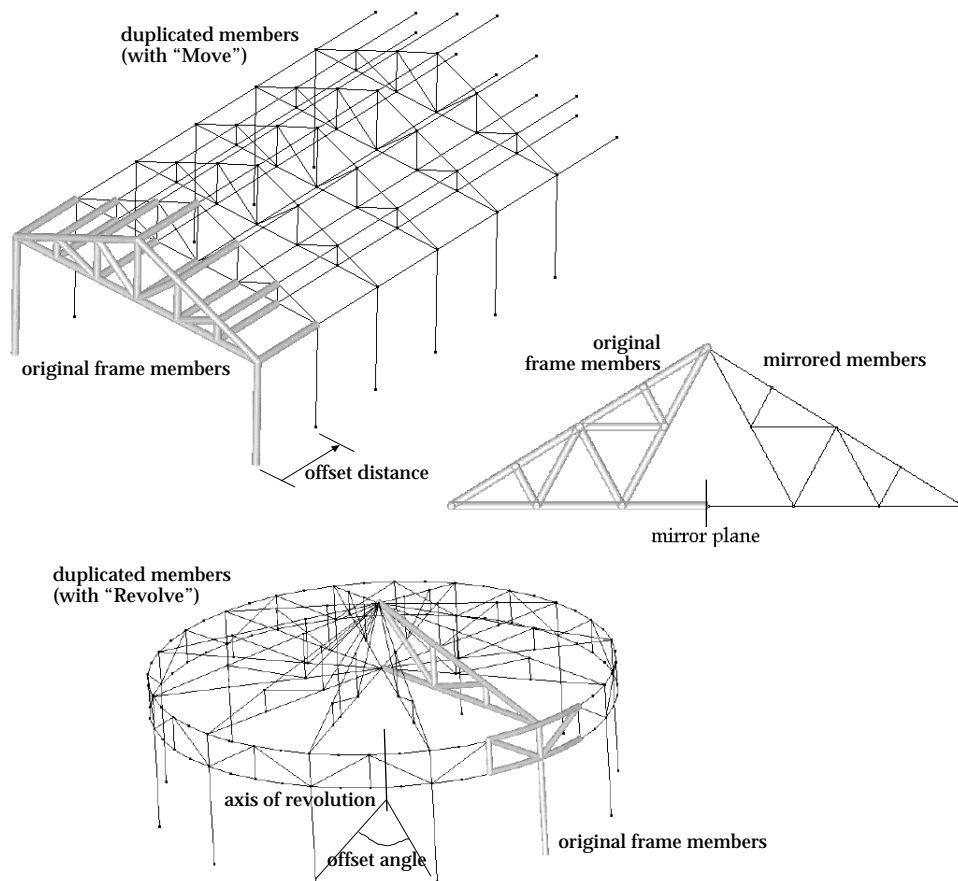
## Duplicating frame elements

New frame elements may be created by duplicating existing ones. The position and orientation of the duplicated frame elements are determined by moving, revolving or mirroring coordinates of original meshes with specified offsets.

Duplicating frame elements is identical to duplicating curves except that not only the curves but also the elements are duplicated. However, individual elements cannot be duplicated independently. In order to start duplicating frame elements, click the curve selection tool  if it is not in action. Then, select one of the submenu items for duplications: “Move”, “Revolve” or “Mirror”. Further details of duplication procedure are explained in Chapter 3, and are not repeated here.

The following figure shows examples of duplication of frame elements with moving, revolving or mirroring.

*You should be careful not to create overlapping members by duplication. It is advisable to avoid duplicating members on the axis of revolution or mirror plane. That is the reason why the members on the axis of revolution or mirror plane are absent in the following examples. Those members can be added manually before or after duplication.*



< Examples of duplicated frame members >

## Mesh Editing

You may edit or modify the meshes. The current version of VisualFEA has somewhat limited but essential functions of mesh editing. Part of the editing commands for curves and surface primitives are also effective for mesh editing. Their usage and context are almost identical except that they are applied for different types of objects.

### General mesh editing commands

Selected meshes can be deleted, copied, pasted, or merged simply by issuing the corresponding **Edit** menu item.

#### ■ Undoing mesh generation

In order to cancel the last mesh generation, select "Undo" item of **Edit** menu immediately after the mesh generation. Undoing will remove all meshes generated by the last processing, regardless of the method of mesh generation. For example, if meshes are generated by multiple sets of duplication, all the duplicated sets will be removed by undoing.

After undoing, the menu item changes into "Redo". Selecting the item again will recover the meshes and revert the menu item into "Undo". Thus, "Undo", and "Redo" alternates as the menu item.

#### ■ Deleting meshes

In order to delete meshes, first activate the corresponding object selection tool. In other words, click the curve selection tool to delete frame members if it is not currently in action. Click the surface or the volume mesh selection tool to delete surface or volume meshes respectively. In the next step, select the objects to delete, and select "Clear" item from **Edit** menu, or press **Delete** key.

When a mesh of any type is deleted, its subordinate objects are also deleted. For example, if a volume mesh is deleted, all surface meshes and curves belonging to the mesh are deleted. However, the seed meshes or seed curves used for generation of the mesh are not deleted. And, surface meshes and curves shared with other volume meshes will not be deleted. Likewise, if a surface mesh is deleted, all the curves, except seed curves, belonging to the mesh are deleted. Curves shared with other meshes will not be deleted. Accordingly, a volume mesh should first be deleted in order to delete surface meshes belonging to the volume mesh. Nodes and elements cannot be deleted individually. In order to delete them, all the meshes possessing them should be deleted.



### ■ Copying meshes

In order to copy meshes of any object type, select the objects to copy, and select "Copy" item from **Edit** menu. The copied meshes are stored in the computer memory and can be pasted later.

### ■ Cutting meshes



Cutting deletes the selected meshes, and at the same time, stores the deleted meshes in the computer memory. This is done by "Cut" command in **Edit** menu, and has the same effect as deleting after copying the selected meshes.

### ■ Pasting meshes

"Paste" command in **Edit** menu is enabled only when there exist objects of any type stored in the memory by "Cut" or "Copy" command. Otherwise, the item is dimmed and cannot be selected. When meshes have been stored in the computer memory, "Paste" command recovers and pastes them on the screen

### ■ Merging meshes

Two or more meshes can be merged into one. It is sometimes more convenient to handle a large lump than a number of small pieces. Or, it may be better to treat a few adjacent meshes with homogeneous properties as one body. In such cases, merging meshes is very useful. Merged meshes form a single object.

Only meshes of the same object type can be merged. In order to merge surface meshes, first click surface selection tool , if it is not in action. Select two or more surface meshes. Then, "Link" item in **Edit** menu is enabled. Selecting the item will merge the meshes into one. Likewise, volume meshes can be merged by issuing "Link" command after selecting two or more volume meshes using volume selection tool .


"Link" command for meshes can be canceled by "Undo" command in the same menu immediately after the action. However, once merging is completed, the merged meshes cannot be separated later.

## Modifying nodal coordinates

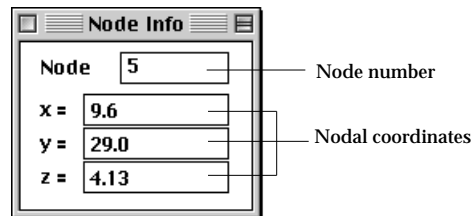
Meshes consist of a number of nodes and elements. You may modify the coordinates of nodes, one by one, and accordingly the shape of the elements connected to the nodes. This can be done either by using mouse or by using keyboard. But, nodes on the mesh boundaries cannot be modified individually.

### ■ Dragging nodes using mouse

You may drag individual nodal points interactively by using the mouse as explained in the following:

- 1) Click node selection tool , if it is not in action.
- 2) Double click the node to drag.

The node is marked with a small blue rectangle, and “Node Info” dialog box appears. The dialog has the basic information on the marked node. If “Node Info” dialog is already on the screen, you don’t have to double click to invoke the dialog. Instead, click the node only once. Then, the contents of the dialog are renewed.



- 3) Position the screen cursor over the node and press the mouse button.
- 4) Move the mouse with the button pressed.

The screen cursor moves, and the nodal point moves along with the cursor movement. While the node is moving, the updated nodal coordinates are continuously echoed as texts in the “Node Info” dialog. At the same time, the boundary lines of the elements connected to the node are also modified. Accordingly the modified shape of the elements are drawn on the screen.

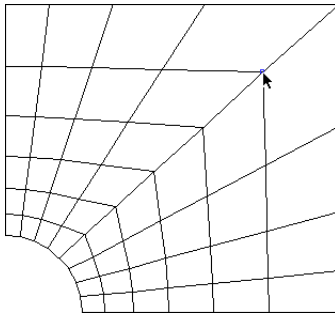
- 5) Release the mouse button.

The node is settled the the point where the mouse button is released.

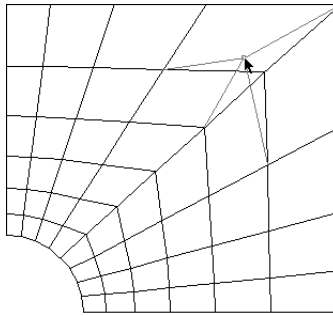
- 6) Repeat step 3), 4), 5) until the node is placed on the desired position.

While “Node Info” dialog box is on, clicking another node will switch the node to drag. The contents of the dialog box are also renewed. So, you may drag nodes one after another by repeating step 3) through 6). In order to quit dragging nodes, close “Node Info” dialog by clicking the close box of the dialog.

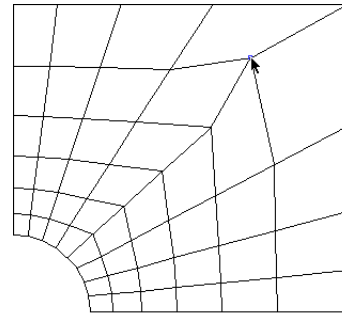
*You must make sure that the corresponding grid plane is on and contains the point to which the node is to be dragged. The reason is that VisualFEA constrains the movement of cursor by the grid plane.*



Place the cursor over the node to drag, and press mouse button.



Move the mouse with mouse button pressed.






Release the mouse button.

< Process of dragging a node >



### ■ Modifying nodal coordinates by keyboard input

Nodes can be moved to the desired position by directly entering the coordinates of the nodes using keyboard.

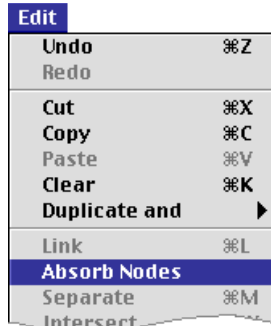
- 1) Click node selection tool , if it is not in action.
- 2) Double click the node to drag.  
The node is marked with a small blue rectangle, and “Node Info” dialog box appears.
- 3) Edit the coordinates of the node to modify using the dialog.  
The coordinates of the node are displayed as editable texts on the dialog, and can be edited one by one.
- 4) Press  key (Windows :  key).  
The edited texts are entered as the new coordinates of the node, and the nodal point moves to the new position. The shape of the elements connected to the node is accordingly modified.

While “Node Info” dialog is on the screen, the contents of the dialog are renewed with the information of another node by simply clicking the node. You can modify the coordinates of the node, the information of which is currently displayed on the dialog.


### ■ Changing node number

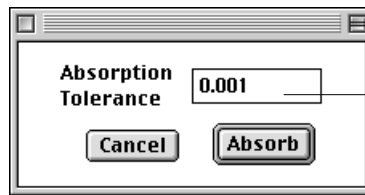
The node numbers can also be modified using “Node Info” dialog described above. Edit the text representing the node number and press  key (Windows :  key). Then, the node number is changed into the newly entered number. If the modified number is less than original one, all the node numbers between them will be increased by one. If the modified number is greater than original one, all the node numbers between the original and the modified nodes will be decreased by one.

### ■ Absorbing nodes



A node shared by 2 or more adjacent mesh regions may be unintentionally separated into more than one nodes, owing to limited precision of computation. This happens rarely, but its consequence may be critical. Such unintentionally separated nodes may be merged into one by the following steps.

- 1) Click node selection tool , if it is not in action.
- 2) Select nodes in the range of suspected nodal separation.  
Assume roughly the range which may contain separated nodes, and select the nodes within the range. Or select all the nodes in the model, if it is difficult to assume the region.
- 3) Choose "Absorb Node" item from **Edit** menu.  
A dialog for setting absorption tolerance pops up on the screen. Edit the editable text of absorption tolerance, which is the limiting distance for node absorption.



Set the tolerance of absorption by editing this text.

- 4) Click **Absorb** button in the dialog.  
The nodes within tolerance distance from each other are merged into one. New nodal coordinates are determined by averaging the coordinates of the merged nodes.




## Transforming meshes

Meshes can be moved, resized or rotated as a whole. In other words, the geometry of selected meshes and their subordinate objects can be transformed. This capability of VisualFEA is useful more frequently in displaying analysis results which is described in the “Visualizing Scalar Data by Contours” section of Chapter 7. Mesh transformation applies only to surface and volume meshes, but not to frame meshes.

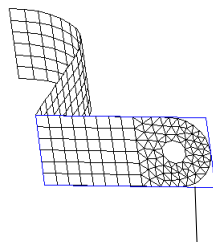
### ■ Moving meshes

You may move the selected meshes either by dragging the bounding box or by entering distances of movement in X, Y and Z directions as explained in the following:

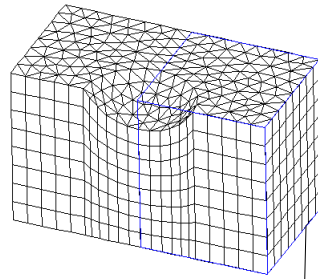


- 1) Click surface selection tool  or volume selection tool  depending on the type of mesh to move.
- 2) Select the meshes to move.  
Selected meshes are included in the list of the meshes to move.
- 3) Click object movement tool .

A bounding box surrounding the selected meshes is drawn on the screen. The bounding box may be a rectangle or a right hexahedron, as shown in the following example, depending on the spatial dimension formed by the meshes to move. If all the nodes within the meshes have an identical coordinate in any one of X, Y or Z, the bounding box is reduced to a rectangle.



The bounding box is a rectangle.



The bounding box is a right hexahedron

#### < Bounding boxes surrounding planes and 3-D volumes >

Move Distance

X	4.01
Y	0
Z	0

At this stage, editable text boxes displaying the distances of movement in X, Y and Z directions appear at the bottom of the tool palette as shown at left.

- 4) Place the screen cursor on one edge of the bounding box, and press the mouse button.

The direction of movement is set as parallel to the selected edge.

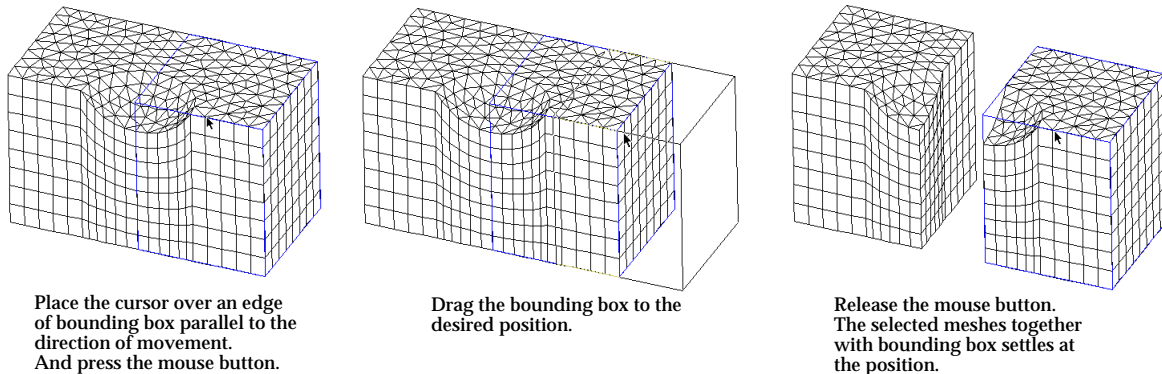
- 5) Move the mouse with the button pressed.

The bounding box moves by sliding on the selected edge. The distance of movement is determined by the cursor movement, and is displayed on the text boxes at the bottom of the tool palette.

- 6) Release the mouse button.

The bounding box settles at the final position where the mouse button is released. The selected meshes also are moved to the new position.

The meshes can be moved to the desired position by repeating step 4), 5) and 6) with different edges. Or, the desired movement may be achieved at once by directly entering the offset distances in the editable text boxes of the tool palette.






< Moving meshes by dragging the bounding box >

## ■ Rotating meshes

You may rotate selected meshes either by turning the bounding box using the mouse or by entering the angles of rotation about X, Y and Z axis as explained in the following:



- 1) Click surface selection tool  or volume selection tool  depending on the type of mesh to rotate.
- 2) Select the meshes to rotate.  
Selected meshes are included in the list of the meshes to rotate.
- 3) Click object rotation tool .

A bounding box surrounding the selected meshes is drawn on the screen. Editable text boxes displaying the angles of rotation about X, Y and Z axes appear at the bottom of the tool palette as shown left.

Object Angle	
$\theta_x$	0
$\theta_y$	51.9
$\theta_z$	0

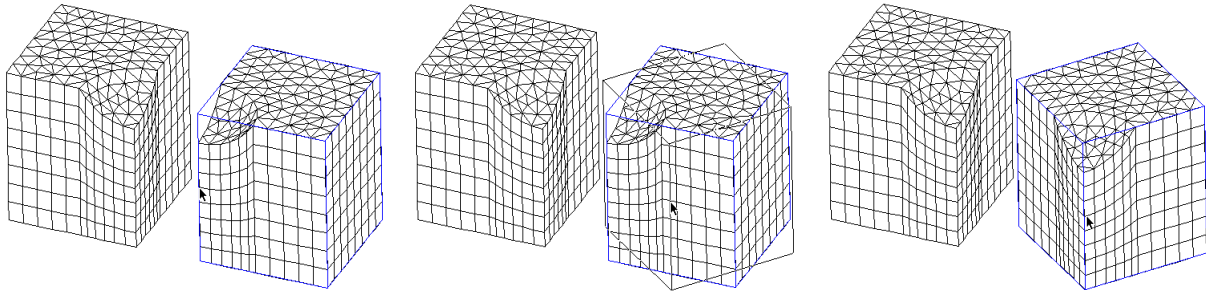
- 4) Place the screen cursor on one edge of the bounding box, and press the mouse button.  
The axis of rotation is set to be parallel to the selected edge, and to pass through the centroid of the bounding box.
- 5) Move the mouse with the button pressed.

The bounding box rotates about the coordinate axis parallel to the selected edge. The angle of the rotation is determined by the edge moving along the cursor movement. The angle of rotation is continuously updated on the text boxes at the bottom of the tool palette, while the bounding box is being rotated.

- 6) Release the mouse button.

The bounding box settles with the final orientation where the mouse button is released. The selected meshes are also rotated to the new orientation in accordance with the bounding box.

The meshes can be rotated to the desired orientation by repeating step 4), 5) and 6) with different edges. Or, the desired rotation may be achieved at once by directly entering the angles of rotation about each coordinate axis in the editable text boxes of the tool palette.



Place the cursor over an edge of bounding box parallel to the axis of rotation. And press the mouse button.

Drag the edge. Then, the bounding box rotates along with the edge. Rotate the box to the desired orientation.




Release the mouse button. The selected meshes are rotated to fit within the rotated bounding box.

< Rotating meshes by dragging an edge of the bounding box >

## ■ Resizing meshes



You may resize the selected meshes either by rubber-banding the bounding box or by entering the relative scale in X, Y and Z axis as explained in the following:

- 1) Click surface selection tool  or volume selection tool  depending on the type of mesh to resize.
- 2) Select the meshes to resize  
Selected meshes are included in the list of the meshes to resize.
- 3) Click object resizing tool .

A bounding box surrounding the selected meshes is drawn on the screen. Editable text boxes displaying the relative scales of the meshes in X, Y and Z axes appear at the bottom of the tool palette as shown left. The relative scales are set to 1 at the beginning.

Object Scale	
$r_x$	1
$r_y$	1
$r_z$	1.7

- 4) Place the screen cursor on one edge of the bounding box, and press the mouse button.  
The direction of the resizing is set as parallel to the selected edge.
- 5) Move the mouse with the button pressed.

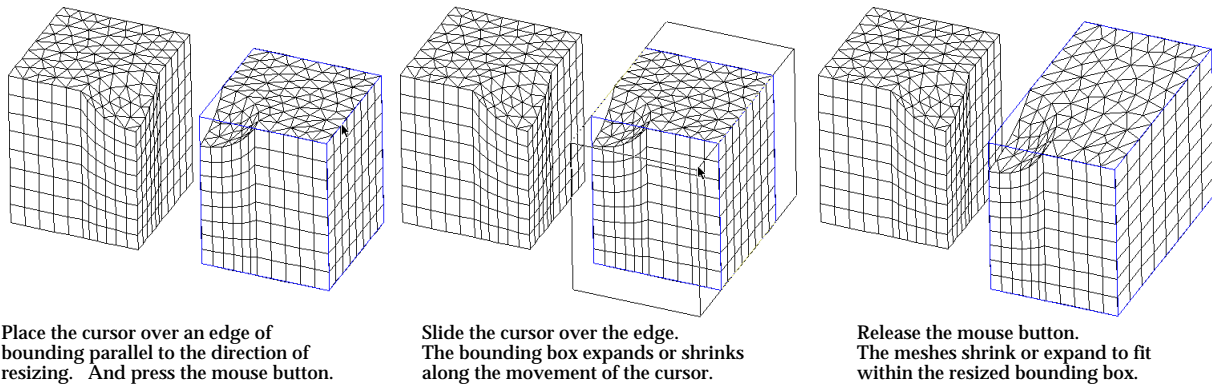
The bounding box expands or shrinks in the direction parallel to the selected edge. The relative scale is determined by the movement of the cursor on the edge. If the cursor moves toward the center of the edge, the relative scale is reduced and the selected meshes shrink. If the cursor moves toward the ends

of the edge, the meshes expand. The relative scale is continuously updated on the text boxes at the bottom of the tool palette, while the bounding box is being resized.

6) Release the mouse button.

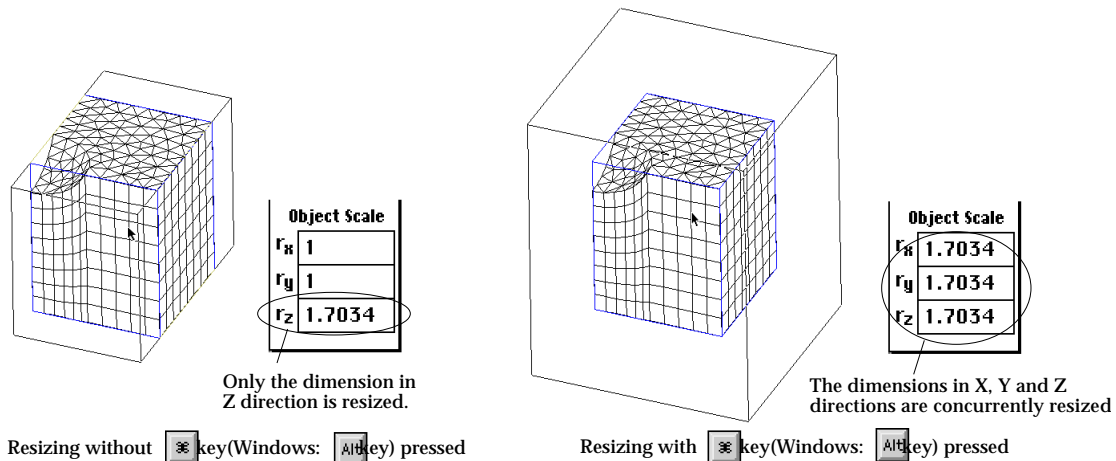
The bounding box settles with the final scale designated by the cursor position where the mouse button is released. The selected meshes are also resized to the new scale in accordance with the bounding box.

The meshes can be resized to the desired scale by repeating step 4), 5) and 6) with different edges. Or, the desired resizing may be achieved at once by directly entering the relative scale in each coordinate axis in the editable text boxes of the tool palette.



#### < Resizing meshes by rubber-banding an edge of the bounding box >

Instead of resizing only in one edge direction, you may resize the meshes with equal relative scale in X, Y Z direction. This can be done by pressing key (Windows : key) while resizing the meshes following the above procedures.

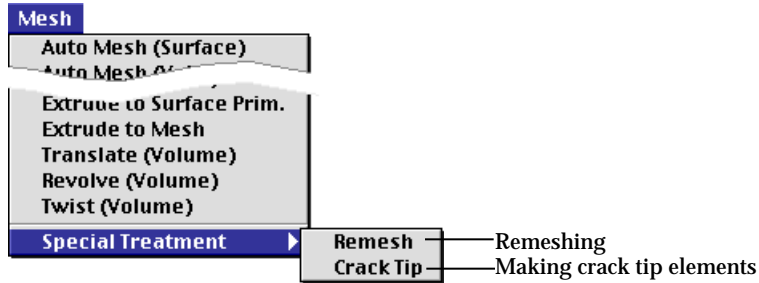


#### < Resizing meshes with and without key (Windows : key) pressed >




## Other mesh treatments

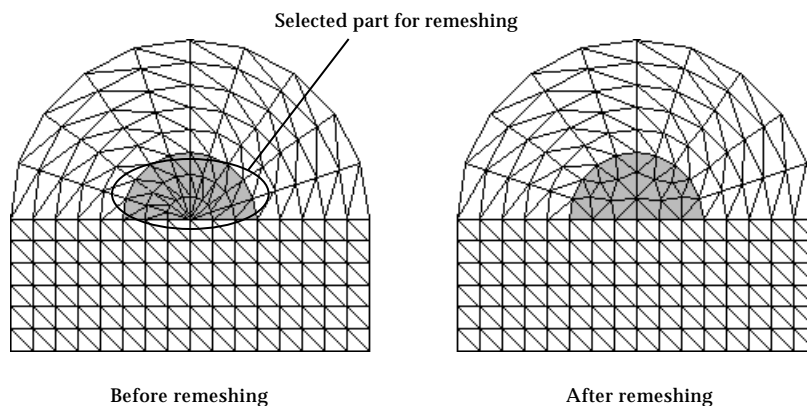
There are functions for special treatments of meshes. The current version of VisualFEA supports 2 functions: creating a crack tip and remeshing.



### ■ Remeshing

A selected part of a surface mesh can be remeshed. Automatic mesh generation algorithms are used in remeshing operation.

- 1) Click the element selection tool , if it is not in pressed state.  
There appears **Select** menu at the right end of the menu bar. "Select Surface Elem." item of the menu should be checked.
- 2) Select the region of elements to remesh.  
The selected elements should be within a surface mesh and form a unified region. Otherwise, remeshing will not work.
- 3) Choose "Remesh" item from **Special Treatment** submenu.  
If all the selected elements have quadrilateral shape, quadrangle elements are generated by remeshing. Otherwise, triangular elements will be generated.




< Example of remeshing >

*Remeshing cannot be undone, and therefore the original mesh cannot be recovered. It is a good strategy to save the model before remeshing, so that the original mesh can be recovered in case the remeshing produces unsatisfactory result.*

### ■ Making crack tip elements

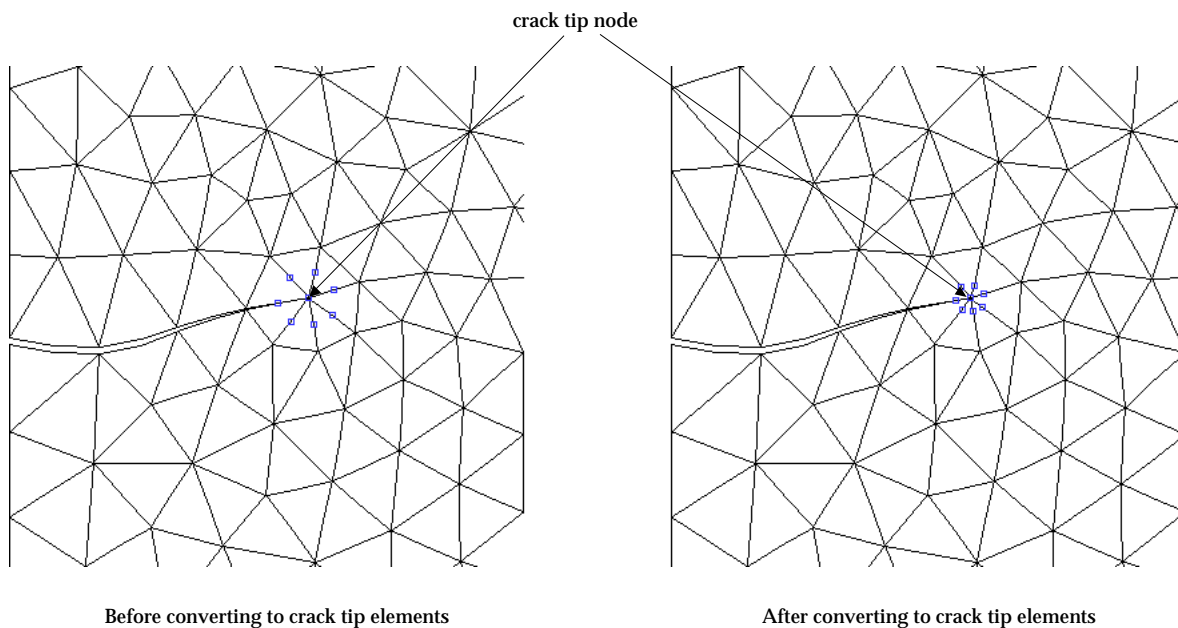
The  $1/\sqrt{r}$  stress singularity at a crack tip is usually modeled by using quarter-point elements in which the mid-side nodes are positioned at one quarter of the edge length from the crack tip node. The elements surrounding a crack tip can be converted into quarter-point crack tip elements by the following steps:

- 1) Click the node selection tool , if it is not in pressed state.
- 2) Select the crack tip node.

The crack tip node should be at the tip of the crack, and the surrounding elements should be of quadratic order, i.e., 6 node triangular or 8 node quadrilateral element.

- 3) Choose "Remesh" item from **Special Treatment** submenu.

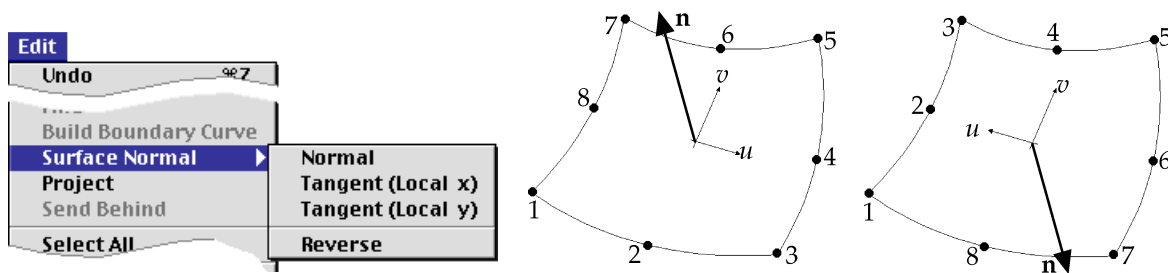
The surrounding elements are converted into quarter point crack tip elements. The mid-side nodes of the elements move to the quarter points along the edges toward the crack tip.



<Example of creating crack tip elements >

## Surface normal and tangent directions

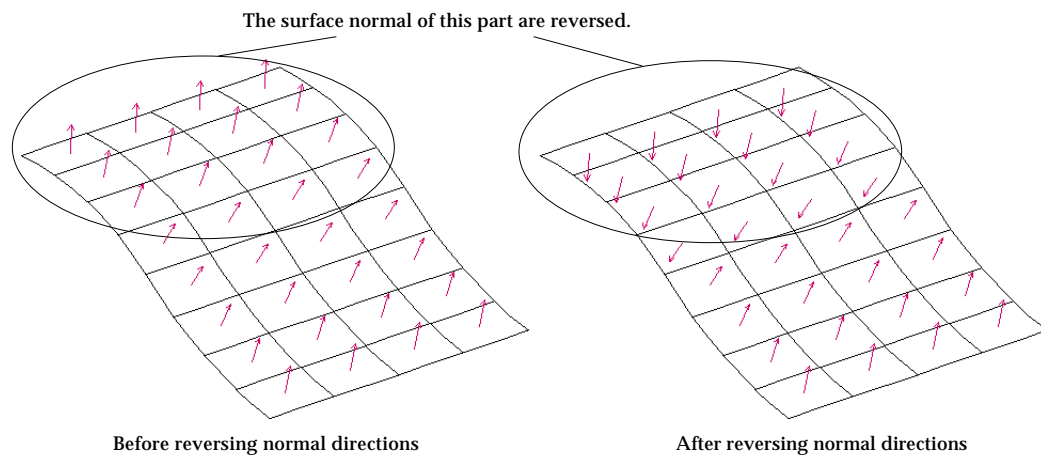
VisualFEA uses surface normal directions for various purposes including model rendering, force assignment, local coordinates of a shell element and so on. The normal direction at a specific point on a surface mesh is determined by the element containing the point. It is related to the node numbering within the element. The tangent directions are especially important for a shell element with its local coordinates based on the surface normal and tangent directions. They are defined by the surface normal and the global coordinate axes. You can identify the surface normal and tangent directions, and reverse the directions if necessary.



<Node numbering and normal direction on an element>

### ■ Displaying the surface normal direction

To display the surface normal, select surface meshes whose normal directions are to be displayed. And choose the "Normal" item of **Surface Normal** submenu. Then, a surface normal vector is drawn at the center of each element constituting the surface meshes.



<Example of reversing surface normal direction>

### ■ Displaying surface tangent directions

There are two tangent directions which makes 3 orthogonal axes along with the surface normal. The tangent directions are also the local coordinates axes used in shell elements. To display the directions tangent to the surface, select surface meshes whose tangent directions are to be displayed. And choose the "Tangent (Local x)" or "Tangent (Local y)" item of **Surface Normal** submenu. The local x direction of a shell element is displayed by selecting "Tangent (Local x)" item, and the local y direction by "Tangent (Local y)".

### ■ Reversing surface normal directions

The surface normal directions can be reversed by selecting surface meshes and choosing "Reverse" item of **Surface Normal** submenu. Reversion of a normal vector on an element results in reversing the node numbering within the element.

## **Chapter 5**

### **Data Assignment**

## *Chapter 5   Data Assignment*

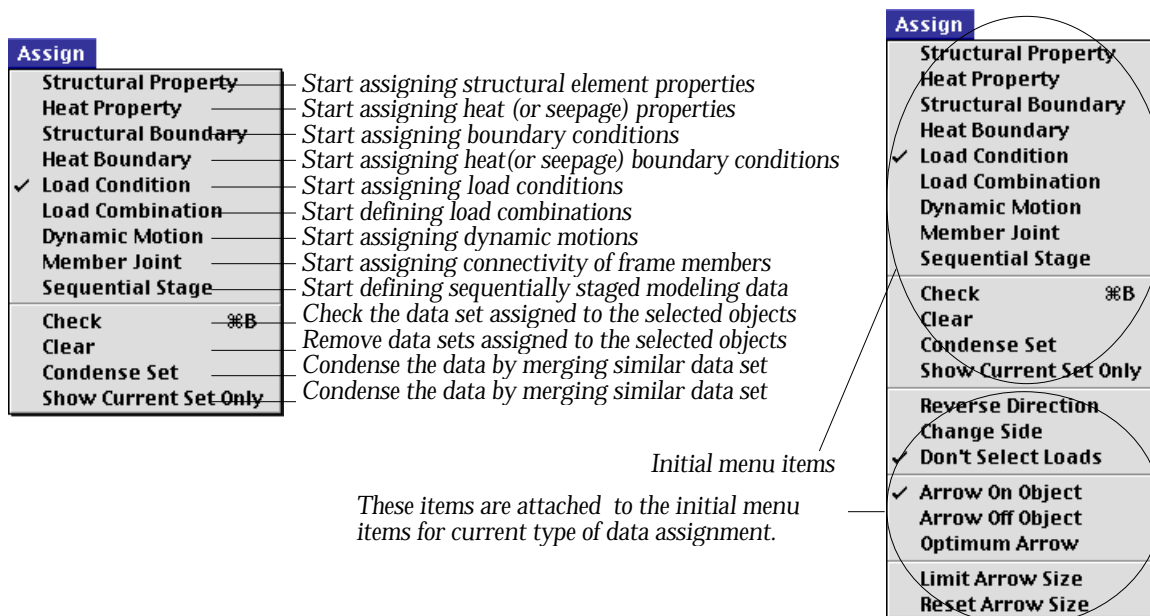
## Chapter 5 Data Assignment

A problem to be solved by the finite element method is defined by the geometry and the attributes of the solution domain. The geometry is modeled by nodes and elements created by mesh generation as described in Chapter 4. The attributes consist of various data such as material properties, boundary conditions, load conditions and so on. VisualFEA provides functions for creating these data and assigning them on the nodes and elements. The attribute data handled by VisualFEA can be classified basically into 3 categories: properties of the solution domain, conditions at the boundaries, and data related to external effects such as load and temperature.

The data in each category have many items varying with the type of analysis and the geometry of the domain. Data sets are first defined with specific values of all relevant items, and then assigned to the selected nodes or elements. These actions are carried out interactively using various functions provided in the tool palette, menu or dialog box. The states of the data assignment can also be visualized and checked for their correctness. The major functions related with data assignment can be summarized as follows:

- creating and deleting data sets : define a new data set and enter values of the individual items of the data set using a dialog box, and delete a data set.
- assigning and unassigning data : assign a data set to the selected objects, and unassign a data set from the selected objects.
- checking and visualizing data assignment : visualize the assignment of a specific data set or identify the data set assigned to the selected objects.

There are also functions specific to certain data types, as shown in the following **Assign** menu.



## Overview of Data Assignment

There are a few different types of data to be assigned to the finite element model. But the procedures of assigning data are similar and consistent regardless of the data type as described below.

### Basic composition of data

The minimum amount of data necessary for finite element analysis is varied with the type of problem to be solved. VisualFEA can do 3 types of analysis : structural, heat conduction and seepage. And therefore, data required for these 3 types of problems are described below.

#### ■ Data for structural analysis

The attribute data necessary for structural analysis are basically composed of the following components:

- material properties: elastic constants, unit weight, thermal expansion coefficients, etc.
- structural boundary conditions: structural constraints or supporting states such as spring.
- load conditions: various types of applied forces including uniform force, moment, body force, etc.

The above components are essential for structural analysis. Multiple load conditions can also be defined by linear combination of the load condition sets.

- load combinations: combinations of load condition sets defining multiple load condition. This item is available only for linear static structural model with more than 2 load condition sets.

The following data items are additionally required to define other characteristics of the problem.

- local member connectivity: pin ended, rigid ended.
- dynamic motion: dynamic displacement, velocity and acceleration.

When the necessary data are completely assigned, the problem are are ready for finite element solution.

#### ■ Data for analysis of heat conduction

The data necessary for heat conduction analysis are basically composed of the following 2 components:

- material properties: thermal conductivities in X, Y and Z directions.



- heat and temperature boundary conditions: specified temperatures, specified heat flux, convection boundary conditions, heat source, etc.

In order to make the problem solvable by VisualFEA, the material properties should be assigned to all elements, and at least one component of the heat or temperature boundary conditions should be applied.

### ■ Data for seepage analysis

The data necessary for seepage analysis are basically composed of the following 2 components:

- material properties: flow conductivities in X, Y and Z directions, conductivity function, water content function.
- seepage boundary conditions: water head (open and confined), flux, point source, and initial water table.

In order to make the problem solvable by VisualFEA, the material properties should be assigned to all elements, and an open or confined head boundary condition should be applied at least to one point of the model, as subsequently described

## General procedure of data assignment

The procedure of assigning data, regardless of type, may be divided into the following several steps:

- 1) Start the data assignment procedure.

Choose one of the items “Element Property”, “Structural Boundary”, “Heat Boundary”, “Load Condition”, and “Element Connectivity” from **Assign** menu. Then, a corresponding dialog box appears.

- 2) Define a data set.

A data set is a data unit carrying specific values of data items. There are a number of items in a data set. Specific values are entered for them using the associated dialog. Details of data items are described for each data type in the following sections.

*You may define as many data sets as necessary. There is no limit to the number of data sets. Only one data set is active at a time. The currently active data set is effective for assignment. When a new data set is created, the set becomes active automatically.*

- 3) Make the desired set active.

If you want to assign a data set which is not currently active, you should make the desired set become the active one by scrolling. A data set may be created, deleted or scrolled using the functions of dialog boxes, which are

explained later in this section.

4) Select objects to assign data.

A data set may be assigned to different types of objects, but not to all object types. There is a default object type for a given data type. The selection tool of default object type is automatically activated when you start a data assignment procedure. You may switch the object selection tool if necessary, but only applicable object selection tools are enabled.

5) Assign data to the selected objects.

Click **Assign** button of the dialog to assign the currently active data set to the selected objects. The newly assigned data are indicated with highlight on the mesh.

*Some types of data may be assigned to surface meshes, volume meshes or curves. However, they are eventually assigned to the nodes or elements within these objects. The default and applicable object types are described in detail for each data type in the following sections.*

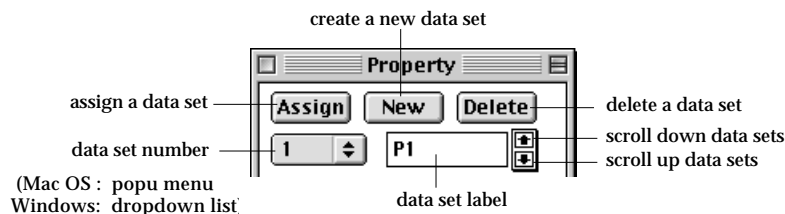
You may repeat step 2) through 5) as many times as necessary without opening the dialog again while the corresponding dialog box is on the main window. You may quit this data assignment procedure by clicking the close box of the dialog, or starting any other procedure.

## Functions common to all types of data assignment

There are a number of functions related to data assignment. Some of them are provided as dialog items, and others as menu items. Among all the various functions related to data assignment, those ones commonly applied to all types of data assignment are described below. Other functions are applied differently depending on the type of data and the analysis subject. So, they are explained in separate sections for various types of data assignment.

### ■ Functions handling data sets

While you are assigning data, a dialog box is always displayed on the main window. The dialog boxes have various items depending on the associated data type and the analysis subject. However, there are items common to all dialog boxes for data assignment as shown below.



All of the above dialog items are for handling data sets including creating, deleting, scrolling and selecting data set. Usage of the items is explained below.




Click this button to create a new data set. The values of the data items in the new set are initially copied from the existing values if applicable, or given with the default values.



The active data set is deleted by clicking this button. The data set is also removed from all the objects assigned with it. Exceptionally in case only one data set is left. Then the set is not deleted, but only its assignment is removed.

The data set numbers are rearranged to fill the gap of the deleted set. The set next to the deleted set becomes active.



Click this button to assign the current data set to the selected objects. (Windows: It is the default push button of the dialog. Therefore, pressing  key has the same effect as clicking this button.)



The number of the data set is displayed as a popup menu (Windows: drop-down list) item. You may choose the desired set using this popup menu (Windows: drop-down list).



The label of the data set. You may label a data set by entering a character string in the text box.



This button is used to scroll up the data set. The current set is scrolled up by this button. Clicking this button once reduces the current set number by one.



This button is used to scroll down the data set. The current set is scrolled down by this button. Clicking this button once increases the current set number by one.




## ■ Entering values of data items

A data set consists of many data items, each of which should be given with specific values. All of the data items are expressed as dialog items: some are editable texts, and others are radio buttons, check boxes, etc.

The first data set is initialized with the default values given by the program. When a new data set is created, each item of the set usually inherits the value of the corresponding item of the previously current set. Some items are interrelated.

You may freely enter or change values of the currently current data set before assigning the set to objects. However, once a data set is assigned to any object, modification of the set is restricted.

## ■ Modifying values of data items

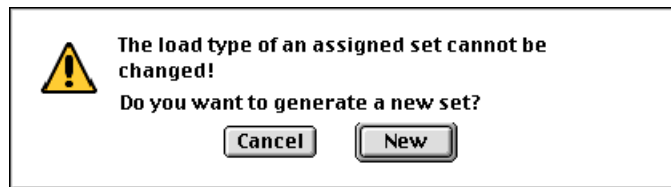
In order to modify the values of a data set, first you must make the set the current set using the dialog items ,  or  as explained above. Only the current

set is displayed on the dialog, and thus, their items can be accessed for modification. If the current set is not yet assigned to any object, the data items can be edited freely. On the other hand, if the set is already assigned, modification of the set either asks for confirmation or is not allowed. So, if you try to modify an item of an assigned data set, you will get a message like this



If you click **Modify** button, the current data become modifiable again so that you may freely edit any item in the set until the modified set is assigned to any object. In this case, the modification will affect all the objects assigned with this data set. If you click **New** button, the current set remains intact, and instead, a new set is created. If you click **Cancel** button, your editing action is ignored, and nothing happens.

There are also cases in which any assigned set cannot not be modified. Then, you will get a notice like this,

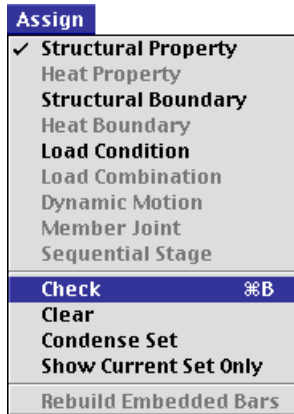


If you click **New** button, a new set is created and becomes current. So, you may edit the data items of the new set and assign the set to the selected objects. If you click **Cancel** button, your editing action is ignored, and nothing happens.

## ■ Assigning data sets

If new data assignment is ready, **Assign** button of the dialog is enabled. Pressing the button assigns the current data set to the selected objects. Data assignment can be canceled by "Undo" command in **Edit** menu before any other action is taken. You may assign a data set to objects which have already been assigned with other data sets. The results of such overlapping assignment depend on the type of data and setting of the related option, and are explained in conjunction with assignment of corresponding data sets.

### ■ Checking data assignment



The state of the data assignment can be checked using “Check” item. When you choose “check” item in **Assign** menu, the data set assigned on the selected object becomes active, and the dialog shows the data items of this set. At the same time, the assignment of the new current set is displayed with highlight.

If two or more objects are selected and they are not assigned with a single data set, the new current set cannot be determined, and thus requested checking may not be realized.

Instead of using this function, you may identify the data set by scrolling the current data set.

### ■ Clearing data assignment

In order to clear data assignment from certain objects, first select the objects to be cleared of data assignment, and choose “Clear” item. All data sets of the current dialog are unassigned from the selected objects. The data sets themselves are not affected by this action.

In order to delete data sets, use **Delete** button in the dialog.

### ■ Condensing data sets

If you want to remove data sets which are not assigned to any object, choose “Condense” item. As for structural boundary conditions, data sets with identical contents will also be merged into one by this action. If the current set is merged into another set, the merged set becomes the current set.

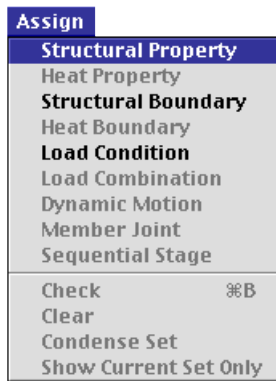
### ■ Ending data assignment

In order to end the current data assignment, simply click the close box of the dialog box, or start any other command.

## Structural Element Properties

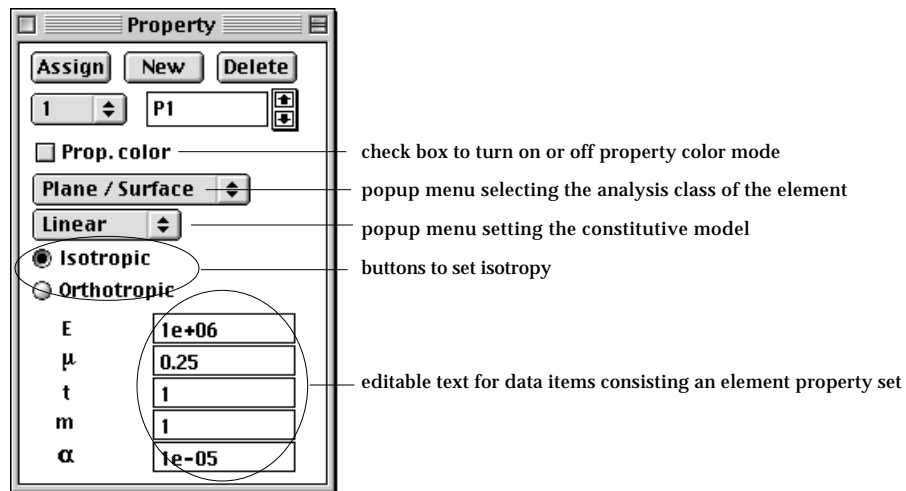
The attributes of the solution domain in finite element analysis are defined by the data representing the characteristics of each element. These data are termed here as element properties. The element property related action is initiated by one of the 2 different menu items, i.e., "Structural Property" and "Heat Property" (or "Seepage Property" for a seepage analysis) depending on the type of property for assignment. In the case of coupled analysis, it is necessary to assign two different types of properties to one model.

### Defining structural element properties



In order to start assigning structural properties, choose "Structural Property" item in **Assign** menu. "Property" dialog appears, and the current state of their assignment are displayed in the main window.

The element properties are defined and assigned by the data unit called element property set. A set consists of many data items, all of which are displayed on "Property" dialog. Structural element properties may include geometric characteristics as well as material properties. The items differ depending on the subject of analysis, or analysis class of the element as described below.



#### ■ Analysis class of element



The first popup menu in "Property" dialog is provided to enable mixing different types of structures in one analysis, as explained in the next section. Using this menu, you can select the type of analysis-related characteristics to impose on the element. It is termed here as "analysis class of element." Each item of the popup menu represents an analysis class of an element.

- "Plane/Surface": plane stress, plane strain, axisymmetric, plate bending, and shell element.
- "Solid": 3-d solid element.
- "Truss": 2-d or 3-d truss element.
- "Frame": 2-d or 3-d frame element.
- "Interface": interface or gap element
- "Slip Bar": slip bar element
- "Embedded bar": embedded bar element
- "Heat": heat conduction element

Only classes compatible for the current project are selectable. In case of frame analysis, for example, "Truss" and "Frame" items are selectable, and others are disabled.

### ■ Constitutive model

The second popup menu in "Property" dialog is to select the constitutive model. This is applicable for material nonlinear analysis of structures. For linear analysis, the menu contains only "Linear" item. The current version of VisualFEA supports only those nonlinearities shown as the menu items. The items vary depending on the analysis class of element as described in the previous section. For "Plane/Surface" and "Solid", the following items are available.

Linear  
Elasto-plastic:V-M  
Elasto-plastic:M-C  
Elasto-plastic:Tresca  
Elasto-plastic:D-P  
Compression Only  
Tension Only

- "Linear": linear elastic model.
- "Elasto-plastic:V-M": Elasto-plastic model with Von Mises yield criterion.
- "Elasto-plastic:M-C": Elasto-plastic model with Mohr-Coulomb yield criterion.
- "Elasto-plastic:Tresca": Elasto-plastic model with Tresca yield criterion.
- "Elasto-plastic:D-P": Elasto-plastic model with Drucker-Prager yield criterion.
- "Compression Only": Linear constitutive relationship for compression, and no stress for tension.
- "Tension Only": Linear constitutive relationship for tension, and no stress for compression.

The following are the items available for interface elements.

Linear Interface  
No Tension Slip  
No Compression Slip  
Gap

- "Linear Interface": linear elastic properties defined in the longitudinal and the thickness direction respectively.
- "No tension slip": The interface delivers compressive normal force across the element, but not tensile force. The maximum resistance against slippage between the two faces across the element is defined by the friction coefficient. The maximum resisting stress is obtained by normal stress multiplied by the friction coefficient.

- "No compression slip": The interface delivers tensile normal force across the element, but not compressive force. The maximum resistance against slippage between the two faces across the element is defined by friction coefficient. The maximum resisting stress is obtained by normal stress multiplied by the friction coefficient.
- "Gap": The interface models a gap between two faces. No force is delivered between the two faces until the gap is diminished by deformation.

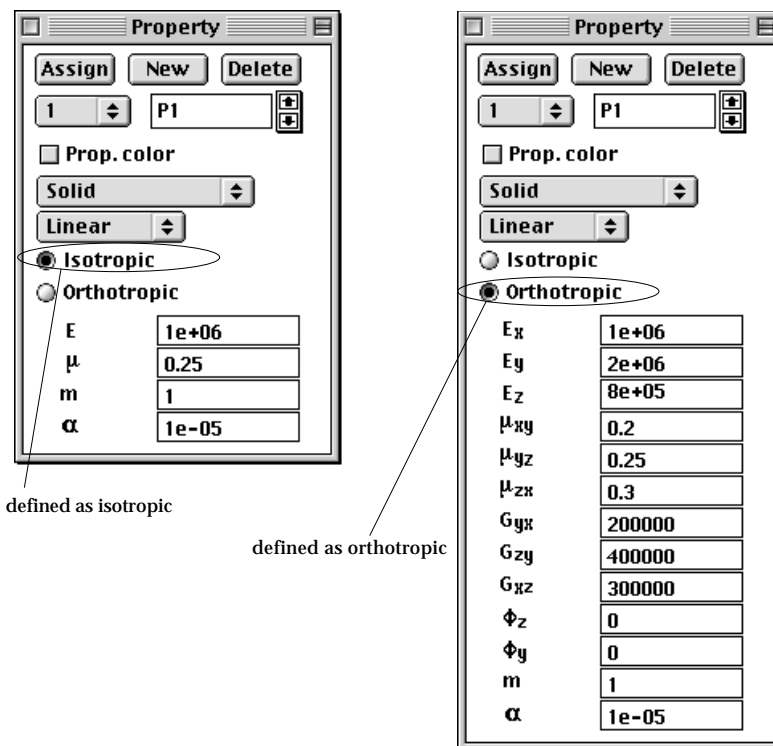
The following are the items available for slip bar elements.

Linear Bonding  
Nonlinear Bonding

- "Linear bonding": The bonding between the slip bar and the surrounding body is represented by linear elastic model.
- "Nonlinear bonding": The bonding between the slip bar and the surrounding body is represented by nonlinear stress-strain relationship.

### ■ Isotropy of the properties

The element properties can be defined as either isotropic or orthotropic using the radio buttons in "Property" dialog. In case the properties are defined as orthotropic, there appear more items in the dialog as shown in the following figure.

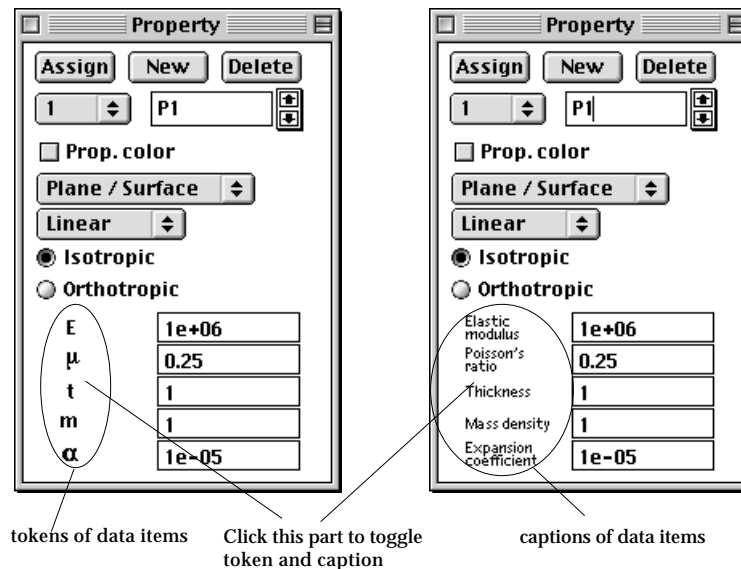


<Data items for isotropic materials and for orthotropic ones>



## ■ Data items of element properties

An element property set consists of a number of data items, which vary depending on the analysis subject, analysis class of the element and the isotropy. As you alter the popup menu items or radio buttons on "Property" dialog, you will notice that the dialog expands or shrinks in its size to accommodate the changing data items properly. Each data item is denoted by a simple token or by a caption. This denotation of data items can be toggled by clicking the part of the dialog as shown in the figure below..

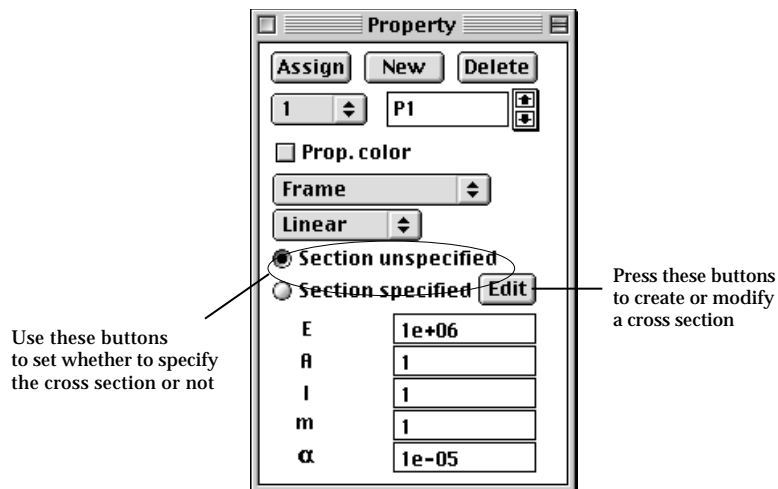


## Defining element properties of truss and frame elements

Frame and truss elements are line segments with specific cross sections. The property set of such elements includes the data related to the cross section such as section area, area moment of inertia, and so on. These data can be defined either directly inserting the values to the corresponding items, or indirectly by specifying the shape and the dimensions of cross section.

"Property" dialog for frame or truss element is different from dialogs for other analysis class elements as shown below.

There are 2 radio buttons to set whether to define the element properties with or without defining the cross section. Turn on "Section unspecified" radio button to define the element property without specifying the cross section. Otherwise, turn on "Section specified" radio button to specify the cross section as explained in the following section.

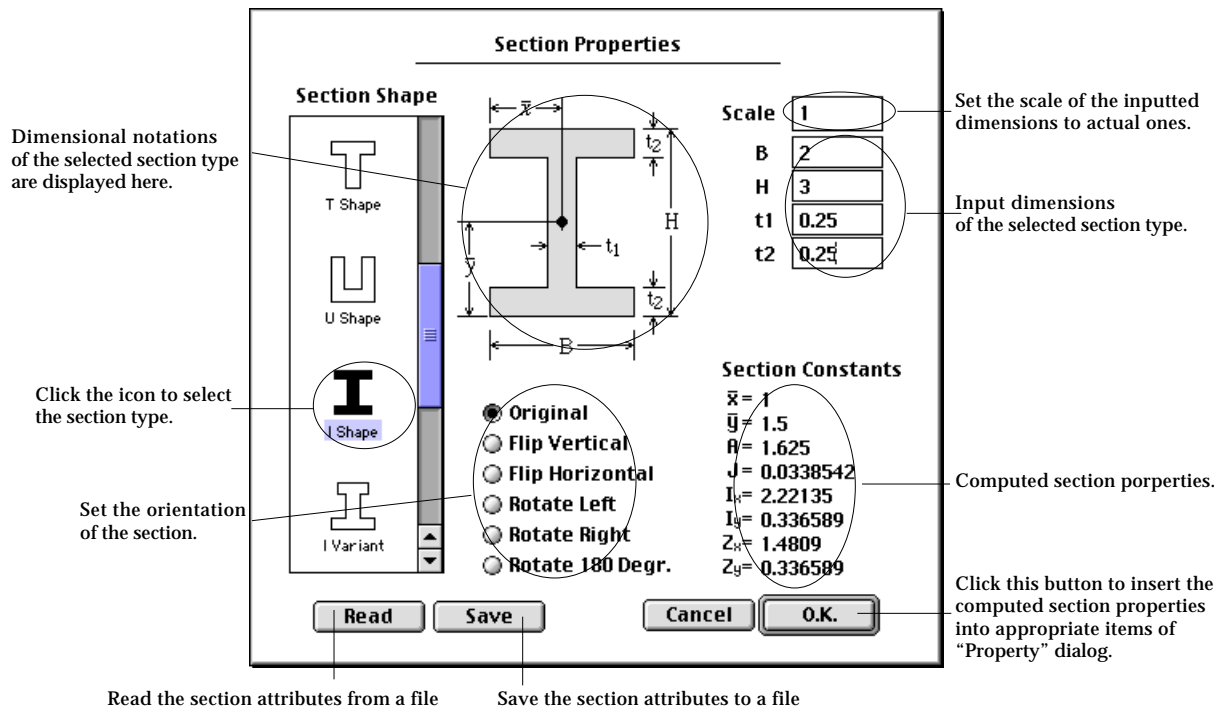


### ■ Defining cross sections of a truss or a frame member

The cross sectional properties of a structural member can be inputted by dimensioning the section of a selected shape in the following order.

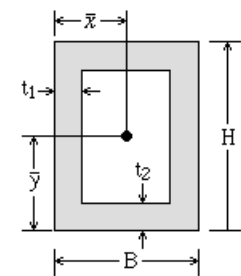
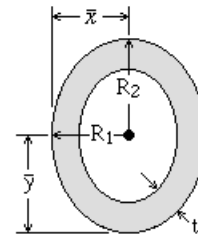
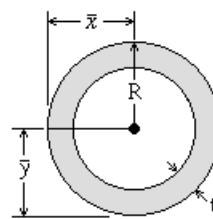
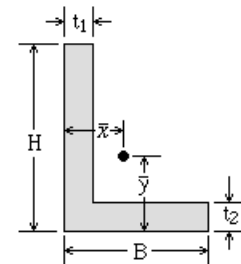
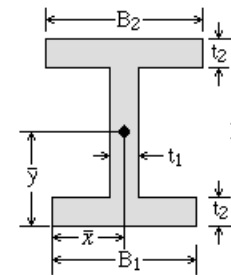
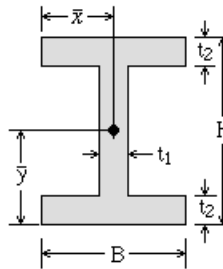
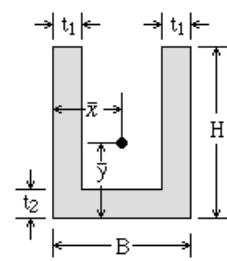
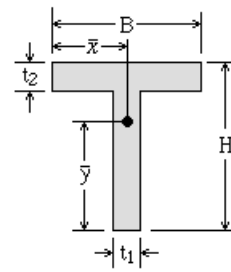
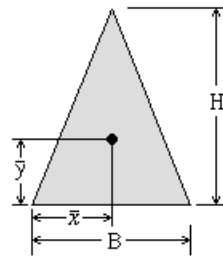
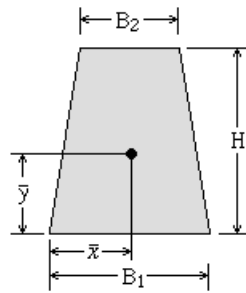
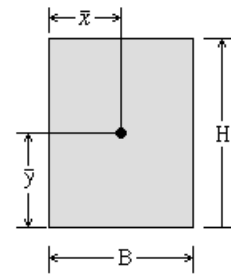
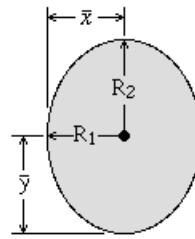
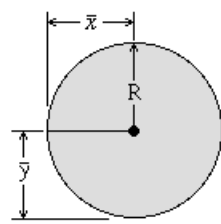
- 1) Click "Defined section" radio button or **Edit** button.

Then, "Section Properties" dialog appears on the screen.



- 2) Scroll the list view of section icons so that the desired icon may be seen.

The current version of VisualFEA has 13 shapes of defined cross section. The list view on the left of the dialog contains the icons of section shape. Only 4 of



<List view icons and dimensional notations of cross sections>

them are shown. The desired one can be made visible by using the scroll bar of the list view.

- 3) Select the desired shape of cross section.

Click the icon of the desired shape. Then, there appears the detailed view of the selected cross section with dimensional notations, and editable text boxes for inputting the dimensions.

- 4) Input dimensions of the section.

Insert the texts into the editable text items representing the dimensions of the cross sections. If all the dimensions are supplied, the computed sectional properties are displayed at the right bottom portion of the dialog.

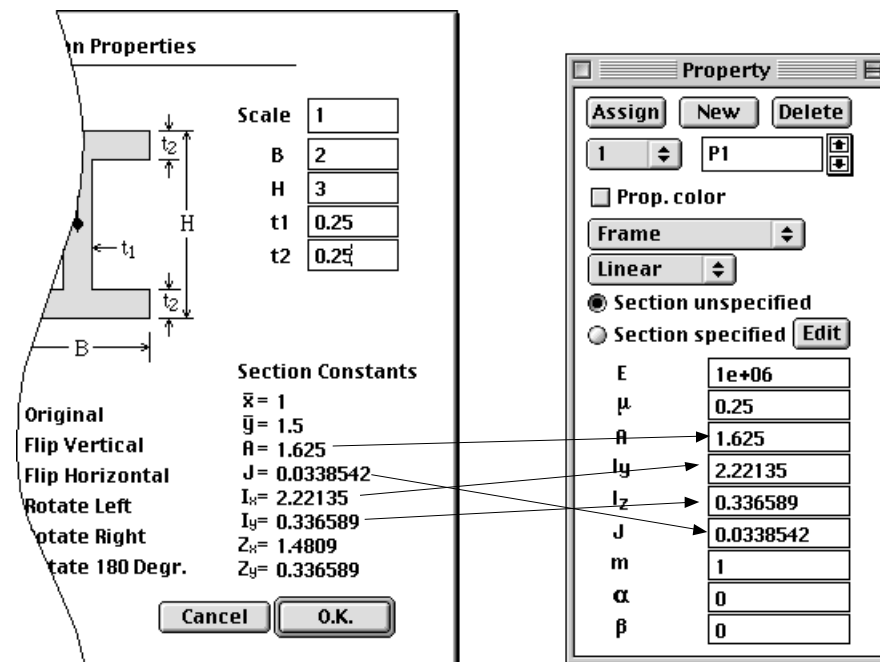
- 5) Set the orientation of the section if necessary

If the desired orientation of the section is not the same as the one displayed on the dialog, that can be altered by choosing one of the following radio buttons:

- "Original" : The orientation is maintained as it is.
- "Flip vertical": The section is flipped about a horizontal axis.
- "Flip horizontal": The section is flipped about a vertical axis.
- "Rotate right": The section is rotated 90° clockwise.
- "Rotate left": The section is rotated 90° counter clockwise.
- "Rotate 180° ": The section is rotated 180°.

- 6) Click **O.K.** button.

The computed cross sectional properties are automatically inserted into the corresponding items of "Property" dialog.



< Automatic insertion of computed cross sectional properties >

Once the cross sectional properties are automatically inserted into the appropriate text items of "Property" dialog, the cross section data is linked to the property set. Thus, "Defined section" radio button is turned on for this set. The cross section of this property set can be edited later by clicking **Edit** button and modifying the text items of "Section Properties" dialog. However, if any item automatically inserted from the cross sectional definition is manually modified, the link between the property set and the section properties is broken and cannot be recovered. Thus, "Undefined section" radio button is turned on for this property set.

### Assigning element properties

Element properties are assigned to objects as a set. Only the currently active set is assigned to selected objects by clicking **Assign** button of the dialog. The general procedures of assigning data are explained in a previous section, and so will not be repeated here. Rules of object selection for property assignment and the display of the assignment are briefly explained below.

#### ■ Selecting objects to assign element properties

Element property sets can be assigned to various objects. However, the data sets are eventually assigned to elements constituting the objects. For example, assigning a data set to a volume is equivalent to assigning the set to all elements within the volume. Therefore, the data sets can be assigned either to individual elements or to the kind of objects which contain the actual elements involved in the finite element analysis. The assignable objects differ depending on the analysis subject or the analysis class of element as summarized in the following table.

< Assignable objects for element properties >

analysis subject	analysis class of element	assignable objects
plane stress/strain axisymmetric, plate bending shell structure	plane/surface	surface mesh, surface element
3-D solid structure	volume	volume mesh, volume element
2-D truss 3-D truss	truss	curve, truss element
2-D rigid frame 3-D rigid frame	frame	curve, frame element
interface	interface	curve
slip bar	slip bar	curve
embedded bar	embedded bar	curve

### ■ Overriding previous assignment of element properties

If you assign a data set to objects on which another data set has already been assigned, the newly assigned set will replace the old one. Thus, one data set at most is assigned to an element.

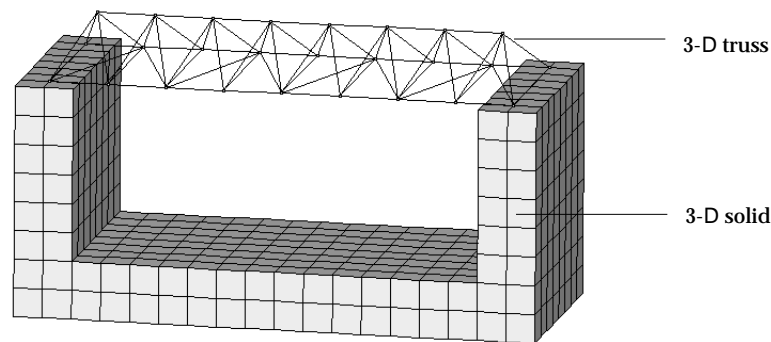
*Because of this overriding behavior, it is more efficient to start assigning property sets from larger regions and to proceed to smaller parts. For example, if most of a mesh has one property and a remaining small part has another, assign one property set to the mesh as a whole first and another set to the relevant elements.*

### ■ Representation of element property assignment

Elements assigned with element properties are drawn in red. The elements assigned with the currently active data set are highlighted in dark red. The assignment is always represented by boundary lines of each element, regardless of the rendering mode.

## Mixing different structural types in one analysis

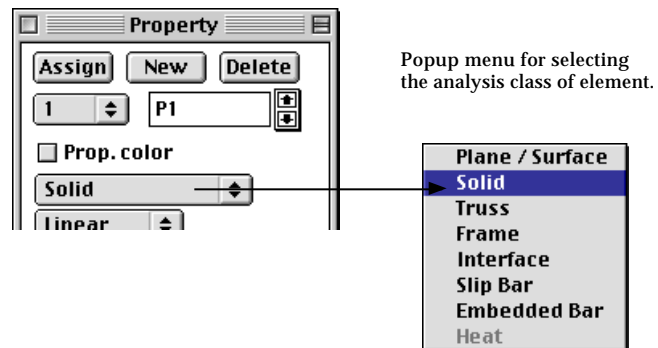
Different types of structures may be included in one analysis. For example, you may model a 3-D solid structure in which 3-D truss members exist as shown below. This can be achieved defining and assigning element property sets of different analysis classes using the popup menu in “Property” dialog.



< Example of a model with mixed structural types >

### ■ Changing the analysis class of element

The popup menu in “Property” dialog has a few items representing analysis classes. When the dialog appears for the first time, the popup menu is set to the item of analysis class corresponding to the current analysis subjects. If you change the item, all the dialog items are also altered accordingly. At the same time, the assignable objects are changed, and the relevant object selection tool is automatically activated.



### ■ Applicable analysis classes of element

In VisualFEA, not all types of structures can be mixed with each other. There are applicable types for a given analysis subject. In the popup menu shown above, only the items of applicable analysis classes are enabled. Listed in the following table are the structural types which can be included for given analysis subjects.

< Structural types applicable for a given analysis subject >

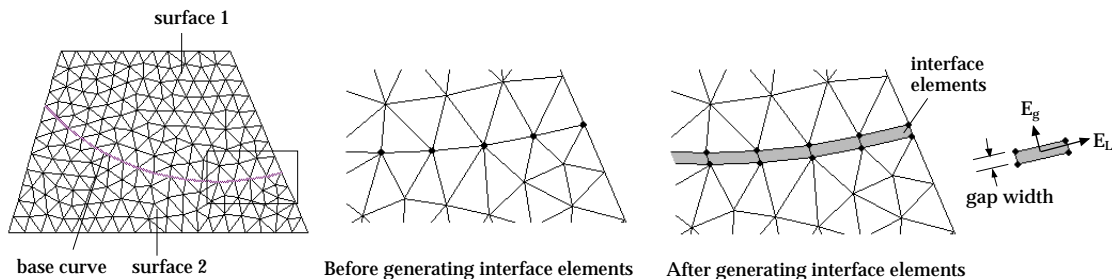
analysis subject	applicable structural types	
	structural type	analysis class of element
plane stress/strain axisymmetric	2-D truss 2-D frame interface slip bar embedded bar	truss frame interface slip bar embedded bar
3-D solid structure	2-D truss 2-D frame shell slip bar embedded bar	truss frame plane/surface slip bar embedded bar
2-D rigid frame	2-D truss	truss
3-D rigid frame	3-D truss	truss

## Using interface elements

Interface elements are used in modeling the behavior of gaps with negligible width. They cannot be used alone, but may be included in plane stress, plane strain or axisymmetric problems. (The current version of VisualFEA supports only 2 dimensional interface elements.)

### ■ Characteristics of interface elements

Interface elements fill the gap separating two adjacent surfaces. They connect the nodes on the boundaries of both surfaces. VisualFEA supports only 4 node interface elements. The structural behavior of the gap can be controlled by the stiffness of the interface elements in width and length directions. These stiffness are determined by the moduli of elasticity in both directions, and the gap width of the elements. A pair of nodes facing with each other are separated by as much as the specified gap width. Assigning zero or very small value of gap width or large modulus of elasticity in width direction will restrain the relative movement in width direction, of the pair of nodes. On the other hand, the modulus of elasticity in length direction controls the relative movement in length direction. So, you may model the desired behavior of the gap, or relative movement of the contacting surfaces by assigning appropriate values to the properties of interface elements.



< Characteristics of interface elements >

### ■ Creating interface elements

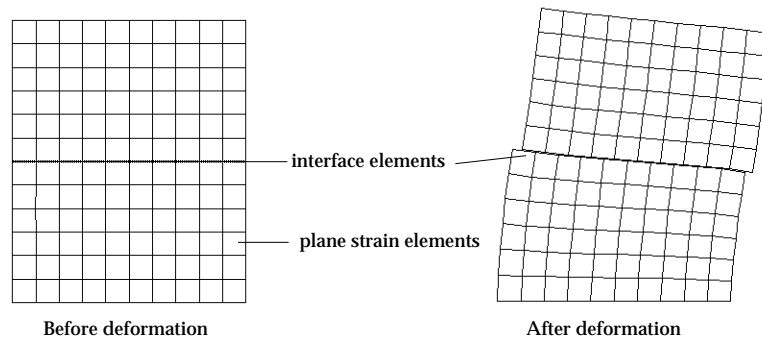
Interface elements are created somewhat differently from other structural types of elements. They are not created by mesh generation or curve input, but by assignment of element properties as explained below.

- 1) Choose "Interface" item from the popup menu in "Property" dialog.  
The data items in "Property" dialog are altered.
- 2) Enter the value for each data item.  
The interface element can be made to represent specific behavior by entering appropriate values for the data items.



- modulus of elasticity in length direction ( $E_L$ ): The Young's modulus of elasticity in the direction parallel to the length of the interface elements.
  - modulus of elasticity in gap width direction ( $E_g$ ): The Young's modulus of elasticity in the direction normal to the length of the interface elements.
  - gap width (**gap**): The width of interface of elements. This value should be small as compared with the dimension of the whole structure.
  - thickness (**t**): This item is applicable only for plane stress problems. Usually, this value is set as identical to the thickness of the main structure in which the interface elements are embedded.
- 3) Select one or more base curves to assign interface elements.  
The selected base curves are to be used as the basis of the interface elements. Therefore, they should be serially connected, and completely embedded within the main structure.
- 4) Click **Assign** button in the dialog.  
Interface elements are created on the selected curves. The active set of element properties are assigned to the newly created elements.

When interface elements are created, the basis curves together with the nodes on them are duplicated. The interface elements are composed of the nodes on the new and existing base curves.



< Example of a plane strain case with interface elements >

### ■ Deleting interface elements

Interface elements can be deleted only by deleting the set of element property assigned to them. The meshes are restored to the state before creating interface elements. That is, the surface meshes separated by the interface elements are joined again.

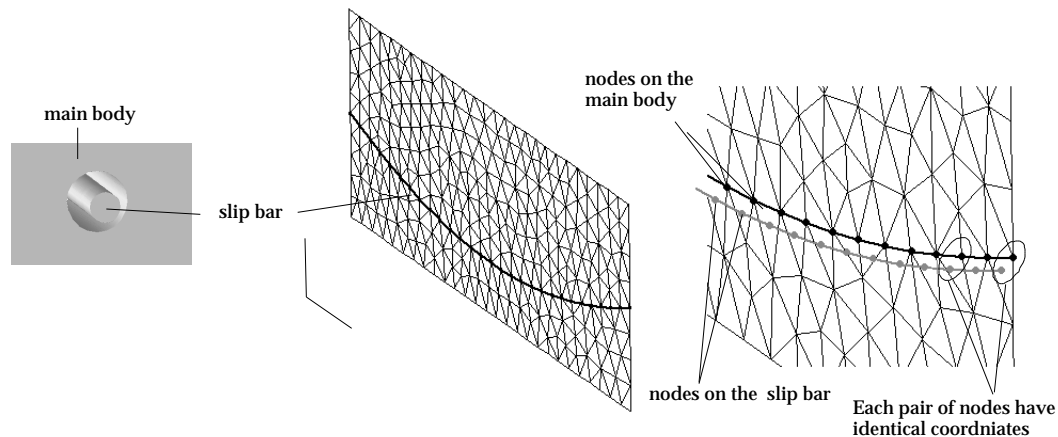
## Using slip bars

Slip bar elements are used in modeling bars, cables, rods or other reinforcement in planar or 3-D solid structures. For example, tendons within prestressed concrete beams can be best modeled by using slip bars. Slip bars cannot be used alone, but may be included in plane stress, plane strain, axisymmetric, or 3-D solid structures.

### ■ Characteristics of slip bar elements

Slip bars are objects embedded within, but independent from, a body of planar or 3-D solid. There can be bonding effects between the slip bar and the main body. These behaviors are modeled by using slip bar elements, which are a kind of truss elements combined with bonding effects. Conceptually, nodes on slip bar are surrounded by the nodes paired within the main body. Numerically, a pair of nodes, one in the slip bar and the other in the main body, are independent from each other, but share identical coordinates.

A pair of nodes are allowed to slip against each other in the direction along the slip bar element. But they are forced to move together in its normal direction. The bonding effect between the slip bar and the main body is represented by the stiffness against the slip of paired nodes.



< Concept of slip bar elements >

### ■ Creating slip bar elements

Slip bar elements are created somewhat differently from other structural types of elements, but similar to interface elements. They are not created by mesh generation or curve input, but by assignment of element properties as explained below.

- 1) Choose "Slip Bar" item from the popup menu in "Property" dialog.

The data items in “Property” dialog are altered. A property set of slip bar element is always accompanied by an annexed property set. The annexed set has the attributes for truss elements representing truss behavior of the slip bar. This annexed set can be modified but cannot be assigned independently.

- 2) Enter the value for each data item.

The slip bar element can be made to represent specific behavior by entering appropriate values for the data items.

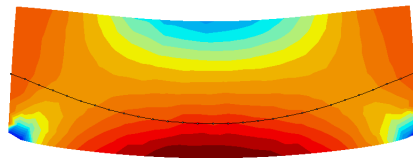
- modulus of elasticity of the slip bar ( $E_{\text{bar}}$ ): The Young’s modulus of elasticity of the slip bar elements.
- modulus of elasticity representing ( $E_{\text{bond}}$ ): The Young’s modulus of elasticity representing the bonding effect between the bar and the solid.
- section area (A): The section area of the bar.

- 3) Select one or more curves to assign slip bar elements.

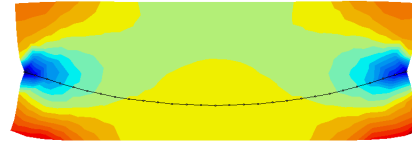
The selected curves are to be used as the basis of the slip bar elements. Therefore, they should be serially connected, and completely slip within the main body.

- 4) Click **Assign** button in the dialog.

Slip bar elements are created on the selected curves. The active set of element properties are assigned to the newly created elements.



Without prestressing



With prestressing

< Example of modeling a prestressed beam using slip bar elements >

## ■ Deleting slip bar elements

Slip bar elements can be deleted only by deleting the set of element property assigned to them. When a property set for slip bar is deleted, its annexed set is also removed.

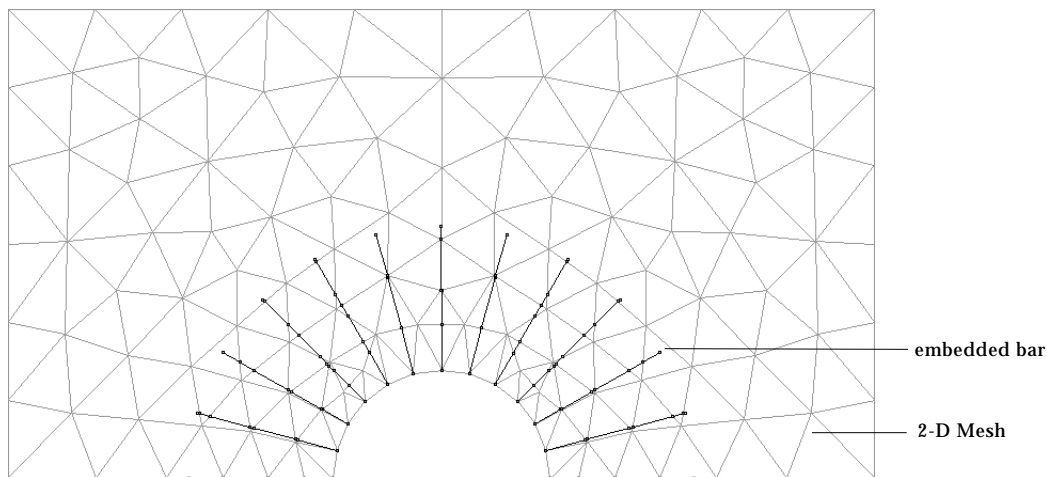
## Using embedded bars

Similarly to slip bars, embedded bars are also used in modeling stiffening material embedded within planar or 3-D solid structures. Embedded bars are more convenient and flexible to use than the slip bars, because they can be arbitrarily placed within continuum. Embedded bars cannot be used alone, but may be included in plane stress, plane strain, axisymmetric, or 3-D solid structures.

### ■ Characteristics of embedded bar elements

Embedded bars and slip bars are similar in their usage as stiffening line segments embedded within continuum, but different in their characteristics and in their modeling method. A slip bar is an element which has nodes and its own displacement field, while an embedded bar is not an independent element but a stiffening segment embedded within continuum elements. Thus, an embedded bar has neither independent displacement field nor nodes. Its displacements are defined only through the deformation of the surrounding elements. The stiffness of the embedded bars are directly added to the surrounding elements. The strains of embedded bars are computed from the displacements of the surrounding elements.


An embedded bar can pass through a number of elements without restriction in the position or in the direction of its path. Embedded bars may be placed freely within continuum without involving nodal connectivity conditions as shown in the figures of this and the next pages. So, embedded bars can be employed conveniently in modeling complex reinforcement within continuums, especially in combination with unstructured meshes.



< An example of embedded bars placed within a 2-D unstructured mesh >

### ■ Creating embedded bars

Any straight line can be defined as an embedded bar by assignment of element properties as explained below.

- 1) Create straight lines using the line tool .

Many embedded bars with a certain regular arrangement may be created efficiently by using duplication of curves. The figure at the bottom of this page shows an example of thousands of embedded bars created mostly by "Duplicate and Move" and "Duplicate and Revolve". Refer to "Duplicating curves and surface primitives" section of Chapter 3.

- 2) Start property assignment procedure.

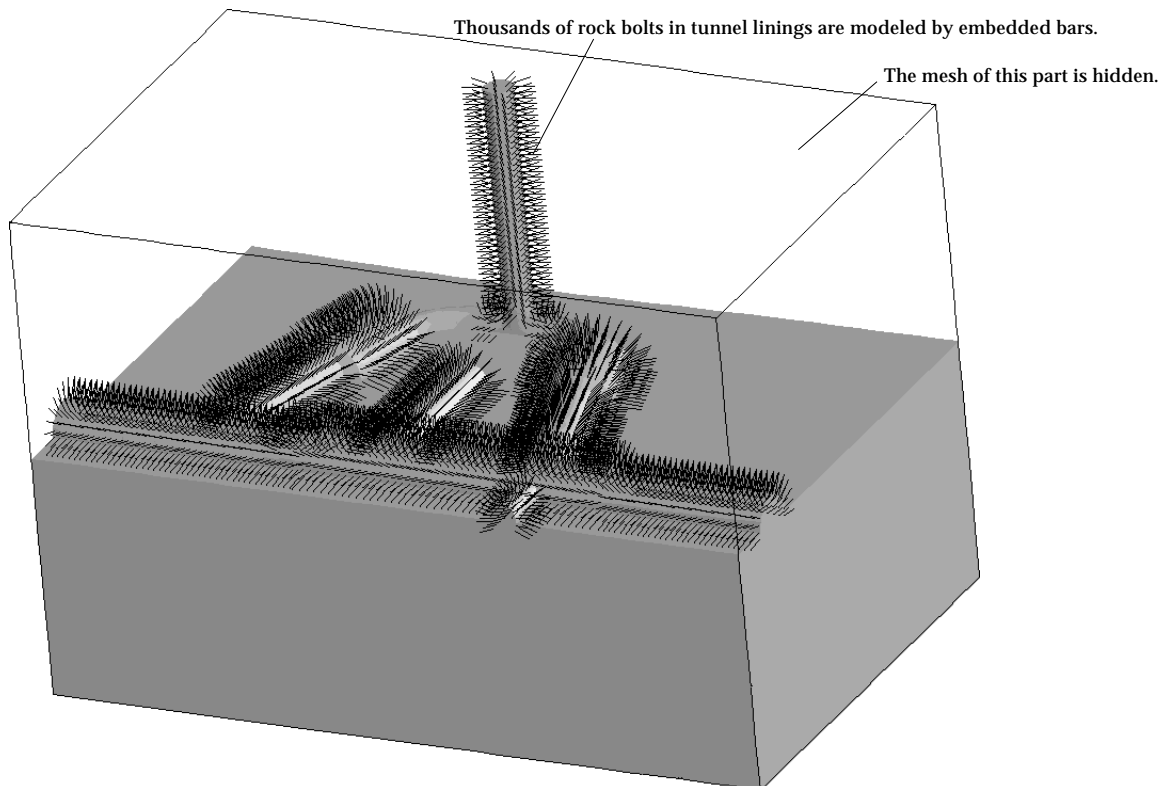
Choose "Element Property" item from **Assign** menu.

- 3) Choose "Embedded Bar" item from the popup menu in "Property" dialog.

The data items in "Property" dialog are altered to the items of embedded bars.

- 4) Enter the value for each data item.

The data items are related to the axial stiffness of the embedded bar. It is assumed that the section area of the embedded bar is subtracted from the embedding elements. Let  $A_e$  and  $E_e$  be the section area and the elastic



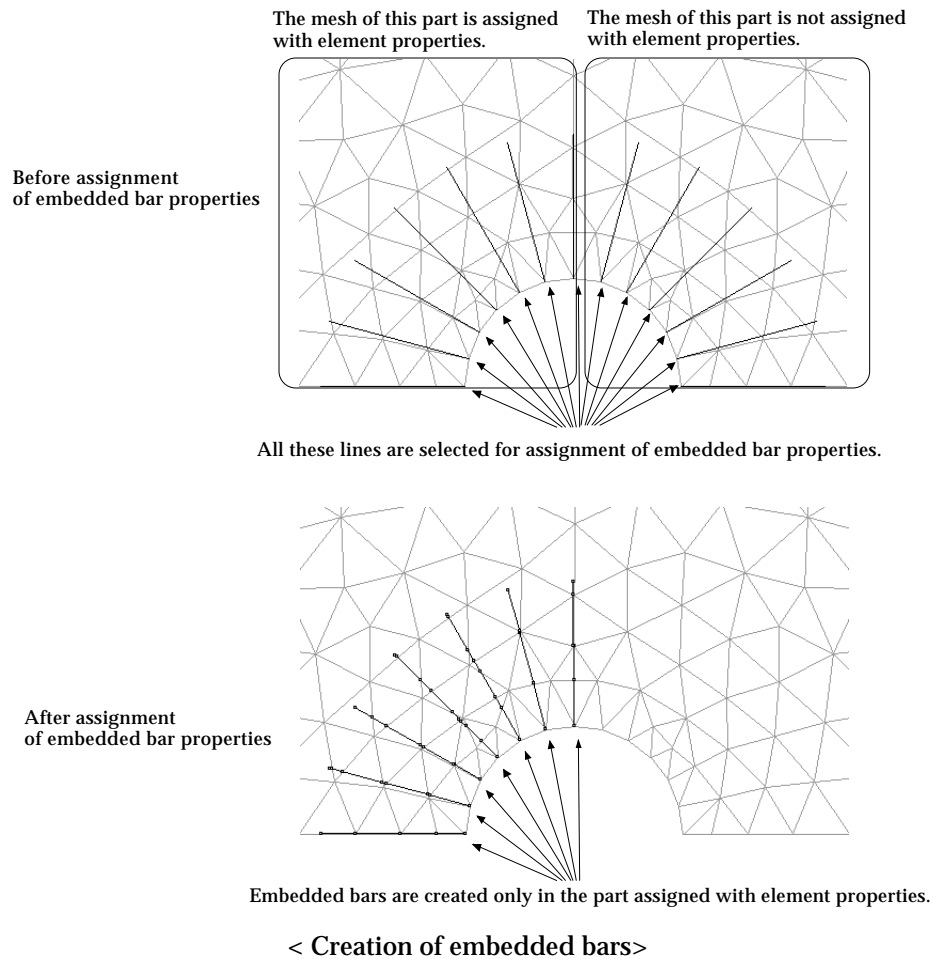
< An example of embedded bars in a 3-D solid model >

modulus of the embedded bar, and  $E_o$  be the elastic modulus of the embedding element. Then, the stiffness contribution of the embedding bar is

$$k = A_e (E_e - E_o)$$

- 5) Select one or more lines to assign embedded bar properties.
- 6) Click **Assign** button in the dialog.

The selected curves turn into embedded bars only when the surrounding continuum elements are assigned with element properties as illustrated by the figure below. The intersections of embedded bars and element edges are marked by small circles.



### ■ Deleting embedded bar

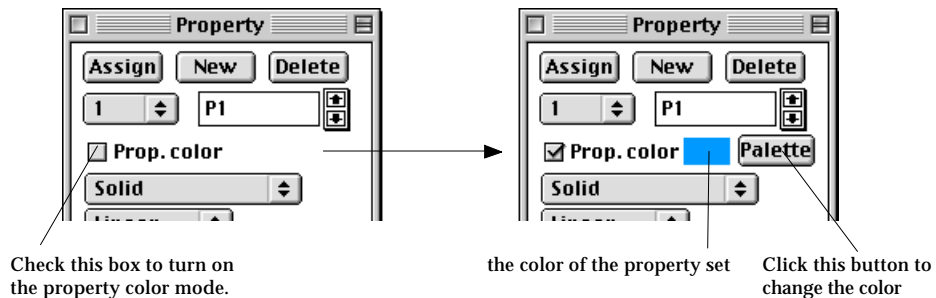
The embedded bar properties are removed by deleting the embedded bar property set, or by clearing the assigned property set. The line segments remain undeleted, although the property assignment is removed.

## Color coding of property sets

Property sets can be coded by colors. If the property color mode is turned on, assignment of property set to each part of the model is distinguished by the color used for rendering the part either in wireframe or in shaded image. Any parts without property assignment are rendered in gray. The default color codes are predefined for the first 16 property sets. The color of each set can be altered using the color palette functions of the corresponding operating system.

### ■ Turning on the property color mode

The property color mode is turned on by checking the "Prop. color" box of "Property" dialog. The color box filled with the color of the current property set and "Palette" button appear on the dialog when the property color mode is turned on.



### ■ Changing or setting the property color

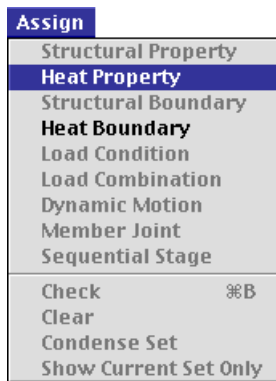
Click **Palette** button to change the property color. Then, the color palette dialog appears on the screen. The default color for the first 16 sets are shown in the palette. (Windows only) You may change the default color, or set a new color code for the property sets beyond the first 16 sets.

It should be noted that gray is the reserved color for rendering the part without property assignment, and cannot be used as a property set color.

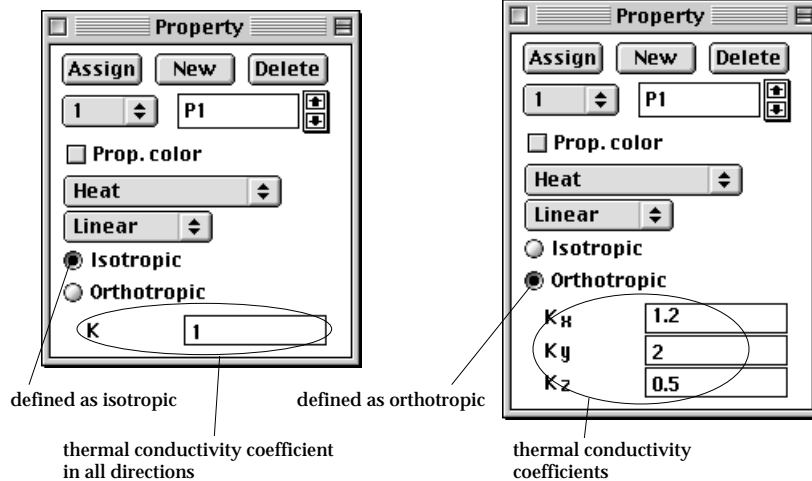
## Heat Conduction and Seepage Properties

The element properties for heat conduction or seepage analysis are defined and assigned in the similar way as that of structural analysis. The related actions are initiated by "Heat Property" (or "Seepage Property") item of **Assign** menu.

### Heat conduction properties



If the analysis subject is defined as a heat conduction problem, "Heat Property" item appears in **Assign** menu and is enabled. Choose the item in order to start assigning heat conduction properties. "Property" dialog appears, and the current state of their assignment are displayed in the main window.



### ■ Defining heat conduction properties

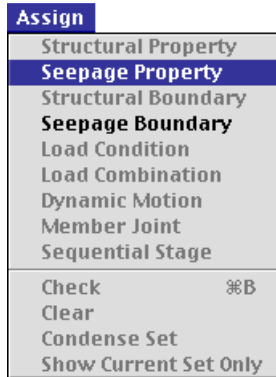
Heat conduction properties include the thermal conductivity coefficients in X and Y directions for 2-D heat conduction problems, and X, Y and Z directions for 3-D heat conduction problems. If the conductivity is isotropic, click "Isotropic" button in the dialog. Then, the dialog shows only one editable text item "K". If the conductivity is orthotropic, click "Orthotropic" button. Then, the dialog shows the editable text item of the thermal conductivity coefficient for each of the coordinate axis directions. Inserting the value(s) in the editable text box(es) complete the defining of heat conduction properties.

### ■ Assigning heat conduction properties

Heat conduction properties can be assigned only to surface meshes in the case of plane and axisymmetric heat conduction problems, and only to volume meshes in the case of 3-D heat conduction problems.



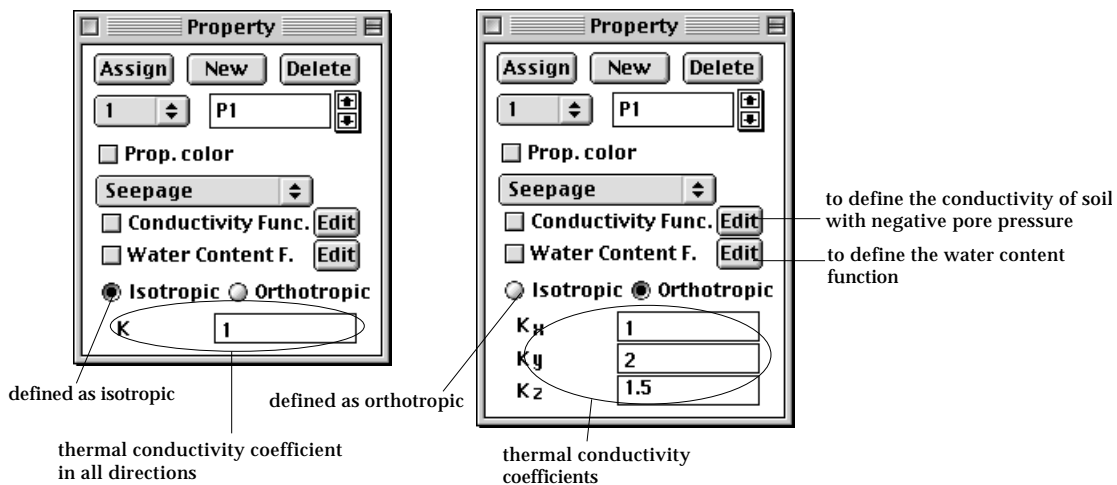
## Seepage properties



If the analysis subject is defined as a seepage problem, "Seepage Property" item appears in **Assign** menu and is enabled. Choose the item in order to start assigning seepage properties. "Property" dialog appears, and the current state of their assignment are displayed in the main window.

### ■ Defining seepage properties

Seepage properties include the hydraulic conductivity coefficients in X and Y directions for 2-D seepage problems, and in X, Y and Z directions for 3-D seepage problems. The conductivity can be represented by one value for isotropic material as shown in the figure below. Check "Isotropic" radio button to input the conductivity coefficients as isotropic material.



Seepage properties include conductivity functions for unconfined seepage analysis. The conductivity function defines the ratio of the reduced conductivity to the full conductivity as a function of pore pressure.

### ■ Defining hydraulic conductivity functions

A conductivity function is associated with each one of the seepage property sets. Click **Edit** button in the "Property" dialog to input a conductivity function for the current property set. Then, "Hydraulic Conductivity Function" dialog appears on the screen. The conductivity function can be defined either by equation or by table using this dialog as described below.

- 1) Click **Edit** button of "Property" dialog to open "Hydraulic Conductivity Function" dialog.

The dialog opens with default settings. The dialog consists of 2 parts as

shown in the figure below. The first part is used to define the hydraulic conductivity function by an equation, and the other part to define the function in tabular form.

**Hydraulic Conductivity Function**

☒ **Defined by equation**

$$c = \frac{1}{1 + a|P|^n}$$

$a = 0.1$   
 $n = 2$

$c$  = conductivity factor  
 $a$  = Gardner coefficient  
 $P$  = negative pore pressure

☐ **Defined by table**

No.	Pore Pressure	Conductivity Factor
1	0	1

0 Add Delete Save Import Graph Cancel O.K.

- 2) Turn on "Define by equation" radio button to define the conductivity function by an equation.

The hydraulic conductivity factor  $c$  is computed as a function of negative pore pressure  $P$  by the equation,

$$c = \frac{1}{1 + a|P|^n}$$

where  $a$  is the Gardner coefficient and  $n$  is a power factor.

The conductivity factor implies the ratio of the reduced conductivity in negative pore pressure region to the full conductivity.

- 3) Insert the Gardener coefficient  $a$ , and power factor  $n$ .

The value of  $a$  is normally assumed between 0.001 and 0.1, and the value of  $n$  between 2 and 4.

- 4) Turn on "Define by table" radio button to define the conductivity function in tabular form.

The hydraulic conductivity is defined by the tabulated relation between the

pore pressure and the conductivity factor. The conductivity factor is computed by linear interpolation of the given pore pressures and the conductivity factors.

☒ Defined by table

No.	Pore Pressure	Conductivity Factor
1	-1	0.5
2	-2	0.2
3	-5	0.1
4	-10	0.05
5	-20	0.01

0.01

Add Delete

Save Import

Graph Cancel O.K.

- 5) Click **Add** button to add a new row in the table, and **Delete** button remove a row.

The buttons are enabled when "Defined by table" radio button is turned on. After creating a new row, insert the values of the row using the editable text box.

- 6) Click **O.K.** button to complete defining the conductivity function.

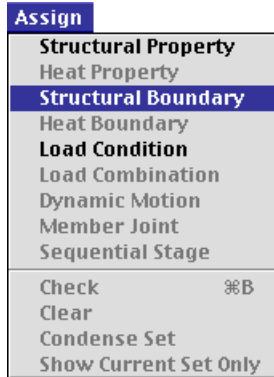
The "Hydraulic Conductivity Function" dialog closes, and the conductivity function is added to the current seepage property set. The check box of "Conductivity Func." item of the "Property" dialog is marked to indicate that the hydraulic function of the property set is defined.

### ■ Assigning seepage properties

In order to assign seepage properties, first select the object to assign, and click **Assign** button of the "Property" dialog. Seepage properties can be assigned only to surface meshes in the case of plane and axisymmetric seepage problems, and only to volume meshes in the case of 3-D seepage problems.

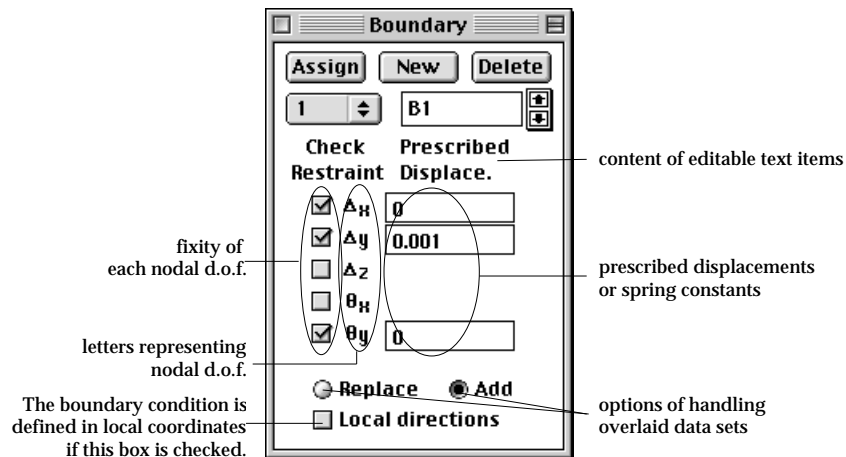
## Structural Boundary Conditions

Structural boundary conditions specify the way a structure is supported. The data items of the structural boundary condition are related to the constraining status of nodes at the boundaries. The status is defined for each nodal d.o.f., and is one of the following four.



- free : free to move. In other words, no constraint is imposed.
- free with spring constant : partially constrained state like hinges with friction, elastic supports, etc. The degrees of constraint are represented by spring constants.
- fixed: No movement is allowed. The 0 displacement is prescribed.
- fixed with prescribed displacements : initial displacement, settlements, etc. The displacement is forced to be the prescribed value.

In order to start assigning structural boundary conditions, choose “Structural Boundary” item in **Assign** menu. “Struct Boundary” dialog appears, and current state of their assignment are displayed in the main window.



## Defining structural boundary conditions

You must first create and define sets of boundary conditions before assigning them on the structural boundaries. All the data items of the active set are displayed on “Struct Boundary” dialog and can be entered or modified using the dialog.

### ■ Nodal degrees of freedom

The boundary condition dictates the state of each nodal degree of freedom. The nodal d.o.f. differ depending on the subject of analysis. If more than one structural type is involved in the analysis, the items are extended to hold all associated structural types.

2-D truss	3-D truss	2-D rigid frame	3-D rigid frame
<input checked="" type="checkbox"/> $\Delta_H$	<input checked="" type="checkbox"/> $\Delta_H$	<input checked="" type="checkbox"/> $\Delta_H$	<input checked="" type="checkbox"/> $\Delta_H$
<input checked="" type="checkbox"/> $\Delta_y$	<input checked="" type="checkbox"/> $\Delta_y$	<input checked="" type="checkbox"/> $\Delta_y$	<input checked="" type="checkbox"/> $\Delta_y$
	<input checked="" type="checkbox"/> $\Delta_z$	<input checked="" type="checkbox"/> $\theta_z$	<input checked="" type="checkbox"/> $\Delta_z$
			<input checked="" type="checkbox"/> $\theta_H$
			<input checked="" type="checkbox"/> $\theta_y$
			<input checked="" type="checkbox"/> $\theta_z$
plane stress plane strain axisymmetric	plate bending	shell	3-D solid
<input checked="" type="checkbox"/> $\Delta_H$	<input checked="" type="checkbox"/> $\Delta_z$	<input checked="" type="checkbox"/> $\Delta_H$	<input checked="" type="checkbox"/> $\Delta_H$
<input checked="" type="checkbox"/> $\Delta_y$	<input checked="" type="checkbox"/> $\theta_H$	<input checked="" type="checkbox"/> $\Delta_y$	<input checked="" type="checkbox"/> $\Delta_y$
	<input checked="" type="checkbox"/> $\theta_y$	<input checked="" type="checkbox"/> $\Delta_z$	<input checked="" type="checkbox"/> $\Delta_z$
		<input checked="" type="checkbox"/> $\theta_H$	
		<input checked="" type="checkbox"/> $\theta_y$	

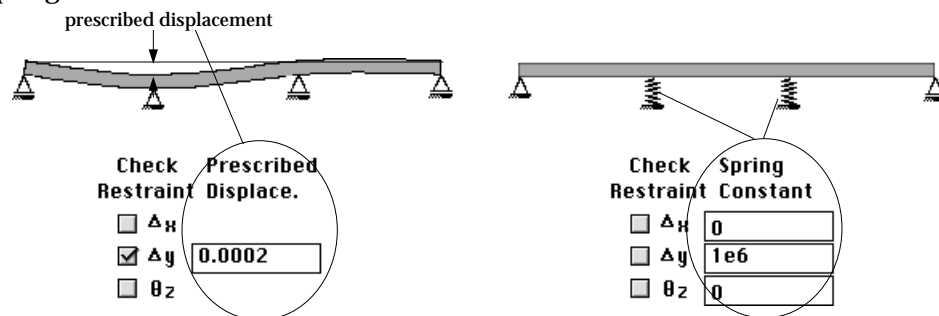
< Nodal d.o.f for various analysis subjects >

### ■ Data items of structural boundary condition

A structural boundary condition set has a few data items. They consist of nodal fixity, initial displacements or spring constants defined for each d.o.f.

- fixity: the state of nodal constraint. Either “free” or “fixed” condition is specified for each nodal d.o.f., and indicated in the check boxes of the dialog. If a box is checked, the corresponding d.o.f. is fixed.
- prescribed displacement: nodal displacements given prior to solving the problem. Nodal displacements can be specified only for fixed d.o.f.
- spring constant: This value represents the degree of partial constraint. The spring constants can be specified only for free d.o.f.

The following figure shows examples of applying prescribed displacements and spring constants.



< Nodal d.o.f for various analysis subjects >

### ■ Entering data items of boundary conditions

The fixity of each nodal d.o.f. is entered by clicking the check box in front of the symbol indicating the respective d.o.f. The prescribed displacements and the

spring constants are entered within the corresponding text boxes. As shown below, not all the text boxes are displayed. Some of them are hidden. Displayed items are related to the heading of the text items. There are 3 headings:

- “Prescribed Displace.”: This heading indicates that only text items for prescribed displacements are shown. Because displacements can be prescribed only for fixed d.o.f., the text items of fixed d.o.f. are displayed.
- “Spring Constant”: This heading indicates that only text items for spring constants are shown. Because spring constants can be applied only for free d.o.f., the text items of free d.o.f. are displayed.
- “Mixed Values”: This heading indicates that text items for spring constants as well as for prescribed displacement are shown. Thus, all text items are displayed.

The heading is initially set as “Prescribed Displace.” You may scroll through the above three headings by clicking any point over the heading. But, only applicable ones will be scrolled. For example, if non-zero values are entered in prescribed displacements, “Spring Constant” will be skipped.

Click here to scroll the heading.

Check Restraint		Prescribed Displace.
<input checked="" type="checkbox"/>	$\Delta_H$	0
<input checked="" type="checkbox"/>	$\Delta_y$	0.006
<input type="checkbox"/>	$\Delta_z$	
<input type="checkbox"/>	$\theta_H$	
<input checked="" type="checkbox"/>	$\theta_y$	0

These checked d.o.f. are fixed.  
These unchecked d.o.f. are free.

indicates that the prescribed displacements are entered in the following text boxes.  
Enter the prescribed displacements in these boxes. Leave 0 for no displacement.

Check Restraint		Spring Constant
<input checked="" type="checkbox"/>	$\Delta_H$	
<input checked="" type="checkbox"/>	$\Delta_y$	
<input type="checkbox"/>	$\Delta_z$	25000
<input type="checkbox"/>	$\theta_H$	0
<input checked="" type="checkbox"/>	$\theta_y$	

indicates that the spring constants are entered in the following text boxes.  
Enter the spring constants in these boxes. Leave 0 for no spring constant, i.e., for completely free d.o.f.

Check Restraint		Mixed Values
<input checked="" type="checkbox"/>	$\Delta_H$	0
<input checked="" type="checkbox"/>	$\Delta_y$	0.006
<input type="checkbox"/>	$\Delta_z$	25000
<input type="checkbox"/>	$\theta_H$	0
<input checked="" type="checkbox"/>	$\theta_y$	0

indicates that either prescribed displacements or spring constants are entered depending on the fixity of the corresponding d.o.f.

< Headings of editable text items >

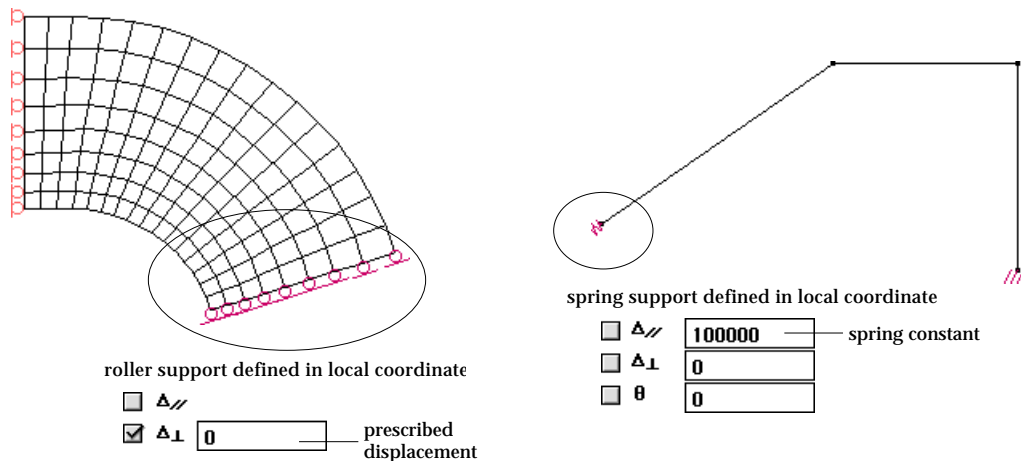
### ■ Defining boundary conditions in local coordinates

The structural boundary conditions can be defined in either in global coordinates or in local coordinates. The local coordinates are based on the axis of a structural member, the tangent direction of a surface edge, or the normal to a surface. Check "Local direction" box of "Struct Boundary" dialog in order to use local coordinates for a boundary condition set. At the moment, the "Local direction" is checked, the d.o.f. labels are altered as shown in the figure below.

2-D truss plane stress plane strain axisymmetric	3-D truss 3-D solid	2-D frame	
<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{\perp}$	<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> $\Delta_{/z}$	<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{\perp}$ <input checked="" type="checkbox"/> $\theta$	$\Delta_{//}$ translation in the axial direction $\Delta_{\perp}$ translation in the direction normal to axis (truss/ frame) or to surface (shell)
3-D frame	shell	plate	
<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> $\Delta_{/z}$ <input checked="" type="checkbox"/> $\theta_{//}$ <input checked="" type="checkbox"/> $\theta_{/y}$ <input checked="" type="checkbox"/> $\theta_{/z}$	<input checked="" type="checkbox"/> $\Delta_{/H}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> $\Delta_{\perp}$ <input checked="" type="checkbox"/> $\theta_{/H}$ <input checked="" type="checkbox"/> $\theta_{/y}$	local coordinates not defined	$\Delta_{/H}$ translation in the local x direction(shell) $\Delta_{/y}$ translation in the local y direction $\Delta_{/z}$ translation in the local z direction $\theta_{//}$ axial rotation (shell) $\theta_{/y}$ rotation about the local y axis(shell) $\theta_{/z}$ rotation about the local z axis(shell)

#### < Boundary constraints in local coordinates >

Use of local coordinates may be most appropriate for roller support and spring boundary condition which are illustrated in the figure below. The prescribed displacements and the spring constants are entered for the local coordinates in the same way as for the global coordinates.



#### < Examples of boundary conditions defined in local coordinates >

In most cases, only the axial direction or the normal direction is concerned with the boundary conditions in local coordinates. The axial and normal directions at every node is determined in association with the object to which the boundary condition is assigned.

- **curve:** In the case the boundary condition is assigned to a curve, the axial (//) direction is defined as tangent to the curve at every node on the curve. For planar models, the normal ( ) direction is uniquely determined from the axial direction. For 3-D models, the 2 normal (/y and /z) directions are determined by vector operations of the unit axial vector,  $\vec{t}$  and the unit vector in the global Z direction,  $\vec{k}$ .

$$\vec{v}_y = \vec{k} \times \vec{t}$$

$$\vec{v}_z = \vec{t} \times \vec{v}_y$$

where  $\vec{v}_y$  and  $\vec{v}_z$  represent unit vectors in the local y and z directions respectively.

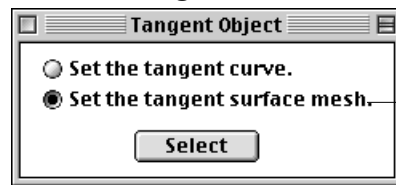
- **surface mesh:** In the case the boundary condition is assigned to a surface mesh, the normal direction ( ) is first determined at every node on the surface mesh. And the 2 other (/x and /y) directions are determined by vector operations of the unit normal vector  $\vec{n}$  and the unit vector in the global Y direction,  $\vec{j}$ .

$$\vec{v}_x = \vec{j} \times \vec{n}$$

$$\vec{v}_y = \vec{n} \times \vec{v}_x$$

where  $\vec{v}_x$  and  $\vec{v}_y$  represent unit vectors in the local x and y directions respectively.

- **node:** If the node belongs to only one curve or only one surface mesh, the normal or the axial direction is determined by the curve and the surface. But, if the node shared by multiple curves or by multiple surfaces, then the program asks to choose the curve or the surface mesh to be used as the basis for determining the local directions.



This item is dimmed in the case of a frame, or a planar problem.

First, choose one of the two radio buttons, "Set the tangent curve", or "Set the tangent surface mesh. (For a frame or a planar problem, the second button is dimmed, and the first button is turned on.) And select the curve or the surface mesh for the basis of the local directions. Then, **Select** button becomes active. Click the button to assign the boundary condition in local coordinates.



## Assigning structural boundary conditions

The structural boundary conditions are assigned to objects as a set. The currently active set is assigned to selected objects by clicking **Assign** button of the dialog. It is assumed that all the nodes are initially free. That is, none of the nodal d.o.f. are constrained fully or partially at the beginning. Assigning a boundary condition set is equivalent to modifying the unconstrained state of each nodal d.o.f. Thus, the boundary condition sets are assigned only to the nodes, one or more d.o.f. of which are fixed or free with spring constant. It is meaningless to define or assign a boundary condition set with all d.o.f. free. If you define such a set and assign it to the selected objects, the assignment is ignored.

### ■ Selecting objects to assign structural boundary conditions

Structural boundary conditions may be assigned to nodes, curves and surface meshes. However, the data sets are eventually assigned to nodes within the objects. For example, assigning a boundary condition set to a surface mesh is equivalent to assigning the set to all nodes within the surface mesh.

For a 3 dimensional solid, boundary conditions are usually assigned only to the outer surfaces, or to nodes on these surfaces. Therefore, it is sometime necessary to prevent assigning boundary conditions to nodes inside the volume inadvertently. This can be done by checking “Shut Invisible Nodes” item in **View** menu so that the inside nodes become unselectable.

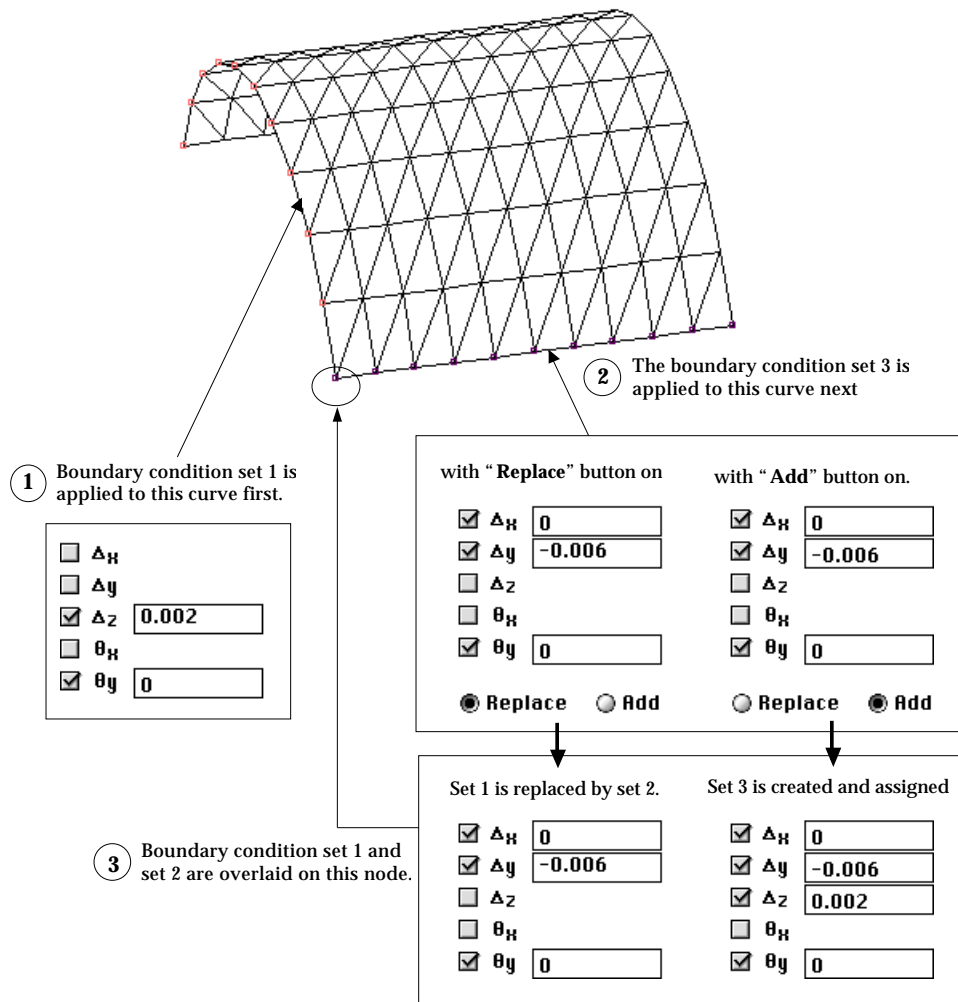
### ■ Replacing or adding previous assignment

When a new structural boundary condition set is assigned to a node with previously assigned boundary conditions, the old set will be either replaced by or added to the new one, depending on the mode of overlaying data sets.

In “Struct Boundary” dialog, there are two radio buttons, “Replace” and “Add”. If “Replace” button is on, the old assignment is replaced by the new assignment. The text data items are also replaced by the corresponding values of the new set.

If “Add” button is on, the resulting assignment is the union of the old and the new assignments. The state of a d.o.f. is determined by bit ORing with ‘0’ for free d.o.f. and ‘1’ for fixed d.o.f. If the editable text data of both old and new sets are of same kind( both prescribed displacements or both spring constants), the item is filled with the one whose absolute value is larger.

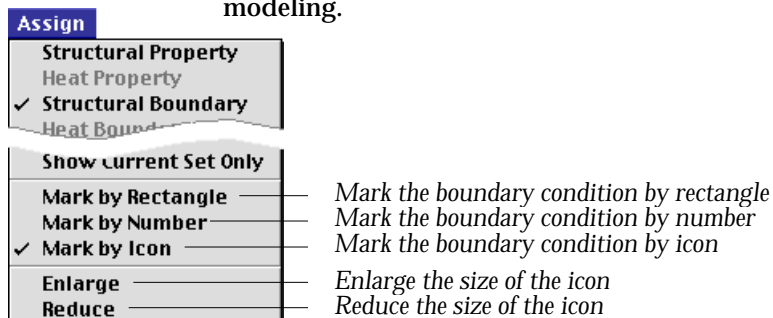
If there exists a boundary condition set identical to the one resulting from the bit ORing, that set is assigned to the node with the overlaid boundary conditions. Otherwise, a new set is created.

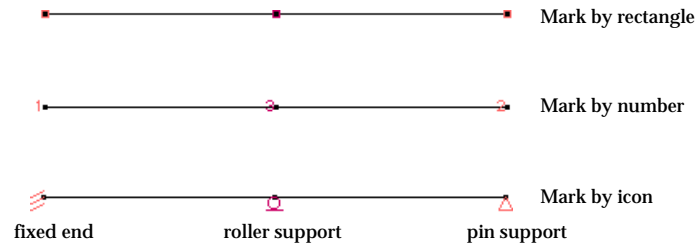


< Example of overlaying boundary conditions >

### ■ Representation of boundary condition assignment

Nodal assignment of structural boundary conditions is represented by icon, number or rectangle marked at the node. Choose the desired option of marking the boundary conditions from **Assign** menu. The item of the currently applied option is checked as shown below. The icon option is available only for 2-D modeling.





<Options of representing boundary conditions>

If a boundary condition set is assigned to curves or surfaces, the individual nodes on the curves or surfaces are marked. The nodes assigned with the currently active data set are highlighted in dark red, and the nodes with other sets are marked in bright red. The boundary conditions are displayed only when "Struct Boundary" dialog is on the screen. Check "Show Str. Boundary" item of **View** menu to make the boundary conditions displayed even when "Struct Boundary" dialog is not on. The size of the icon representing the boundary conditions can be enlarged or reduced by using "Enlarge" or "Reduce" item of **Assign** menu.

## Heat Conduction and Seepage Boundary conditions

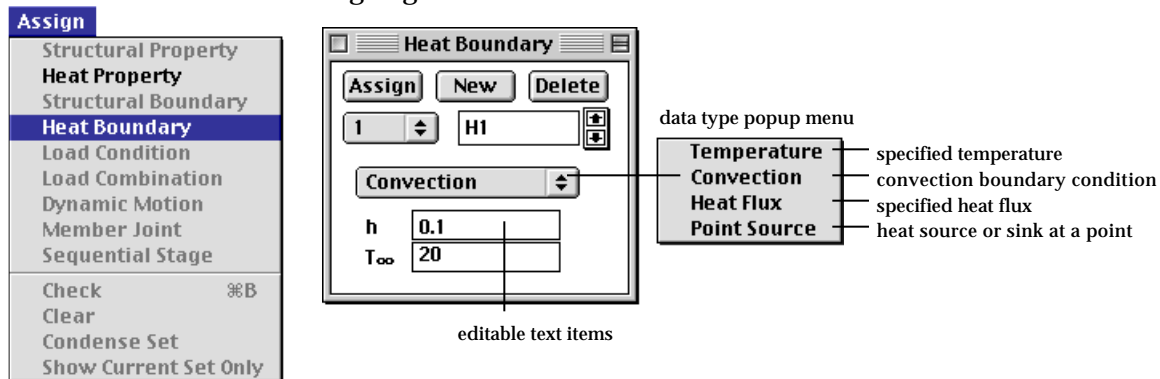
A menu item of **Assign** menu is displayed in one of the two texts "Heat Boundary" and "Seepage Boundary", depending on whether the analysis subject of the project is heat conduction analysis or seepage analysis.

The boundary conditions in heat conduction and seepage analysis can be defined and assigned by choosing the menu item.

Heat conduction boundary conditions are relevant only for heat conduction analysis. Accordingly, "Heat Boundary" item in the menu is enabled only when the analysis subject is set as one of the "Heat Analysis" item in "Problem Setup" dialog. Likewise, seepage boundary conditions are relevant only for seepage analysis. The text of the menu item displayed as "Seepage Boundary" when the analysis subject is set as one of the "Seepage Analysis" item.

### Defining heat conduction boundary conditions

You must first create and define sets of heat conduction boundary conditions before assigning them on the heat conduction model.



### ■ Types of heat conduction boundary condition

There are a few different types of heat conduction boundary conditions as follows:









- temperature: A temperature is specified for various objects such as nodes, curves and surfaces.
- convection: The convection coefficient and the ambient temperature are assigned to an edge curve or a boundary surface.
- heat flux: The heat flux through a curve or a face is specified.
- point source: A node is defined as a heat source. Heat source is more analogous to load condition of structural analysis, and may be treated as such. In VisualFEA, however, heat source is treated as a kind of boundary condition.

The type of heat conduction boundary condition is chosen using the popup menu in “Heat Boundary” dialog. The editable text items are altered subsequent to selection of the popup menu.

### ■ Editable text items

Each type of heat conduction boundary conditions has different editable text items as shown in the following table. The boundary conditions are defined by entering the numerical values into these editable text items. Centigrade, Fahrenheit, or any other unit may be employed, but should be used consistently for all items.

< Editable text items of heat boundary condition >

Editable Text Items	Assignable Object Types
<p><b>Temperature</b> ▾</p> <p>T <input type="text" value="28"/> — Temperature in degree centigrade, Fahrenheit, or any other unit</p>	<p> Temperature distribution can be specified at nodes, along curves (of 2-D plane and axisymmetric), or on surface meshes (of 3-D volume).</p> <p> </p>
<p><b>Convection</b> ▾</p> <p>h <input type="text" value="0.5"/> — Convection coefficient</p> <p>T<sub>∞</sub> <input type="text" value="12.9"/> — Ambient temperature</p>	<p> Convection boundary condition can be assigned to curves ( of 2-D plane or axisymmetric) or to surface meshes ( of 3-D volume).</p> <p></p>
<p><b>Heat Flux</b> ▾</p> <p>q <input type="text" value="10"/> — Heat flux per unit length or area</p>	<p> Heat flux can be assigned to curves ( of 2-D plane or axisymmetric) or to surface meshes ( of 3-D volume).</p> <p></p>
<p><b>Point Source</b> ▾</p> <p>q <input type="text" value="5.72"/> — Amount of heat generation</p>	<p> Point source can be assigned only to nodes</p>

## Assigning heat conduction boundary conditions

After defining a heat conduction boundary condition as a set, you can assign the set to the selected objects by clicking **Assign** button of the dialog. You can scroll the heat conduction boundary condition sets, and make any of them the currently active set, which is always applied for assignment.

### ■ Selecting objects to assign heat conduction boundary conditions

The assignable object types are determined by the type of boundary conditions, as shown in the previous table. Whenever the popup menu item in the dialog is altered, some selection tools are enabled and others disabled accordingly. You can switch to one of the enabled selection tools, and assign the currently active boundary condition to the objects of the relevant type.

### ■ Replacing previous assignment

An object can be assigned with only one heat boundary condition set. If you assign a new set to the object which has already been assigned with other set, the old set will be replaced by the new one.

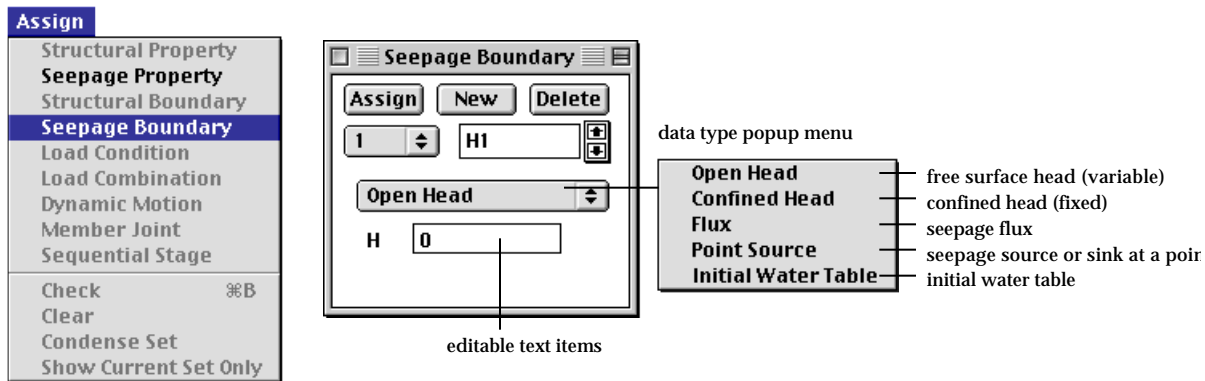
But, it should be noted that one object may encompass other objects of different type, and accordingly multiple sets may actually be assigned to a single object. For example, a temperature may be specified along a curve, while the other temperature is assigned to a node which is on the same curve. Thus, the node is assigned with two condition sets which are in conflict. In this case, the nodal assignment overrides other assignments such that the temperature of the curve is not applied to that node. In general, if one object encompasses the other object, and both objects are assigned with two different heat boundary condition sets, the assignment on the encompassed object always overrides the other assignment.

### ■ Representation of heat conduction boundary condition assignment

The objects assigned with heat conduction boundary conditions are distinguished from others by small square marks or by color. The nodes assigned with the current set are marked by dark red squares. The nodes assigned with other than the current set are marked by bright red squares. The curves, surface meshes, or volume meshes assigned with the current set are drawn in dark red color, and those assigned with other than the current set are drawn in bright red color.

## Seepage boundary conditions

If the analysis subject is set as either "2-D Seepage", "Axisymmetric Seepage", or "3-D Seepage", the text of the menu item "Heat Boundary" changes to "Seepage Boundary". Selection of the menu item pops up the "Seepage Boundary" dialog. Seepage boundary conditions can be defined and assigned, while the dialog is on the screen,



### ■ Types of seepage boundary condition

There are a few different types of seepage boundary conditions as described below:

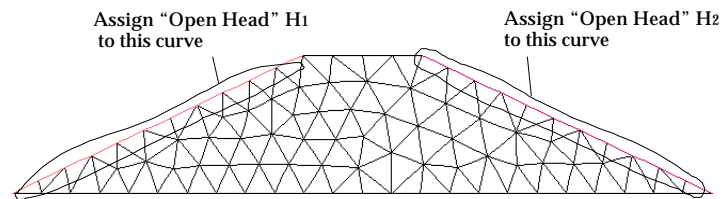
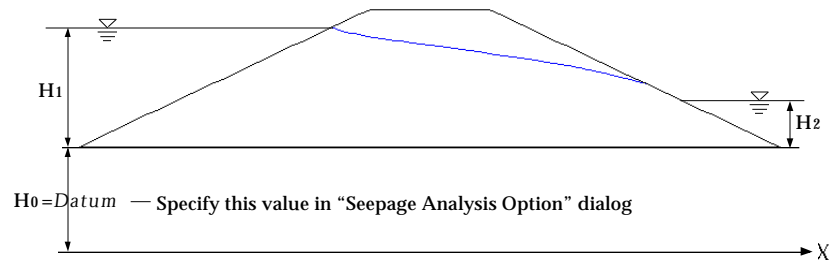
- **open head:** Open head is a type of boundary condition assigned to the domain boundary curves or surfaces such as a upstream or a downstream face of a dam. The boundaries assigned with open head include nodes under and above the specified water head. The nodes under the specified head are prescribed with the water head in the system equations. The nodes above the specified head are so called "review nodes" subject to unknown phreatic water face which are determined through iterative solution process. The open head allocated review nodes automatically to the nodes above water level.
- **confined head:** Confined head is a type of boundary condition prescribing all the assigned nodes with the specified water head .
- **flux:** seepage flow rate through unit area or length.
- **point source:** flow supplied at a point.
- **initial water table:** water table at the initial stage of transient seepage flow, or water table initially assumed in the iterative solution process.

### ■ Assigning open head boundary condition

Open head boundary conditions can be assigned using "Seepage Boundary" dialog by the following procedures.

- 1) Set the popup menu (Windows: dropdown list) to "Open Head".

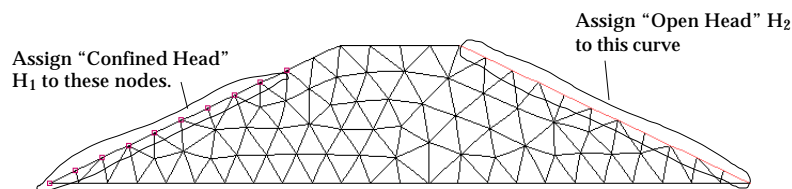
- 2) Insert the value of the water head in the editable text box with label "H".  
The upstream water level  $H_1$  and the downstream water level  $H_2$  are the values of the open water heads in the example below.  
*The water levels are measured from the datum to be specified in "Analysis Option" dialog. Refer to "Setting analysis options for seepage analysis" section of Chapter 6.*
- 3) Select the boundary curves or surfaces to be assigned with the open head..  
Select curves for 2-D analysis and surfaces for 3-D analysis.
- 4) Assign the boundary condition to the selected objects by clicking **Assign** button of the dialog.



<Example of open head boundary conditions>

### ■ Assigning confined head boundary condition

As an alternative to the open head boundary condition in the above example, the nodes of the upstream edge under the water level can be assigned with "Confined Head"  $H_1$ , because the nodes of the upstream edge cannot form a phreatic water face above the upstream water level. However, it should be noted that the confined head should be assigned only to the nodes under the upstream water level.

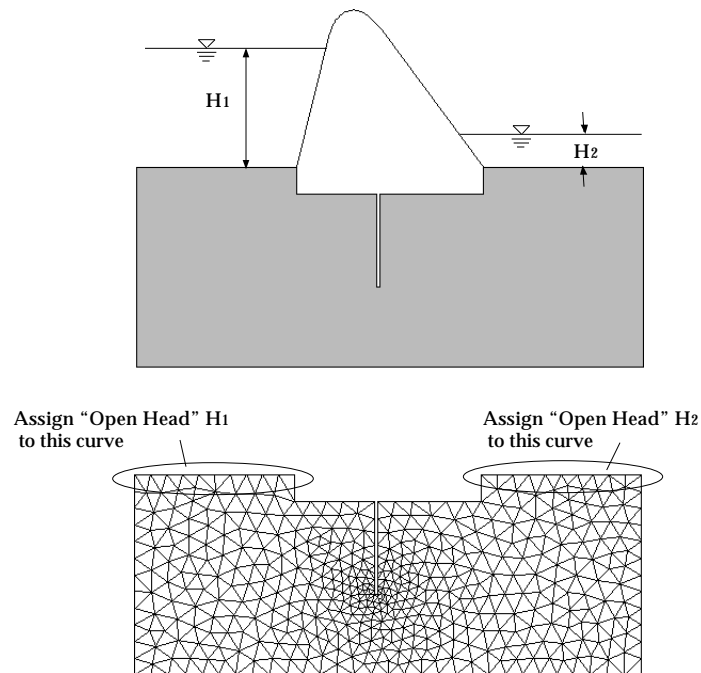


<Example of a confined head boundary condition in unconfined seepage analysis>



The above example is to show that a confined boundary condition may substitute an open head boundary condition in some cases. But confined boundary conditions are commonly used to prescribe the constant head at the selected nodes or nodes along the selected curve, as shown in the example below. In the case of confined seepage analysis, only confined head conditions are applied.

In order to assign confined head boundary conditions, set the popup menu(Windows: dropdown list) to "Confined Head," and follow the similar procedure as that of open head boundary condition.



<Example of confined head conditions in confined seepage analysis>

### ■ Assigning flux

Water supply or drain through the boundary surfaces is represented by the boundary condition of flux. In order to assign fluxes, set the popup menu(Windows: dropdown list) to "Flux". The value of the flux should represent volume of water per unit area per unit time. The flux should be assigned to boundary curves in 2-D seepage analysis and to boundary surfaces in 3-D seepage analysis.

### ■ Assigning point source

Water supply or drain at a point is represented by the boundary condition of point source. In order to assign point sources, set the popup menu(Windows: dropdown list) to "Point Source". The value of the point source should represent volume of water per unit time. The point source should be assigned to nodes.

### ■ Defining the initial water table

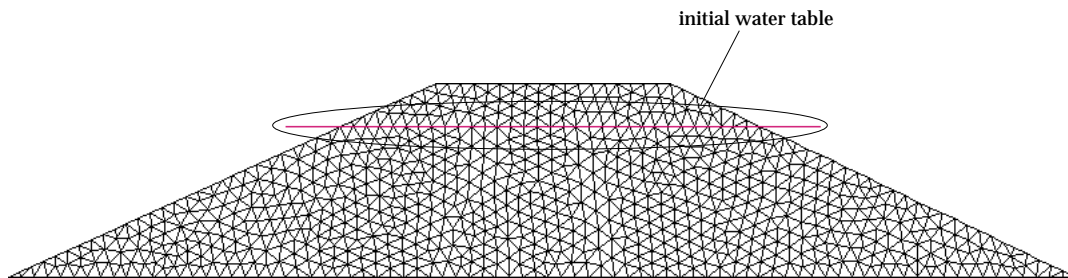
In transient analysis, the initial water table should be defined as the phreatic surface at the initial stage. But in steady state analysis, the initial water table can be considered as the initially assumed phreatic water surface, and therefore, defining the initial water table may not be essential. However, the assumption of the initial water table affects the computational efficiency of the iterative solution process, and is essentially used in obtaining the overall distribution of the negative pore water pressure above phreatic surface. The negative pore water pressure above the initial water table is set to 0.

The initial water table can be defined by the following procedure.

- 1) Create a curve or a surface to be defined as the initial water table.

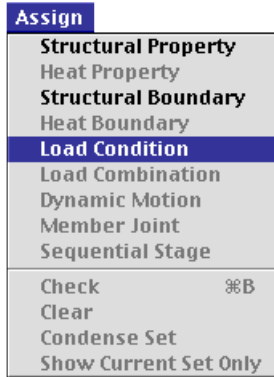
The initial water table is defined by one or more curves in 2-D seepage analysis, and one or more surface meshes in 3-D seepage analysis. The curves or surface meshes can be created as usual. If there already exist curves or surface meshes that can be defined as the initial water table, this step can be ignored.

- 2) Open "Seepage Boundary" dialog if it is not yet opened.
- 3) Set the popup menu (Windows: dropdown list) to "Initial Water Table".
- 4) Select the curves or the surface meshes to be defined as the initial water table.
- 5) Click **Assign** button of the dialog.



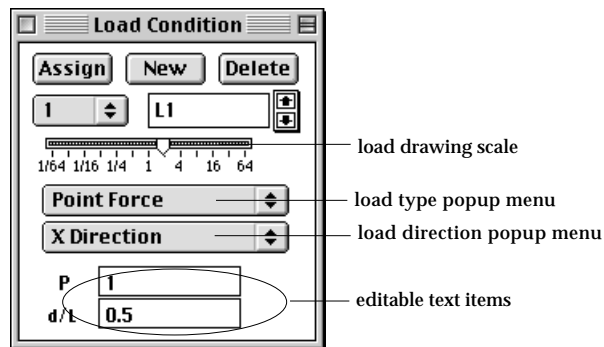
<Example of initial water table>

## Load Conditions



The load condition represents the state of various forces applied to the objects in structural analysis.

In order to start assigning load conditions, choose “Load Condition” item in **Assign** menu. “Load Condition” dialog appears, and the current state of their assignment is displayed in the main window.



### Defining load condition sets

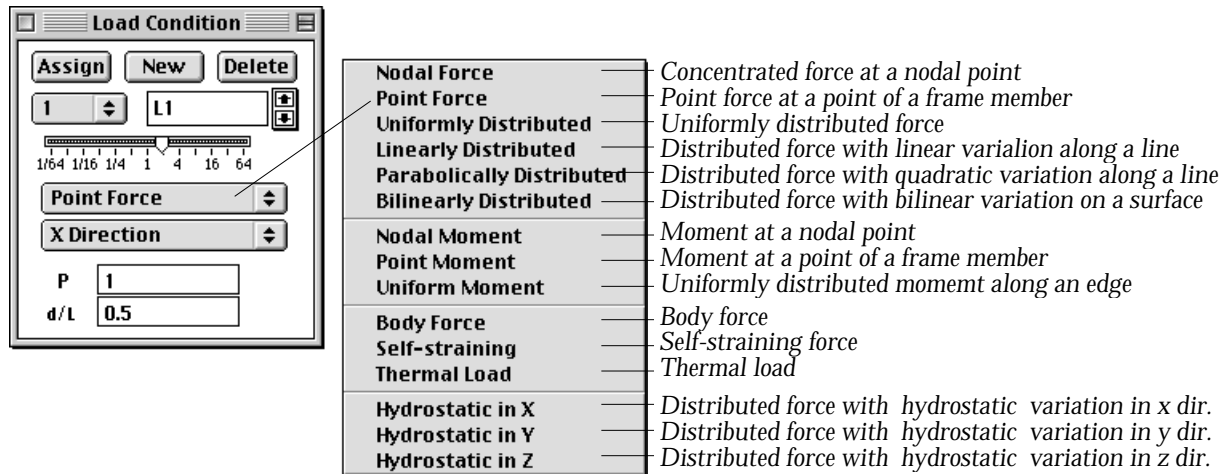
Load conditions are defined and assigned by the data unit called a load condition set. A load condition set is initially created by clicking **NEW** button of “Load Condition” dialog. If a load set is already assigned, additional assignment of the set, either to a single object or multiple objects, creates a new load set automatically by duplicating the set. Loads applied to the assigned object(s) share the new load set.

#### ■ Data items of a load condition









A set consists of many data items including the type, the direction, the magnitude, and the position of a force. The actual data items vary depending on the type of analysis.

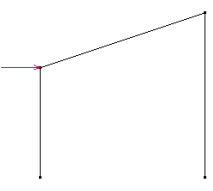
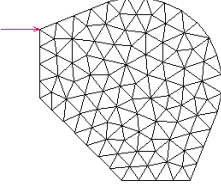
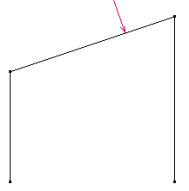
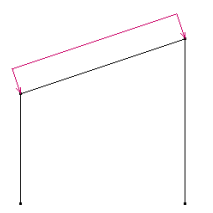
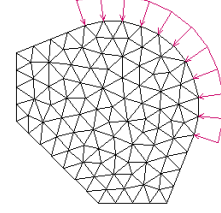
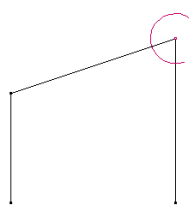
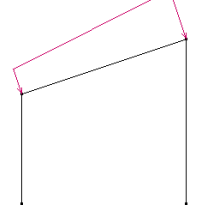
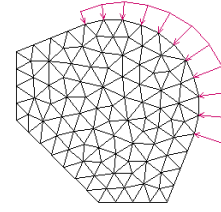
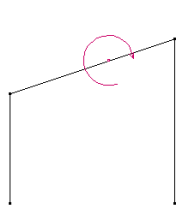
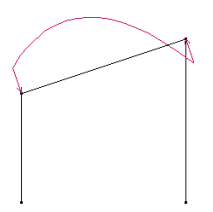
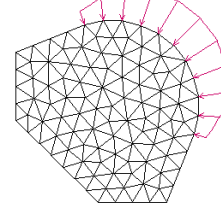
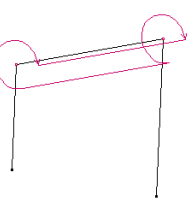
- load type: A number of load types are included in the first popup menu of “Load” dialog.
- load direction: The local directions such as normal and tangential, as well as x, y and z directions in the global coordinates are provided in the second popup menu of the dialog.
- editable text items: Magnitude, position, reference points and others are inputted in the editable text items in the dialog. The editable text items differ depending on the type of the objects.

#### ■ Load types




There are a number of different load types as shown in the the popup menu items. Some of the menu items are enabled, while others are disabled depending on the type of analysis. For example, “Point Force” item is available only for 2-D or 3-D frames.

- nodal force: Nodal forces are applied at nodes. When this item is chosen in the popup menu, the node selection tool  is automatically activated, so that you may select nodes to which the force is applied.
- point force: Point forces are applied at a point between nodes on a frame member. This item is enabled only for 2-D and 3-D frame analysis. When this item is chosen in the popup menu, the element selection tool  is automatically activated.
- uniformly distributed: These are forces distributed uniformly over a curve or a surface. In order to apply uniform forces, you must first activate the curve selection tool , or the surface selection tool , unless one of the two tools is already in action.
- linearly distributed: These are distributed forces with linear variation along a curve. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- parabolically distributed: These are distributed forces with quadratic variation along a curve. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- bilinearly distributed: These distributed forces have bilinear variation on a surface. The force intensity at a point is determined by bilinear interpolation of the intensities at 4 corner points. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- nodal moment: This represents a moment or a couple applied at a node. When this item is chosen in the popup menu, the node selection tool  is automatically activated.

Load Type	2-D Frame and 2-D Elasticity Example		Load Type	2-D Frame Example
Nodal Force			Point Force	
Uniform Force			Nodal Moment	
Trapeziform Force			Point Moment	
Parabolic Force			Uniform Moment	

&lt; Examples of load types &gt;

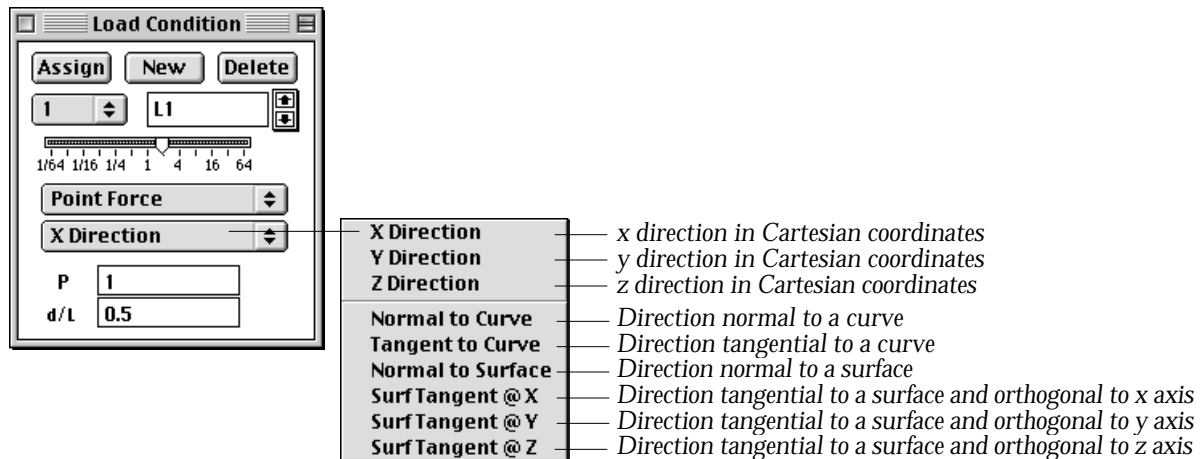
- **point moment:** This represents a moment or a couple applied at a point between nodes on a frame member. This item is enabled only for 2-D and 3-D frame analysis. When this item is chosen in the popup menu, the element selection tool  is automatically activated.
- **uniform moment:** This represents a line moment applied uniformly on a frame member or an edge of a surface.
- **body force:** This is the force due to acceleration of mass. One example of body force is the weight, or gravitational force of an object. Body force is applicable to all types of objects except slip bar and embedded bar elements.
- **self-straining force:** This is internally exerted force due to prestressing, lack of fit, and so on. Self-straining force can be applied to frame members or embedded bar elements.
- **thermal load:** Thermal load implies the temperature variation or temperature gradient which causes strains and stresses. The equivalent

nodal forces are computed from the give temperature distribution. In a coupled analysis of structure and heat conduction, the temperature distribution is not specified by the thermal load but is given by the nodal values computed from heat conduction analysis.

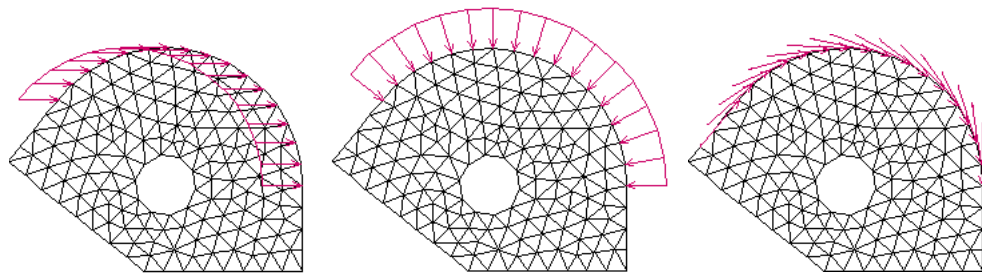
- hydrostatic in X: “Hydrostatic in X” implies linear variation in the X direction. This is to represent forces distributed on an area with intensity varying linearly in X direction and uniform in other directions.
- hydrostatic in Y: “Hydrostatic in Y” implies linear variation in the Y direction. This is to represent forces distributed on an area with intensity varying linearly in Y direction and uniform in other directions.
- hydrostatic in Z: “Hydrostatic in Z” implies linear variation in the Z direction. This is to represent forces distributed on an area with intensity varying linearly in Z direction and uniform in other directions.

### ■ Load direction

The direction of forces may be specified either in X, Y, and Z axis of Cartesian coordinates system or in local directions normal or tangential to curves or surfaces.



- X direction: X direction in XYZ coordinate system.
- Y direction: Y direction in XYZ coordinate system.
- Z direction: Z direction in XYZ coordinate system.
- normal to curve: Direction normal to the curve on which the force is acting.
- tangent to curve: Direction tangential to the curve on which the force is acting.



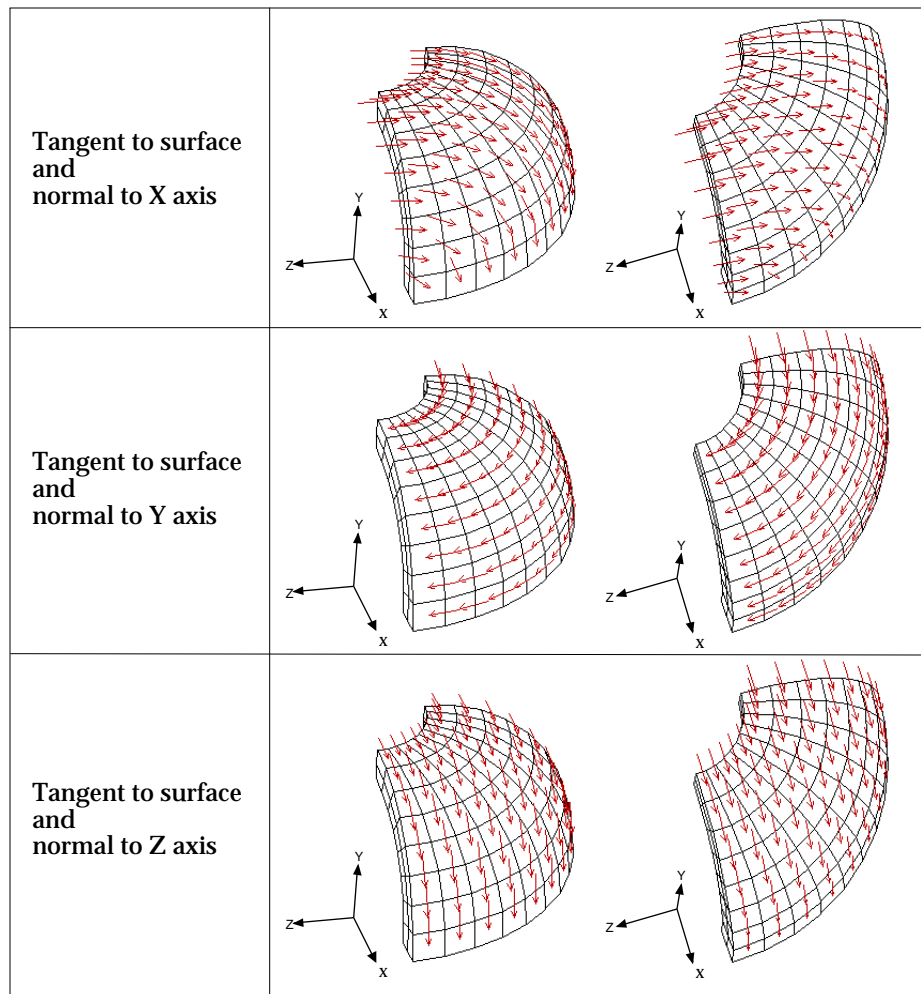
X Direction

Normal to Curve

Tangent to Curve

< Direction of a uniform load applied to a curve >

- **normal to surface:** Direction normal to the surface on which the force is acting.
- **tangent to surface and normal to X:** The direction tangential to the surface on which the force is acting, and normal to X direction. The direction tangential to a surface is not unique. The direction is constrained to be normal to X direction, so that it became unique.
- **tangent to surface and normal to Y:** The direction tangential to the surface on which the force is acting, and normal to Y direction.
- **tangent to surface and normal to Z:** The direction tangential to the surface on which the force is acting, and normal to Y direction.
- **no direction:** This item is provided only for a thermal load which does not have a direction. The load direction popup item automatically turns to this item when the current load type is set as thermal load.




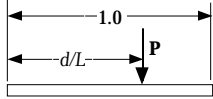
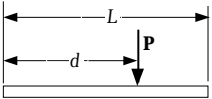





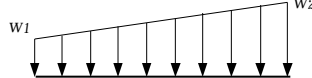

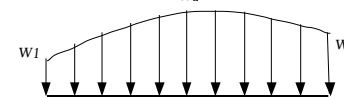

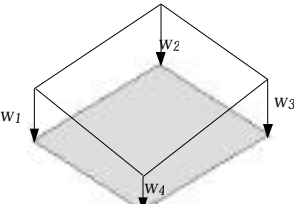

< Directions tangent to surface >

### ■ Editable text items



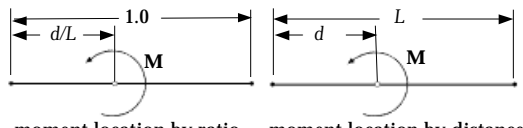


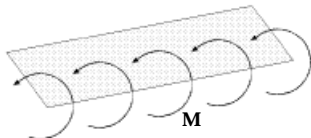






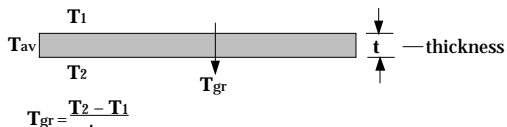


The editable text items are different depending on the type of loads as shown in the following figures. When the popup menu item of load type is selected, corresponding editable text items appear on the dialog. The numerical values related to a force are specified by these editable text items.

< Editable text items of load conditions >



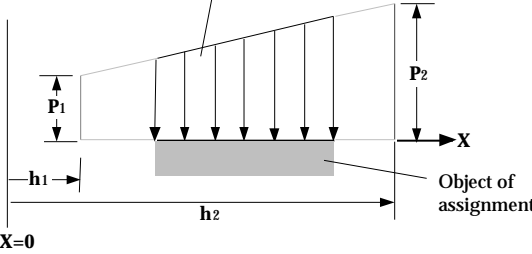





Editable Text Items	Assignable Object Types
<p><b>Nodal Force</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>P <input type="text" value="1"/> force magnitude</p>	<p> Nodal forces can be assigned only to nodal points. If "Nodal Force" is chosen as the type of load to assign, the node selection tool is automatically activated.</p>
<p><b>Point Force</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>P <input type="text" value="4.0"/> force magnitude</p> <p>d/L <input type="text" value="0.7"/> force location by ratio</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>force location by ratio</p> </div> <div style="text-align: center;">  <p>force location by distance</p> </div> </div> <p>P <input type="text" value="4"/></p> <p>d <input type="text" value="3.5"/> force location by distance</p> <p>Click here to toggle d/L and d. <a href="#">↔</a></p>	<p> Point forces can be assigned only to a frame member. Therefore, this load type option is enabled only for 2-D or 3-D frames. Point forces can be applied either to elements (frame member) or to curves.</p> <p>The force location can be specified either by ratio(d/L) or by distance(d).</p> <p>  This type of force cannot be applied to solid elements.</p>
<p><b>Uniformly Distributed</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>w <input type="text" value="1"/> force intensity per unit length or per unit area</p>	<p> Uniformly distributed forces can be assigned to elements (except solid elements), to curves or to surface meshes.</p> <p> This type of force cannot be applied to solid elements.</p>
<p><b>Linearly Distributed</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>w<sub>1</sub> <input type="text" value="0.8"/> force intensity at one end</p> <p>w<sub>2</sub> <input type="text" value="2.3"/> force intensity at the other end</p> 	<p> Linearly varying trapesiform forces can be applied only to curves. In order to assign linear varying forces to surface meshes, use "Hydrostatic in X", "Hydrostatic in Y" or "Hydrostatic in Z"</p>
<p><b>Parabolically Distributed</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>w<sub>1</sub> <input type="text" value="1.0"/> force intensity at one end</p> <p>w<sub>2</sub> <input type="text" value="2.5"/> force intensity at mid point</p> <p>w<sub>3</sub> <input type="text" value="1.6"/> force intensity at the other end</p> 	<p> Quadratically varying parabolic forces can be applied only to curves or frame elements.</p>
<p><b>Bilinearly Distributed</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>w<sub>1</sub> <input type="text" value="2"/> <input checked="" type="checkbox"/> w<sub>1</sub> at corner 1</p> <p>w<sub>2</sub> <input type="text" value="3"/> <input checked="" type="checkbox"/> w<sub>2</sub> at corner 2</p> <p>w<sub>3</sub> <input type="text" value="2"/> <input checked="" type="checkbox"/> w<sub>3</sub> at corner 3</p> <p>w<sub>4</sub> <input type="text" value="1"/> <input checked="" type="checkbox"/> w<sub>4</sub> at corner 4</p> 	<p> Bilinearly varying distributed forces can be applied only to surface meshes.</p>

&lt; Editable text items of load conditions (continued)&gt;

Editable Text Items	Assignable Object Types
<p><b>Nodal Moment</b> ▾</p> <p><b>Z Axis</b> ▾</p> <p>M <input type="text" value="1.3"/> — magnitude of moment</p> 	<p> Nodal moment can be applied to nodes of frame members. Therefore this popup menu item is enabled only for frame analysis.</p>
<p><b>Point Moment</b> ▾</p> <p><b>Z Axis</b> ▾</p> <p>M <input type="text" value="3.21"/> — magnitude of moment</p> <p>d/L <input type="text" value="0.7"/> — moment location by ratio</p>  <p>moment location by ratio      moment location by distance</p> <p>M <input type="text" value="3.21"/></p> <p>d <input type="text" value="2.8"/> — force location by distance</p> <p>Click here to toggle d/L and d.</p>	<p> Point moments can be assigned only to a frame member. Therefore, this load type option is enabled only for 2-D or 3-D frames. Point moments can be applied either to elements (frame member) or to curves. The moment location can be specified either by ratio(d/L) or by distance(d).</p> <p></p>
<p><b>Uniform Moment</b> ▾</p> <p><b>Z Axis</b> ▾</p> <p>M <input type="text" value="3.21"/> — moment intensity per unit length</p> 	<p> Uniformly distributed moments can be applied to frame elements or to edges of a shell or a plate. Therefore, this type of load is valid only for 2-D frame, 3-D frame, plate bending and shell models.</p> <p></p>
<p><b>Body Force</b> ▾</p> <p><b>X Direction</b> ▾</p> <p>a <input type="text" value="1"/> — acceleration</p> <p>The magnitude of body force is determined by the acceleration and the unit mass which is specified as an item of the element property</p> <p>body force <math>F_b = W_u a</math> — acceleration</p> <p>unit mass</p>	<p> Body forces can be assigned to elements, curves(for frames only), surface meshes(for planar elasticity, plate and shell), and volume meshes(for 3-D solid)</p> <p></p> <p></p> <p></p>
<p><b>Thermal Load</b> ▾</p> <p><b>Tangent to Curve</b> ▾</p> <p>T<sub>av</sub> <input type="text" value="24.5"/> — average temperature</p> <p>T<sub>gr</sub> <input type="text" value="2.3"/> — temperature gradient</p>  <p>T<sub>av</sub> <math>T_{gr} = \frac{T_2 - T_1}{t}</math> — thickness</p>	<p> Thermal loads can be applied to frame elements and curves. This type of load is valid only for 2-D and 3-D frame.</p> <p></p>

&lt; Editable text items of load conditions (continued)&gt;

Editable Text Items	Assignable Object Types
<p>Self-straining</p> <p>Tangent to Curve</p> <p>P <input type="text" value="1"/> — magnitude of straining force</p> <p style="text-align: center;">P ←————→ P</p>	<p> Self-straining forces can be applied to elements and curves, which should be members of truss or frame structures, or embedded bars within solids.</p> <p></p>
<p>Hydrostatic in X</p> <p>Normal to Curve</p> <p>P<sub>1</sub> <input type="text" value="1.2"/> — force intensity w at X=h<sub>1</sub></p> <p>P<sub>2</sub> <input type="text" value="2.5"/> — force intensity w at X=h<sub>2</sub></p> <p>h<sub>1</sub> <input type="text" value="-2"/> — X coordinate at which w=P<sub>1</sub></p> <p>h<sub>2</sub> <input type="text" value="4.1"/> — X coordinate at which w=P<sub>2</sub></p> <p>Hydrostatic in Y</p> <p>X Direction</p> <p>P<sub>1</sub> <input type="text" value="-3.8"/> — force intensity w at Y=h<sub>1</sub></p> <p>P<sub>2</sub> <input type="text" value="3.4"/> — force intensity w at Y=h<sub>2</sub></p> <p>h<sub>1</sub> <input type="text" value="2.1"/> — Y coordinate at which w=P<sub>1</sub></p> <p>h<sub>2</sub> <input type="text" value="9.84"/> — Y coordinate at which w=P<sub>2</sub></p> <p>Hydrostatic in Z</p> <p>SurfTangent @ X</p> <p>P<sub>1</sub> <input type="text" value="5.3"/> — force intensity w at Z=h<sub>1</sub></p> <p>P<sub>2</sub> <input type="text" value="-1.5"/> — force intensity w at Z=h<sub>2</sub></p> <p>h<sub>1</sub> <input type="text" value="1.234"/> — Z coordinate at which w=P<sub>1</sub></p> <p>h<sub>2</sub> <input type="text" value="0"/> — Z coordinate at which w=P<sub>2</sub></p> <p style="text-align: center;">“Hydrostatic in X”</p>  <p style="text-align: center;">Object of assignment</p>	<p> “Hydrostatic in X”, “Hydrostatic in Y”, and “Hydrostatic in Z” can be applied to elements (surface elements only), curves and surface meshes.</p> <p></p> <p></p>

## Defining load condition sets for dynamic analysis

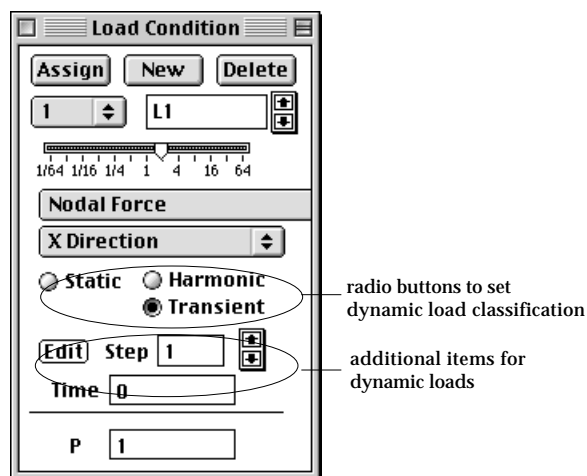
For dynamic analysis, load conditions may or may not have time dependent characteristics. Therefore, assignment of dynamic load conditions involves more steps than that of static load conditions, as described below.

### ■ Setting the time dependency

The dynamic load condition is classified further its time dependency as follows.

- static: static load condition independent of time.
- harmonic: time-dependent load condition with periodically changing magnitude expressed by a sinusoidal equation.
- transient: time-dependent load condition represented in multiple time steps.

Thus, "Load Condition" dialog for dynamic analysis has radio buttons, "Static", "Harmonic" and "Transient." Time dependency of a dynamic load is set by turning on the corresponding radio button on "Load Condition" dialog.



As you alter the time dependency, you will notice the changing items below the radio buttons. Time dependent attributes of dynamic loads are defined and modified using these items.

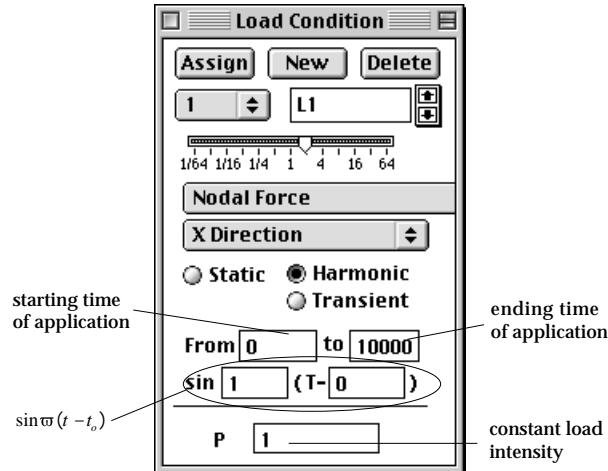
### ■ Defining static load for dynamic analysis

Static load condition is assumed to have constant magnitude, direction and position throughout the whole duration of analysis, and can be defined and assigned in the same way as for static analysis. "Static" radio button should be turned on to start defining static loads.

### ■ Defining harmonic load for dynamic analysis

A harmonic load has periodic characteristics with changing magnitude expressed by the sinusoidal function  $\sin \omega(t - t_o)$ . The intensity of the load acting at a

specific time is obtained by multiplying the function by the constant load intensity given at the bottom of the dialog.

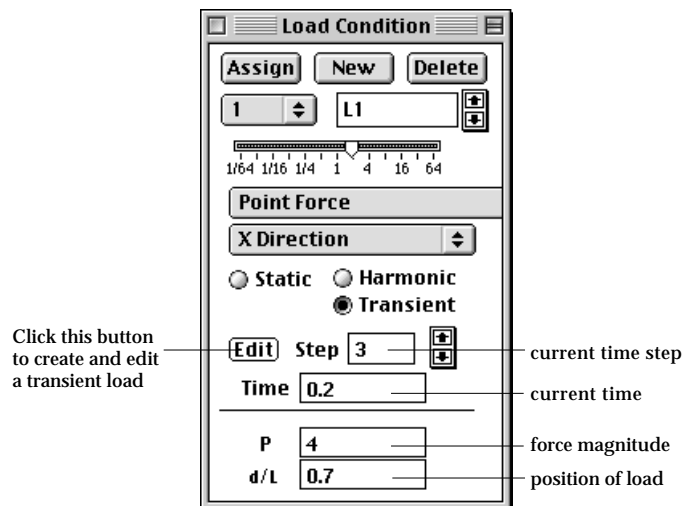


### ■ Defining transient load for dynamic analysis

A transient load is defined by a series of values at a number of time steps, which can be created and edited by the following steps.

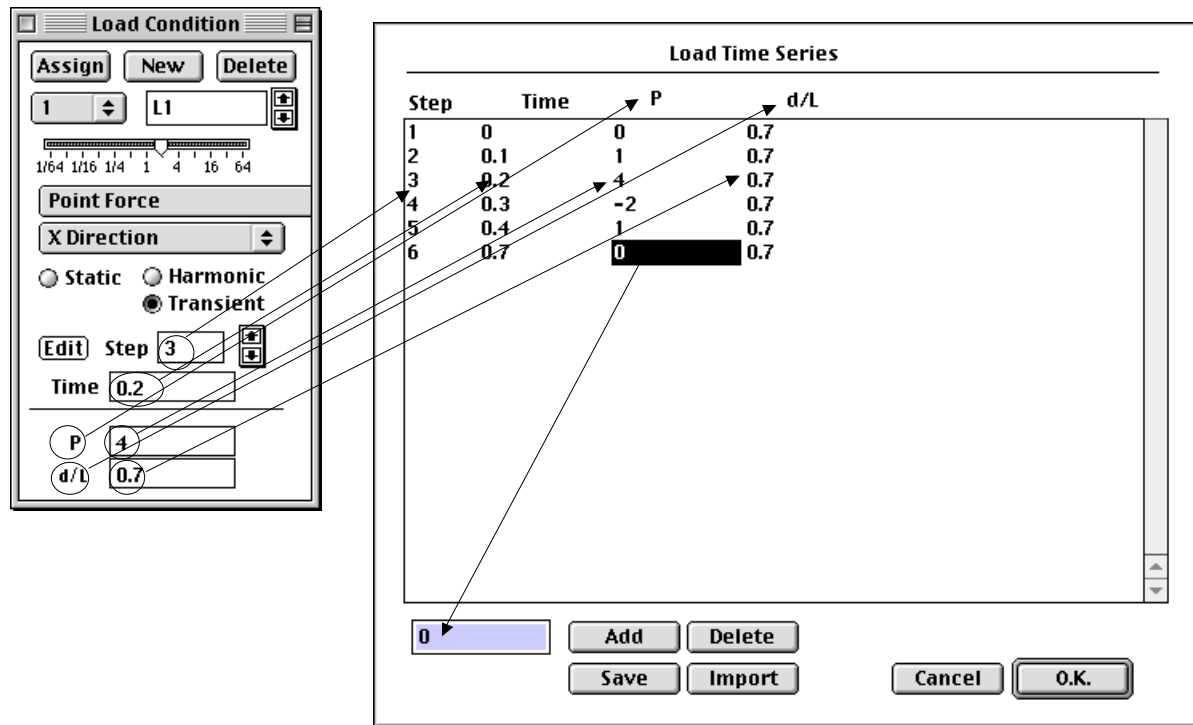
- 1) Turn on "Transient" radio button in "Load Condition" dialog.

Then, items below the radio buttons take on the appearance shown below. They include the current step number and the current time. The current time step can be edited by directly changing the number in the box, or by scrolling the number.



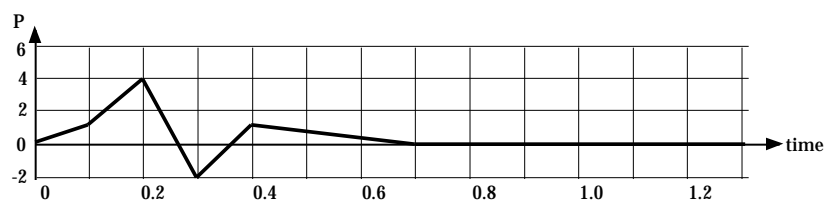
- 2) Click **Edit** button.

Then, "Load Time Series" dialog pops up. The dialog produces editable data items for each time step you create. The data items are equivalent to the data items in "Load Condition" dialog.



- 3) Click **Add** button to add a time step.  
A line of data items will be created for the new time step. In order to delete a time step, click **Delete** button after selecting the corresponding data line.
- 4) Select a line or a cell to edit.  
Then, the data on the line or in the cell will be reflected in the editable text boxes at the bottom of the dialog
- 5) Edit the text in the editable text box.  
As you edit the text in a text box, it will immediately be echoed in the actual data line.
- 6) Click **O.K.** button after completing creation of time steps.  
"Load Time Series" closes and the data from the dialog are automatically reflected in "Load Condition" dialog.

The data on the last line are assumed to be maintained up to the last time step in the analysis. Thus the data shown in the above example dialog will create the following time series data.



< An example of transient load >

## Assigning load condition sets

A load condition set is defined as explained in the previous section. The next step is to assign the set to a certain object. You can assign the currently active set to selected objects by clicking **Assign** button of the dialog.

### ■ Selecting objects to assign load conditions

The load condition sets may be applied to various objects. However, assignable object types are limited and determined by the type of load conditions, as indicated in the previous figures of “Editable text items”. Only the selection tools of assignable objects are enabled and others are disabled, whenever the type of the current load set is altered.

### ■ Assigning a load condition set to multiple objects

You can apply the current load condition set to more than one object. However, once the set is assigned, extra new load condition sets are automatically generated so that each of the selected objects is assigned with an independent set which has the same characteristics as that of the current set. So, you will find that the number of load condition sets is increased as much as the number of assigned objects minus one.

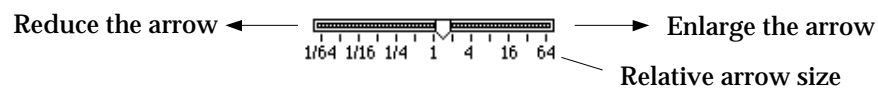
### ■ Assigning multiple load condition sets to a single object

One object may be assigned with more than one load condition sets. But there is a possibility that you assign, by mistake, an identical load conditions to the same object more than once. In order to prevent such a trouble, VisualFEA does not allow assigning the same load condition to one object twice or more.

### ■ Representation of load assignment

The assigned forces, except body forces, are represented by arrows indicating the point and the direction of their application. Body forces are indicated by x mark, instead of arrow, at the centroid of each element.

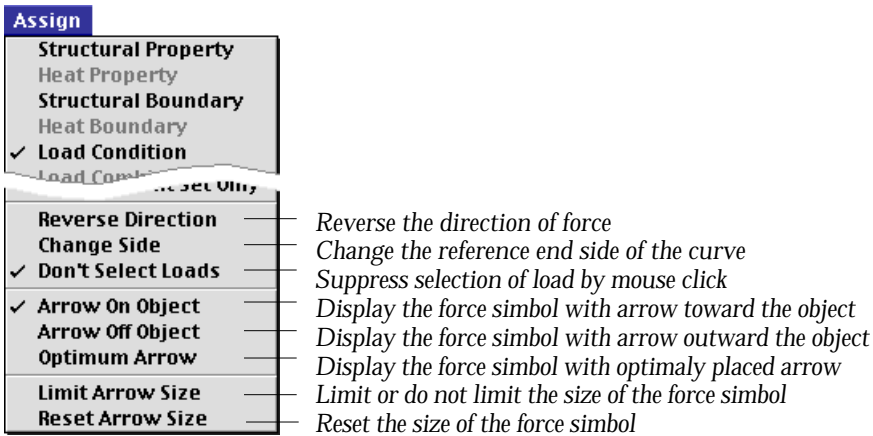
The size of arrow can be adjusted using the slide bar of load drawing scale. The slide bar has relative scale of arrow size. The arrow size can be lengthened by moving the thumb of the bar to the right, and shortened by moving the thumb to the left.



< Load drawing scale >

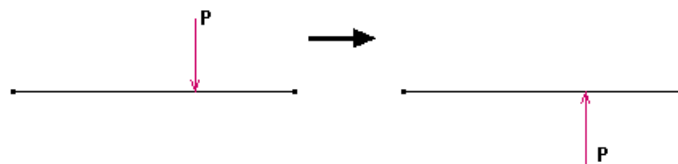
## Other functions related to assigning load conditions

Load conditions can be defined and assigned while the load condition dialog appears on the screen. At this time, the few menu items related with load conditions are attached to **Assign** menu as shown below.



### ■ Reversing the force direction

The direction of the force can be reversed by simply choosing the item “Reverse Direction” from **Assign** menu. The force direction of the current load condition set is reversed, and accordingly, the corresponding editable text items of “Load Condition” dialog is altered with reversed sign. If a load condition set is applied to multiple objects, and “Reverse Direction” command is issued immediately, then the direction of all the lastly assigned loads will be reversed.

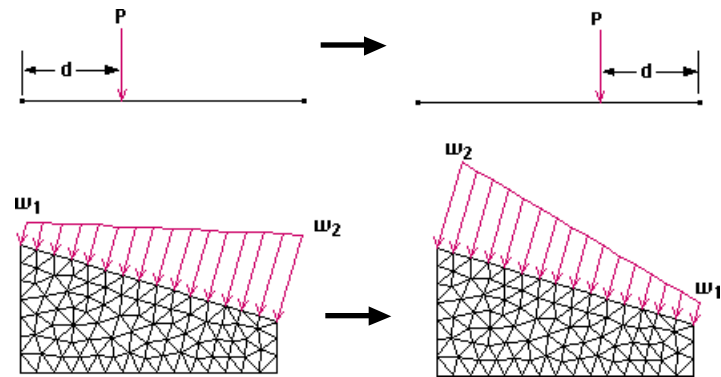


< Result of “Reverse Direction” command >

### ■ Exchanging the reference end of the curve

The location of point force is represented by the distance from one end of a curve or a frame element. In this case, the starting point of distance is the reference end of the curve or the element. The magnitude of trapezoidal force is represented by two load intensity  $W_1$  and  $W_2$ . The end point with  $W_1$  may be the reference point. It is sometimes necessary to switch the reference point. This can be done simply choosing the menu item “Change Side” from **Assign** menu.





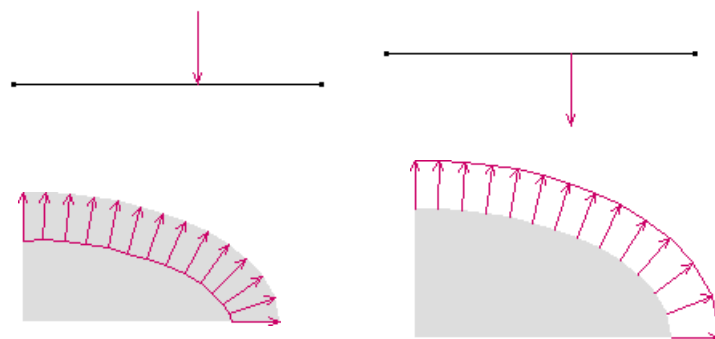
< Result of “Change Side” command>

### ■ Suppressing load selection

Loads can be selected simply by clicking their symbols. You may sometimes be annoyed by unintentionally selecting loads instead of other objects which you actually want to select. This may happen especially when many load symbols are densely located at the point of mouse click. Under such situation, it is convenient to suppress the selection of loads temporarily. Suppressing and desuppressing can be toggled by choosing “Don’t Select Loads” item from menu. The current state is indicated by the check mark in front of the menu item.

### ■ Changing the placement of force symbol

Forces are represented by arrows. These arrows may be placed in two different ways: one with arrows on the object, and the other with arrows off the object. The placement of the arrow can be switched by selecting either “Arrow on Object” or “Arrow off Object” from **Assign** menu.



“Arrow on Object”

“Arrow off Object”

<Placement of force symbol>

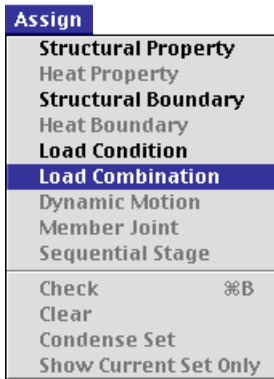
### ■ Limiting the size of force symbol

The size of the arrow representing a force is determined to be approximately proportional to the magnitude of the force. When there are big differences in the magnitudes of the forces, some of arrows become too large to be appropriately drawn on the screen. For such circumstances, it is necessary to limit the size of arrows. This can be done by selecting “Limit Arrow Size” item from **Assign** menu.

### ■ Resetting the size of force symbol

As you continue assigning load conditions, the size of arrow may become inappropriate, even though the load drawing scale can be adjusted using the slide bar of "Load Condition" dialog. In such case, it is desirable to reset the default size of the arrow by selecting “Reset Arrow Size” item from **Assign** menu.

## Load Combination

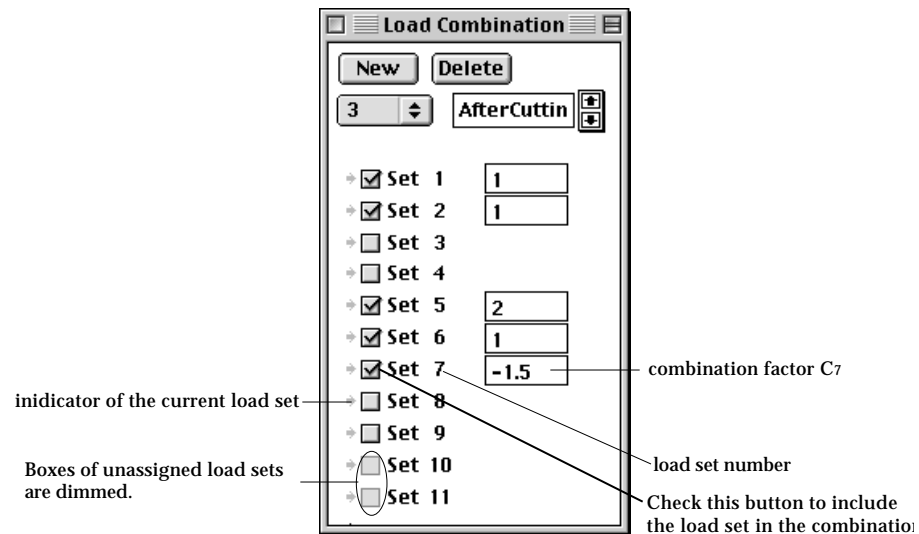


Multiple sets of analysis results can be obtained at once by solving a model with multiple load cases. Each load case is defined by linear combination of the load sets. A force vector  $\mathbf{F}_r$  for the system equations is obtained by linear combination of  $m$  force vectors,  $\mathbf{F}_1, \dots, \mathbf{F}_m$ , which may be expressed by the following equation,

$$\mathbf{F}_c = \sum_{i=1}^m c_i \mathbf{F}_i$$

As many load cases as desired can be formed by various combinations of the load sets, and can be used for independent analyses. Instead of solving the equations with each one of the load cases, the program solves all the cases at once, and thus saves the computing time significantly. The results from various load cases can be combined further with desired factors to form a new sets of analysis results.

The function of load combination is valid only for linear static analysis.



### ■ Defining load combinations

Load combination is meaningful only when there are 2 or more load condition sets applied to the model. A load case is defined by combining all or parts of the load sets with certain combination factors by the following factors.

- 1) Choose "Load Combination" item from **Assign** menu.  
Then, "Load Combination" dialog appears on the screen.
- 2) Create or delete a combination set using **New** or **Delete** button in the dialog.

The usage of the dialog is similar to that of assignment dialogs except that there is no **Assign** button.

- 3) Check the load sets to be include in the combination.

Click the check boxes to turn on or off the inclusion of load sets in the combination. Only the load sets applied to the model can be included in a combination.

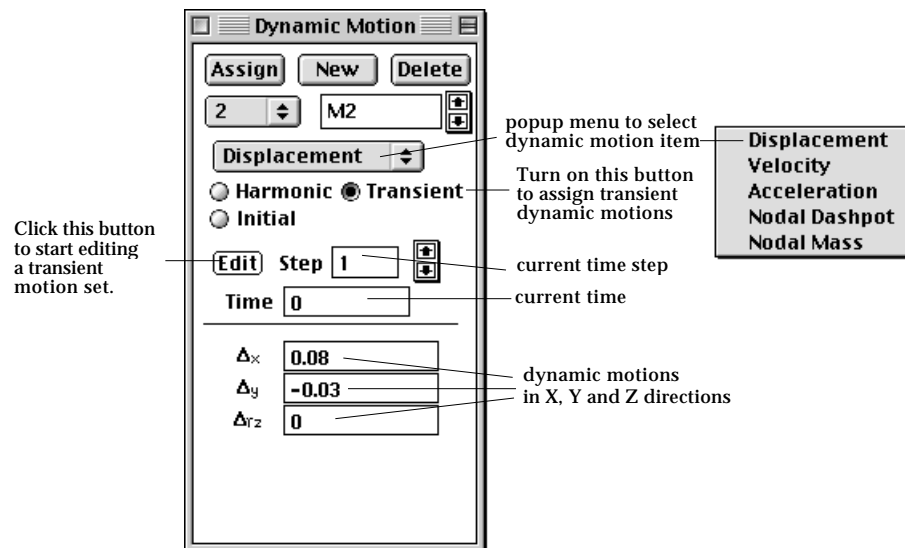
- 4) Insert the combination factors  $c_i$  in the editable text boxes.

There appears an editable text box to the right of the included load set text. Insert the text of the combination factor in the box. The factor can be either positive or negative.

You may define as many load cases as necessary. However, it should be considered that further combination of analysis results is also possible after solving the model with multiple load combinations.

### ■ Handling load condition sets in "Load Combination" dialog

The "Load Combination" dialog may be displayed along with the "Load Condition" dialog as shown in the figure below. You may change the current load set by clicking the corresponding indicator in the "Load Combination" dialog. Alternatively, you may also change the highlighted indicator by selecting a popup menu(Windows:dropdown list) item in the "Load Condition" dialog.

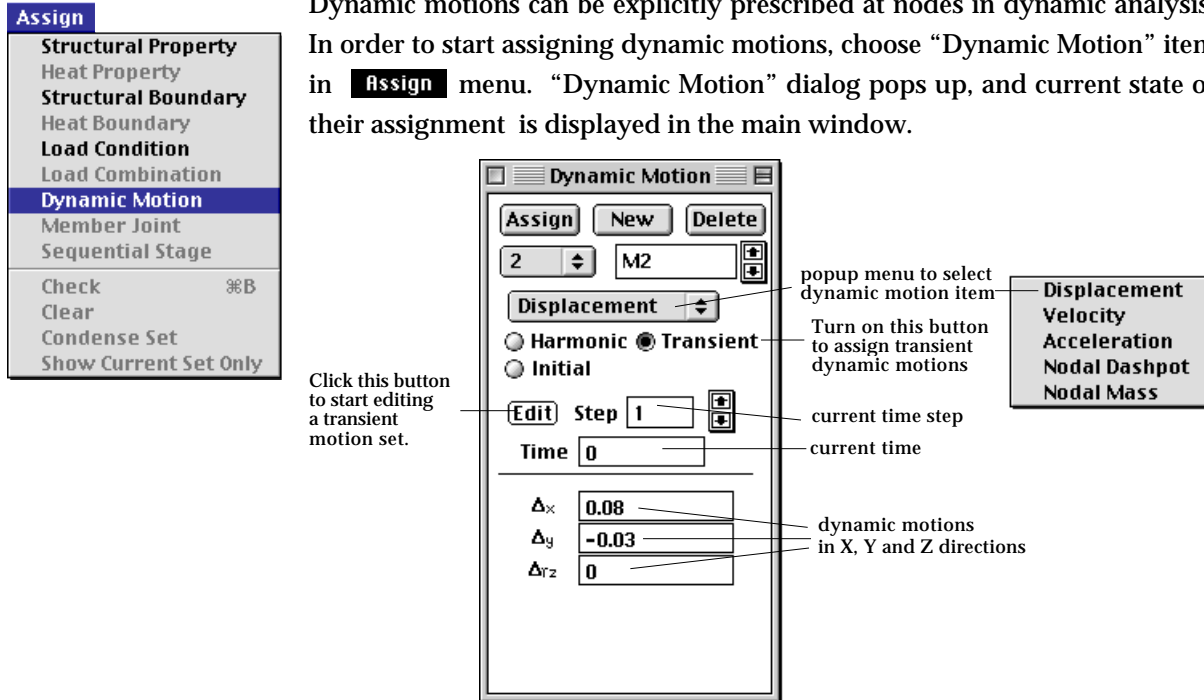


<"Load Combination" dialog displayed along with "Load Condition" dialog>

### ■ Combining analysis results of multiple load conditions

The analysis results with various load combinations can be combined further as described in Chapter 7.

## Dynamic Motions



Dynamic motions can be applied to fixed degrees of freedom as well as to free d.o.f. However, application of motion to fixed d.o.f. tends to increase the solution time.

### Defining and assigning dynamic motion sets

Dynamic motions can be defined and assigned by the unit of data set similar to load condition set. A dynamic motion set is initially created by clicking **New** button of "Dynamic Motion" dialog. If a dynamic motion set is already assigned, additional assignment of the set, either to a single object or multiple objects, creates a new set automatically by duplicating the original one.

The popup menu of "Dynamic Motion" dialog has the following items:

- "Displacement": transnational displacements assigned at nodes.
- "Velocity": transnational velocity assigned at nodes.
- "Acceleration": transnational acceleration assigned at nodes.
- "Nodal Dashpot": damping dashpot attached at nodes.
- "Nodal Mass": mass assigned to nodes.

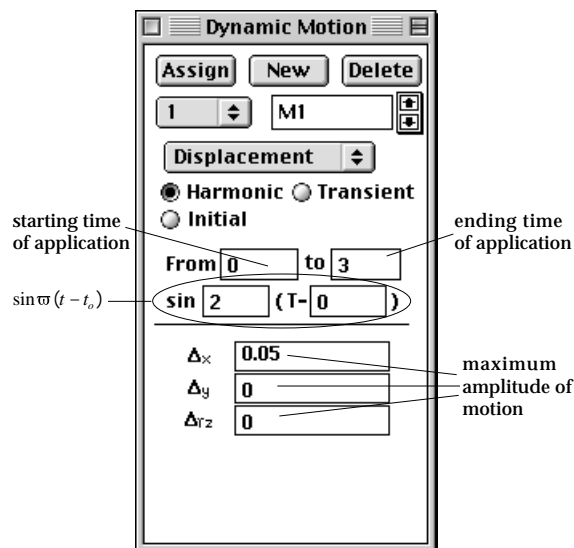
The top 3 items correspond to types of dynamic motion and the other two to dynamic resistance, which is described in the next section.

### ■ Setting the time dependency of dynamic motion

The time dependency of dynamic motion can be set as one of the 3 types, harmonic transient and initial types. In order to set the time dependency, click the corresponding radio button, "Harmonic", "Transient". or "Initial" in "Dynamic Motion" dialog. As you alter the option of time dependency, you will notice the changing items below the radio buttons. Time dependent variation of dynamic motions is defined and modified using these items.

### ■ Defining harmonic motions

A harmonic motion has periodic characteristics with changing magnitude expressed by a sinusoidal function  $\sin \omega(t - t_o)$ . The dynamic motion acting at a specific time is obtained by multiplying the function by the amplitude of motion given at the bottom of the dialog.



Click "Harmonic" radio button to define a set of harmonic motions. The amplitudes of the motion components are defined by inserting the values in the editable text boxes of motion components.

### ■ Defining transient motions

A transient motion is defined by a series of values at a number of time steps, which can be created and edited by the following steps.

- 1) Select an item of dynamic motion from popup menu in "Dynamic Motion" dialog.  
One of "Displacement", "Velocity" and "Acceleration" items should be selected.
- 2) Turn on "Transient" radio button in the dialog.  
Then, the items below the radio buttons take on the appearance shown at the

beginning of this section ("Dynamic Motion"). They include the current step number and the current time. The current time step can be edited by directly changing the number in the box, or by scrolling the number.

- 3) Click **Edit** button.

Then, "Dynamic Motion Series" dialog pops up. The dialog produces editable data items for each time step you create. The data items are equivalent to the data items in "Dynamic Motion" dialog.

**Dynamic Motion Series**

Step	Time	Dx	Dy	Dz
6	1.2	0.02	-0.03	0
7	1.3	0.03	-0.02	0
8	1.4	0.05	0	0
9	1.5	0.03	0.02	0
10	1.6	0.01	0.03	0
11	1.8	-0.01	0.05	0
12	2	-0.03	0.06	0
13	2.1	-0.06	0.05	0
14	2.2	-0.02	0.02	0
15	2.3	0.01	0	0
16	2.5	0.04	-0.02	0
17	2.6	0.07	-0.03	0
18	2.7	0.05	-0.01	0
19	2.8	0.03	-0.02	0
20	2.9	0.06	-0.04	0
21	3	0.08	-0.03	0
22	3.2	0.05	-0.01	0
23	3.3	0.02	0.01	0
24	3.4	-0.02	0.03	0
25	3.5	-0.05	0.05	0

- 4) Click **Add** button to add a time step.

A line of data items will be created for the new time step. In order to delete a time step, click **Delete** button after selecting the corresponding data line.

- 5) Select a line or a cell to edit.

Then, the data on the line or in the cell will be reflected in the editable text boxes at the bottom of the dialog

- 6) Edit the text in the editable text box.

As you edit the text in a text box, it will immediately be echoed in the actual data line.

- 7) Click **O.K.** button after completing creation of time steps.





"Dynamic Motion Series" closes and the data from the dialog are automatically reflected in "Dynamic Motion" dialog.

The data on the last line are assumed to be maintained up to the last time step in the analysis.

### ■ Defining initial motions

Initial motions are used to define the initial state of the analysis at time 0. The type of initial motion is either displacement, velocity or acceleration. Click "Initial" radio button of the "Dynamic Motion" dialog to define and assign initial motions.

### ■ Assigning dynamic motions

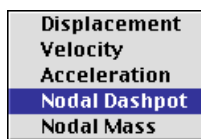
Dynamic motions can be assigned to various types of objects including nodes, curves, surface meshes or volume meshes. However, they are eventually assigned to nodes contained in the assigned objects. You should first choose the selection tool, , ,  or  in order to assign the dynamic motion sets to objects of desired type.

## Defining and assigning nodal dynamic properties

As described in the previous section, the popup menu of "Dynamic Motion" dialog has items other than dynamic motion itself. They are concerned with dynamic properties assigned at nodal points. "Nodal Dashpot" is to assign damping characteristics to nodes, and "Nodal Mass" is to assign concentrated mass to nodes. A nodal dashpot is a damper decelerating nodal movement, and nodal mass carries inertia force conceptually concentrated at nodal points.

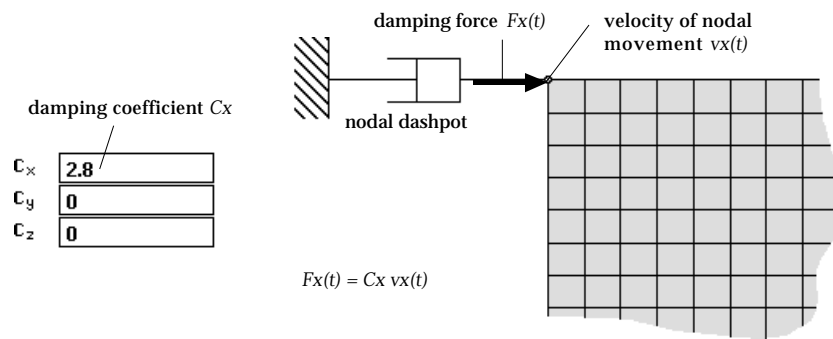
### ■ Nodal dashpot

Nodal dashpot represents a damper attached to a node. A nodal dashpot can be defined and assigned by the following steps:



- 1) Select "Nodal Dashpot" item from popup menu in "Dynamic Motion" dialog.  
The editable text items on the dialog turn into states for defining a damping coefficient for each nodal d.o.f.
- 2) Insert values into the editable text items.  
A damping coefficient is specified independently for each nodal d.o.f. For example,  $C_x$ ,  $C_y$ , and  $C_z$  are damping coefficients for 3 nodal d.o.f in 3-D solid case.
- 3) Select nodes or objects containing the nodes to assign the nodal dashpot..
- 4) Click **Assign** button in the dialog.  
The nodes assigned with the dashpot are marked by red rectangle.

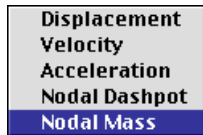




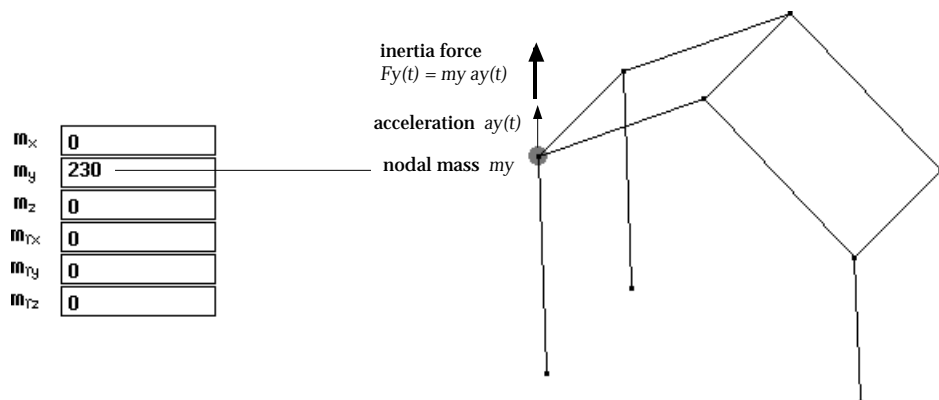
&lt;Conceptual diagram of nodal dashpot&gt;

### ■ Nodal mass

A nodal mass is a mass conceptually concentrated at a node, which makes the nodal movement produce inertia force or moment. Nodal mass can be defined and assigned by the following steps:

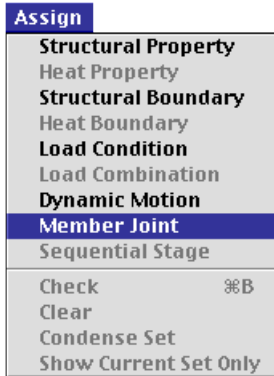


- 1) Select "Nodal Mass" item from popup menu in "Dynamic Motion" dialog.  
The editable text items on the dialog turn into the state to get nodal mass components defined for each nodal d.o.f.
- 2) Insert values into the editable text items.  
The conceptual nodal mass is specified independently for each nodal d.o.f. For example,  $m_x$ ,  $m_y$ ,  $m_z$ ,  $m_{rx}$ ,  $m_{ry}$ ,  $m_{rz}$  are mass components for 6 nodal d.o.f in 3-D frame model. The latter 3 are rotary inertias.
- 3) Select nodes or objects containing the nodes to assign the nodal mass..
- 4) Click **Assign** button in the dialog.  
The nodes assigned with the nodal mass are marked by red rectangle.



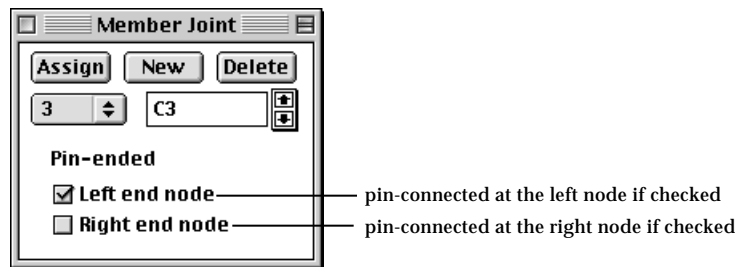
&lt;Conceptual diagram of nodal mass&gt;

## Frame Member Joint Conditions



Frame member joint conditions are meaningful only for 2-D and 3-D frame analysis. Frame members are basically connected rigidly with other members at their joints. But some of them may be pin-jointed. The state of member connection can be defined as member joint condition sets and assigned to corresponding members.

In order to start assigning member joint conditions, choose “Element Joint” item from **Assign** menu. Then, “Member Joint” dialog box appears on the screen.

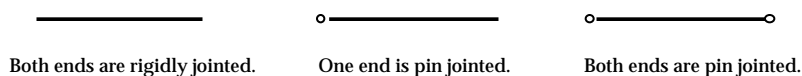


### Defining frame member joint conditions

The frame member joint conditions are defined as a data set using “Member Joint” dialog. There are only two data items, each of which represents the joint condition at one end of a frame member. It is assumed that each frame member has only two end nodes: the left and the right end node. The check mark of the check box in front of “Left end node” or “Right end node” indicates that the corresponding node is pin-jointed. The check mark is turned on and off by clicking the check box. In order to define a truss member joint, turn on both check boxes. Unchecking both of the check box makes the member joint condition with both ends rigidly jointed. Defining such a joint condition is redundant, because all the frame members are originally assumed to be rigidly jointed at both of their ends.

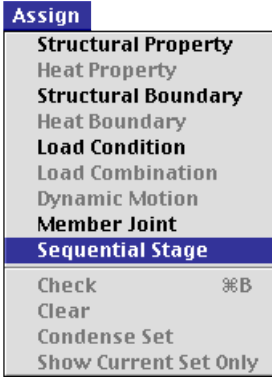
### Assigning frame member joint conditions

In order to assign the currently active member joint condition set, first select the member, and click **Assign** button of the dialog. The member will get the joint condition defined by the set, and will be marked in red colors. The pinned joint is also marked in the drawing of the member as shown below.



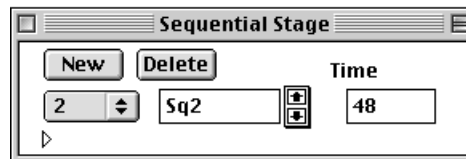
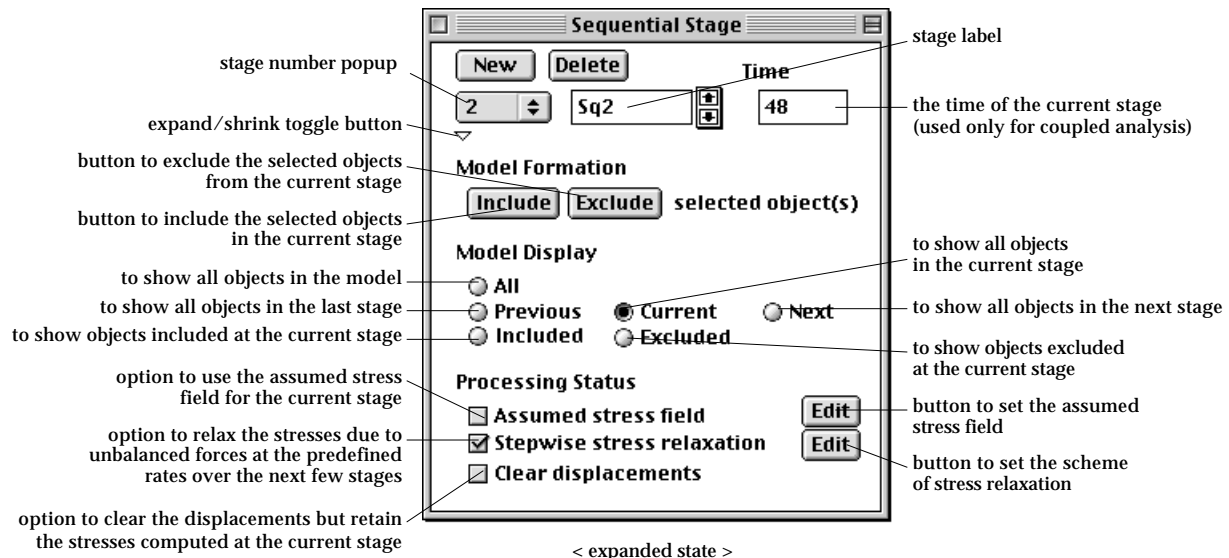
<Representation of member connectivity>

## Sequentially Staged Modeling



Sequentially staged modeling is devised for analysis of objects with changing geometry, material properties and load conditions over a given period of time. Such a modeling is useful in analyzing structures like dams, tunnels and bridges under construction. The sequentially staged modeling option is turned on by checking the "Sequential" button of the "Project Setup" dialog. (Refer to "Project and File" section of Chapter 1.) The menu item, "Sequential Stage" of **Assign** menu is enabled when this modeling option is turned on.

The "Sequential Stage" dialog can be toggled between the expanded and shrunk states using the toggle button. The dialog is in the expanded state, and turns to the shrunk state automatically when an assignment dialog (such as "Property" or "Load Condition" dialog) is opened.



In order to start defining and assigning the sequential stages, choose "Sequential Stage" item from **Assign** menu. Then, "Sequential Stage" dialog opens as shown below. The model of each stage is created by using this dialog. Usage of the dialog items is explained below.



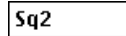
Click this button to create a new stage and display the state of the new stage. The new stage model initially inherit its contents from the last stage model or from the base model.



The current stage is deleted by clicking this button. The next or the previous stage becomes the current stage.



The current stage number is displayed as a popup menu(Windows: drop-down list) item. You may move to the desired stage using this popup menu (Windows: drop-down list).



The label of the current data set. You may label a data set by entering a character string in the text box.

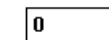


This button is used to scroll up the stage. The current stage is scrolled up by this button. Clicking this button once reduces the current stage number by one.



This button is used to scroll down the stage. The current stage is scrolled down by this button. Clicking this button once increases the current stage number by one.

Time



The point of time for the current stage. This item is used only for structural analysis coupled with transient seepage -analysis.



Expand/shrink toggle button. Click this button to expand or to shrink the dialog.



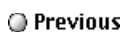
Click this button to include the selected objects into the current stage model. If the selected objects are already included in the current stage model, this button is dimmed.



Click this button to exclude the selected objects from the current stage model. If the selected objects are not included in the current stage model, this button is dimmed.



If this radio button is on, all the objects, i.e. the objects of the base model are displayed. Click this button to show all objects of the base model independent of the stages.



Click this button to display the previous stage model.



Click this button to display the current stage model.



Click this button to display the next stage model.



Click this button to display the objects included at the current stage, i.e., the objects which are not in the last, but in the current stage model.



Click this button to display the objects excluded at the current stage, i.e., the objects which are in the last, but not in the current stage model.

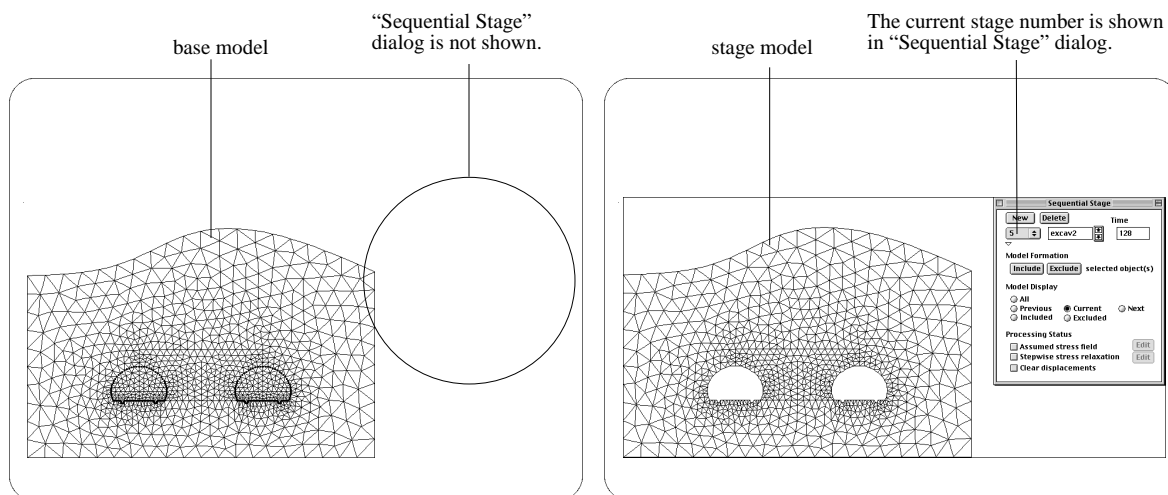
## Concept of sequentially staged modeling

Stage is a core concept of the sequentially staged modeling. This section describes the finite element procedures based on the stages, and the inter-relation of analysis results and modeling data between stages.

### ■ The base model and the stage models

A sequentially staged model consists of 2 or more stages. Each stage has all ingredients of finite element analysis by itself and is modeled to represent the actual situation with changing geometry, properties, and load conditions. The finite element analysis model for a stage is termed here as "stage model", while the model with all data shared by all stages are termed as "base model." A sequentially stage modeling always involves one base model and 2 or more stage models.

The base model is displayed on the screen when the "Sequential Stage" dialog is not shown. The stage model of the current stage is displayed when "Sequential Stage" dialog is opened. You can move to the desired stage either by using the stage number popup menu(Windows: dropdown list), or by using the scroll buttons. The current stage number appears as the popup text. The current stage can also be identified by the label displayed in the label text box. The geometry, properties, boundary conditions and load conditions of the current stage model is displayed in the main window if the dialog is opened and the display mode is set to "Current". At the moment you close the dialog, the display on the main window returns to the base model.



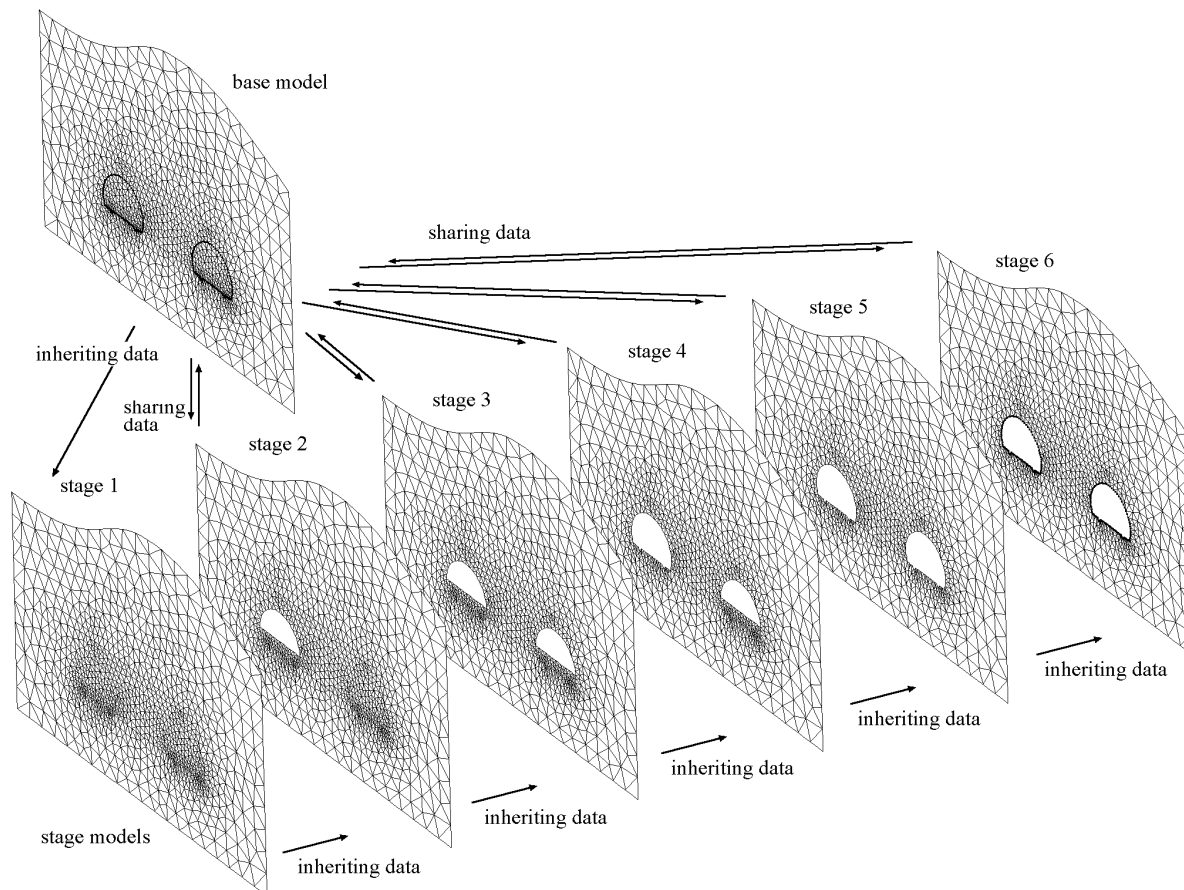
<Base model and stage model>

### ■ Data sharing and inheriting

All the stage models share the data of the base model. In other words, the base model comprises all the data included in the stage models. The first created stage model (the stage model of stage 1) initially inherits its data from the base model, and is later modified to keep only necessary data by using exclusion, assignment or clearing actions. Likewise, a subsequent stage model initially inherits its data from the preceding stage model, and is modified later by various actions.

Creation or modification of geometric objects, i.e., curves and meshes are allowed only for the base model. Thus, new objects cannot be attached to a stage model directly. The geometric objects created for the base model are automatically inherited to all the currently existing stages.

On the other hand, properties and load conditions can be assigned directly to a stage model. Any assignment performed at a stage level is applied to the corresponding stage model, and also included in the base model. But the assignment is not applied to the other stage models. Nevertheless, any assignment to the base model is automatically inherited to all the currently existing stages.



<Concept of data sharing and inheriting>

### ■ Solution process of stage models

A complete processing of finite element solver is performed for each of the stage model. The processes are repeated, starting from the first stage and proceeding sequentially to the last. Each process of a stage model is not independent from the others. A stage model is processed in association with its preceding or subsequent stage. The solution results of a stage model are reflected in the processing of the next stage model. For example, unbalanced forces resulting from excavation at a stage are relaxed over the next few stages with prescribed rates. The solution results such as displacements, stresses and so on are accumulated stage after stage. The data accumulation can be artificially controlled if necessary. As an example, the nodal displacements are cleared at the first stage while the computed stresses are retained, which is usually applied to modeling construction stages in consolidated soil.

### ■ Procedure of sequentially staged modeling

A stage model is constructed in the form of extracting necessary components from the base model. The procedure of sequentially staged modeling can be summarized as follows:

- 1) Create the base model.

It is recommended to complete creating the base model before creating the stages, although the base model may be added or modified in the middle of working with stage models.

- 2) Create stage models and build the geometry of the stage models.

The geometry of a stage is constructed initially by inheriting the geometry from the base model or from the previous stage, and then modified by "Include" and "Exclude" actions.

- 3) Build the data assignment of each stage model.

Property and load condition data of a stage is constructed initially by inheriting the data from the base model or from the previous stage, and then modified by assigning or clearing data.

- 4) Execute the solution process.

The finite element solution process is launched in the same way as non-staged models. Select "Solve" item of **Solve** menu to go through the solution process. (Refer to "Processing of structural analysis" section of Chapter 6.) However, a complete solution process is repeated for each of the stages. The analysis results of a stage are accumulated and applied as a part of input data for the analysis of the subsequent stages.

- 5) Postprocess the analysis results.

The analysis results of the staged model are arranged and saved as a sequential process. The visualization of the analysis results can also be controlled sequentially in accordance with the staged progress.

## Creating stage models

Sequentially staged modeling is initiated by choosing "Sequential Stage" button from **Assign** menu. Stage 1 exists at the beginning of staged modeling. Stage 1 initially includes all the objects created so far, and the current assignment of properties, boundary conditions and load conditions.

### ■ Creating a new stage

A new stage is added to the staged model by clicking **New** button. If the current stage is the last one, the new stage is attached to the last. If the current stage is not the last one, the new stage is inserted between the current and the next stage. Accordingly, the next and the following stages are reassigned with subsequent numbers.

The new stage model initially inherits the geometry and the attributes from the current stage which is now stated as previous stage. The newly created stage takes the status of current stage.

### ■ Deleting a stage

The current stage is deleted by clicking **Delete** button. If the current stage is the last one, the previous stage takes the status of current stage. Otherwise, the next stage takes the status of current stage, and the subsequent stages are renumbered accordingly. There should be at least one stage, and therefore the current stage cannot be deleted if there is only one stage.

### ■ Moving to the desired stage

You can move to the desired stage either by using the stage number popup menu(Windows: dropdown list), or by using the scroll buttons. The current stage number appears as the popup text. The current stage can also be identified by the label displayed in the label text box. the contents of the main window, i.e., geometry, properties, boundary conditions and load conditions are displayed on the basis of the current stage.

## Building the geometry of stage models

In a sequentially staged modeling, each of the stages represents a complete model with all ingredient of finite element analysis, composed of geometry, properties, boundary conditions and load conditions. However, it is not necessary to create independent models for the stages. Instead, a stage can be constructed simply by including and excluding its components from the predefined base model.

### ■ Including the selected objects to the current stage

It is the case in which the objects not in the previous stage is to be included into the



current stage. Before including the objects into the current stage, they may not be displayed at the current stage depending on how the display option is set. In order to make the desired object visible, check "All" or "Last" radio button. Then, select the object, and click **Include** button. If the selected objects are already within the current model, the button is dimmed.

#### ■ Excluding the selected objects from the current stage

It is the case in which the objects in the previous stage is to be exclude from the current stage. Select the objects to exclude, and click **Exclude** button. If the selected objects are not within the current model, the button is dimmed.

#### ■ Handling objects created after creation of some stages

As mentioned previously, it is desirable to complete the creation of the base model and then to start creating the stages. Owing to some reason, however, a part of the base model might have been created after creation of some stages, all or in part. In such situations, the newly created objects are automatically included into all the stages already created. If unwanted objects are included in this way, you should move to the relevant stage, and exclude the objects manually.

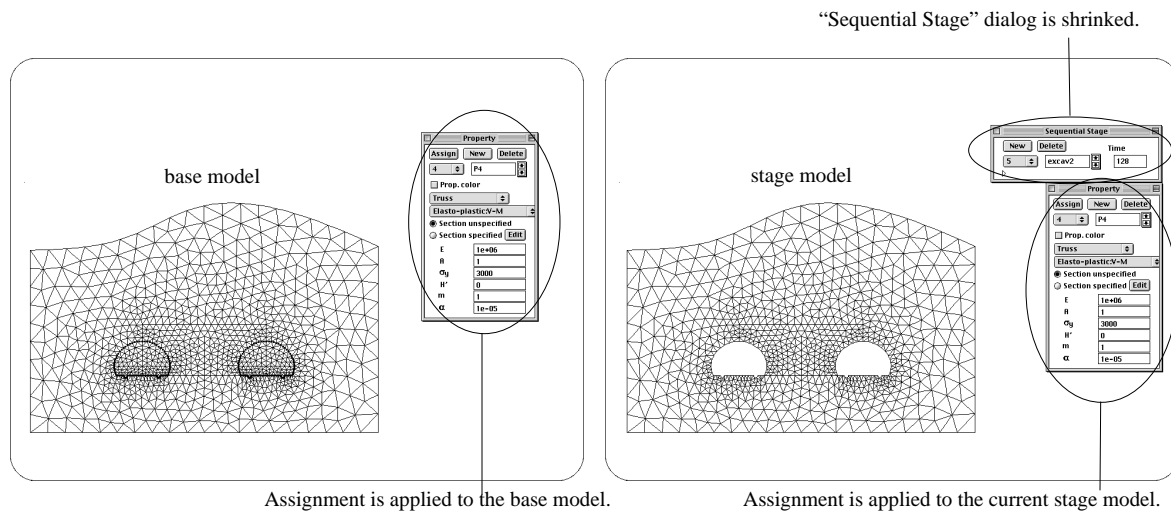
#### ■ "Model Display" options

Only displayed objects are selectable, and thus can be included or excluded as desired. Therefore, it is necessary to control the state of visibility of the desired objects for building the geometry of a stage. The "Sequential Stage" dialog has radio buttons, "All", "Previous", "Current", "Next", "Included" and "Excluded" for the option of object display. The usage of the buttons is described at the beginning of this section.

### Property assignment in sequentially staged modeling

In a sequentially staged modeling, the property assignment also varies with stages. For example, a surface mesh region is assigned with property set A at stage 1, and the property of the same region is altered to set B at stage 2. This is the case of a construction modeling with changing material property in some part. Property assignment can be handled either collectively by working with the base model or individually by working with a stage model.

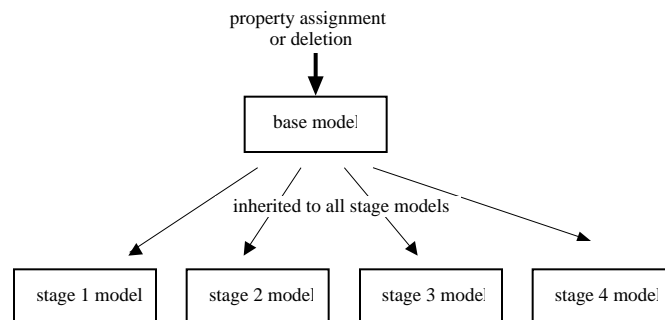
In order to start property assignment, select "Structural Property" item from **Assign** menu. If the "Sequential Stage" dialog is not currently on, the property assignment is performed for the base model. If the dialog is already opened, the dialog shrinks, and the "Structural Property" dialog is positioned at the bottom of the "Sequential Stage" dialog as shown in the figure below. In this case, the assignment is applied to the current stage model.



< Property assignment to the base model and to a stage model >

### ■ Assigning properties to the base model

The base model takes the property assignment and deletion, when "Sequential Stage" dialog is not on. And the assignment to the base model is automatically inherited to all stage models which have the assigned objects. Deletion of a property set in the base model is also applied to all stage models. Thus, property assignment or deletion in the base model overrides the previous assignments in all stage models at once.

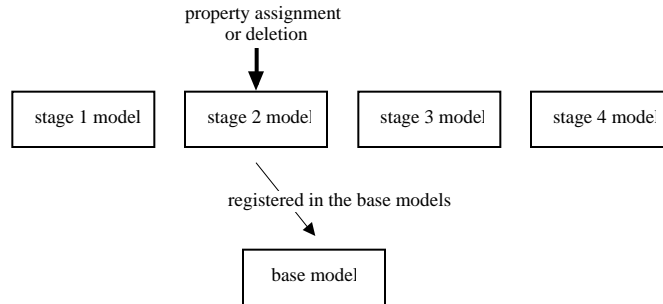


<Property assignment of the base model>

### ■ Assigning properties to stage models

Only one property set can be applied to an object in the base model, although the property of an object can be altered from one stage to the other. Assignment of the changing properties can be achieved by assigning different property sets to the stage models. The current stage model takes the property assignment and deletion, when "Sequential Stage" dialog is on. The current stage is indicated by the popup menu item in the dialog. Property assignment to a stage model does

not affect the assignment of other stage models. However, the assignment is registered in the base model.



<Property assignment of a stage model>

### Load assignment in sequentially staged modeling

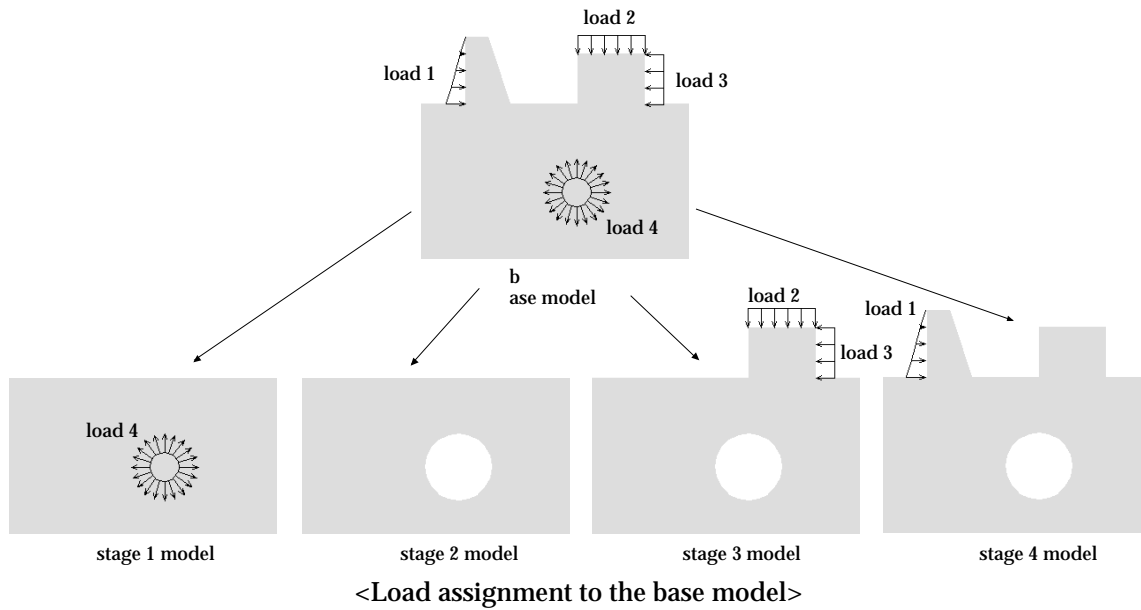
Load assignment can be handled either collectively by working with the base model or individually by working with a stage model. It is similar to the case of property assignment. The load assigned to the base model is inherited to the stage model. However, in solving the whole staged model, a load assignment to an object is effective only for one stage model. In the case of assignment with the base model, the load set is effective at the stage in which the assigned object first appears. In the case of assignment with a stage model, the load set is effective at that stage only.

Assignment of load conditions is initiated by selecting "Load Condition" item from **Assign** menu. If the "Sequential Stage" dialog is not currently on, the load assignment is performed for the base model. If the dialog is already opened, the dialog shrinks, and the "Load Condition" dialog is positioned at the bottom of the "Sequential Stage" dialog as in the case of property assignment.

#### ■ Assigning loads to the base model

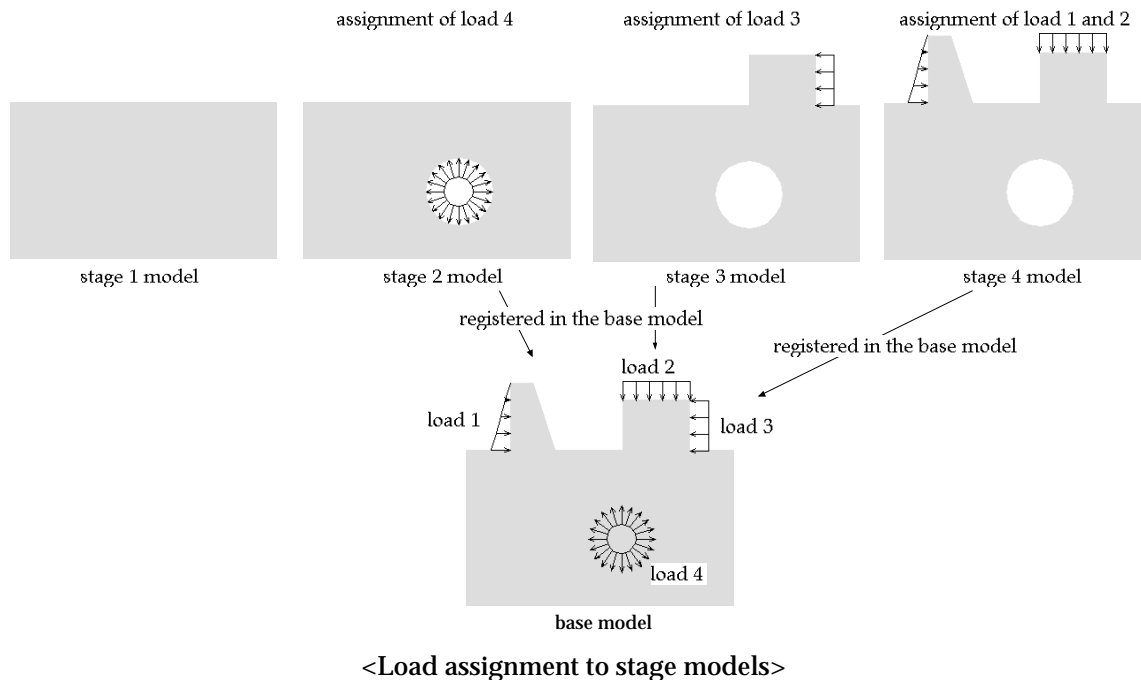
A load condition applied to the base model is inherited to stage models. This does not mean that the load condition is applied to all stage models, but means that the load takes effect at one of the inherited stage models in which the assigned object appears for the first time. Each of the load conditions is involved only once in the system equations, and it is the system equations at the stage the load takes effect. Thus, actual application of the load conditions in the base model is determined automatically by the program, and cannot be controlled by the user.

The following example illustrates how the load assignment of the base model takes effect in stage models. Load 1 to 4 are applied to the base model. Load 4 becomes effective at stage 1, because the object assigned with the load is already in the stage model. Load 2 and 3 take effect at stage 3 in which the assigned object appears. Finally, load 1 appears at stage 4.



### ■ Assigning loads to stage models

A load condition applied to the base model takes effect at the stage the assigned object appears first. Thus, the stage of actual application may not be realized as desired. Such a problem can be solved by assigning the load condition directly to the desired stage. The following figure shows an example illustrating the difference between the load assignment to the base model and the assignment to the stage models.



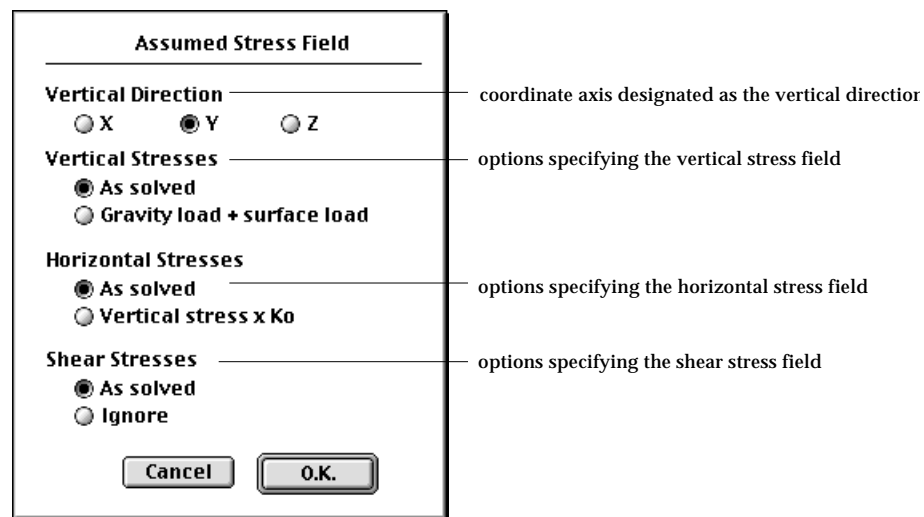
## Control in processing of a sequentially staged model

The solution results of a stage model and their reflection on subsequent stage models can be artificially controlled to embody the actual situation more realistically.

### ■ Assumed stress field of a stage model

The solution results of a stage model can be replaced by assumed stress field as a whole or in part. Such a fabrication of the analysis results is often used to represent the initial state in soil mechanical analyses.

If you check the "Assumed stress field" box, **Edit** button to the right turns to enabled state from disabled state. Click the button to designate how the stress field should be assumed. Then, "Assumed Stress Field" dialog appears.



Using the dialog, the stresses can be controlled as follows:

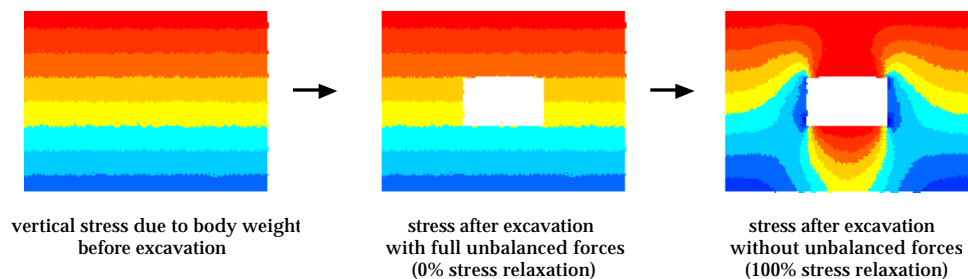
- "Vertical Direction": Select the vertical direction from the coordinate axis, X, Y and Z. This option is enabled only for 3-D problems. The Y direction is assumed to be the vertical direction for 2-D modeling.
- "Vertical Stresses": Select the way how the stresses in the vertical direction should be determined. If "As solved" ratio button is on, the vertical stresses are determined by the finite element solution. Otherwise, the vertical stresses are directly calculated from the vertical loads.
- "Horizontal Stresses": Select the way how the stresses in the horizontal plane should be determined. If "Vertical Stress  $\times K_0$ " option is on, the horizontal stresses are directly calculated from the vertical stresses using the neutral soil pressure coefficient  $K_0$ .

- "Shear Stresses": Select the way how the shear stresses should be determined. If "Ignore" option is on, the shear stresses are set to 0, regardless of the actually computed values.

In order to nullify the rule of stress assumption and use the solved values, uncheck the "Assumed stress field" box.

### ■ Prescribing the staged stress relaxation rate

Excavation during construction can be modeled simply by excluding the elements of the excavation part from the corresponding stage model. Removal of elements results in change of stresses. In sequentially staged analysis, it is sometimes assumed that the stress change does not occur at once, but gradually over a few stages. Such an assumption is often adopted in 2 dimensional modeling of excavation progressing in the direction vertical to the plane. The assumption is based on the concept of unbalanced forces and stress relaxation. At the beginning, there exist unbalanced forces resisting the stress change. Stress relaxation, i.e., transition of stresses to the final state progresses gradually in accordance with the reduction of the unbalanced forces.



< Concept of unbalanced forces and stress relaxation >

The gradual stress transition is represented by the stress relaxation rates prescribed for the stages subsequent to the excavation. The procedure of prescribing the stress relaxation rates is as follows:

- 1) Check "Stepwise stress relaxation" box in the "Sequential Stage" dialog.  
Only when the box is checked, **Edit** button to the right is enabled.  
Otherwise, it is disabled.
- 2) Click **Edit** button to open "Stress Relaxation Rate" dialog.  
The stress relaxation rate is prescribed using this dialog.
- 3) Create the lines of stages by clicking **Add** button.  
The stage offset indicate the number of stages from the current stage. The offset of the current stage is 0. Create as many stages as the relaxation rate is prescribed.
- 4) Set the relaxation rates.

Set the relaxation rate for each line of stages. There are 2 columns of relaxation rates. "No Refilling" column prescribes the relaxation rates of excavation without refilling. "w/ Refilling" column applies to the relaxation rate of excavation with later refilling.

- 4) Click **O.K.** button to complete prescribing the stress relaxation rate.

The relaxation rate data is saved to a text file by clicking **Save** button, and imported from a text file by clicking **Import** button.

### ■ Clearing displacements

There is an option to clear all the displacements accumulated up to this stage. Check "Clear displacement" box to remove the displacements but retain the stresses. This option is often used at the initial stage of soil mechanical analysis to represent the initial state of stresses due to body weight.

The dialog box titled "Stress Relaxation Rate" contains a table with the following columns: "Stage Elapse", "Time", "No Refilling", and "w/ Refilling". The "Stage Elapse" column has values 0, 1, and 2. The "Time" column has values 0, 10, and 20. The "No Refilling" column has values 0.4, 0.3, and 0.3. The "w/ Refilling" column has values 1, 0, and 0. Annotations point to these values with labels: "number of stages from the current stage" (pointing to Stage Elapse), "time from the current stage" (pointing to Time), "stress relaxation rate for excavation without refilling" (pointing to No Refilling), and "stress relaxation rate for excavation with refilling" (pointing to w/ Refilling). Below the table, there are radio buttons for "Apply by stage" (selected) and "Apply by time". At the bottom, there are buttons for "Add", "Delete", "Save", "Import", "Cancel", and "O.K.". Annotations point to these buttons: "delete the selected line" (pointing to Delete), "add a new line" (pointing to Add), "save the data in a text file" (pointing to Save), and "read the data from a text file" (pointing to Import). To the right of the dialog box, four stress distribution plots are shown, corresponding to different stage elapse values: "pre-excitation stage", "stage elapse =0 (excavation stage)", "stage elapse =1", and "stage elapse =2". The plots show the stress distribution in a cross-section of an excavation, with colors representing different stress levels. The "stage elapse =0" plot is annotated with "40% stress relaxation (60 % unbalanced forces)", "stage elapse =1" with "70% stress relaxation (30 % unbalanced forces)", and "stage elapse =2" with "100% stress relaxation (0 % unbalanced forces)".

Stage Elapse	Time	No Refilling	w/ Refilling
0	0	0.4	1
1	10	0.3	0
2	20	0.3	0

Annotations for the dialog box:

- number of stages from the current stage (pointing to Stage Elapse)
- time from the current stage (pointing to Time)
- stress relaxation rate for excavation without refilling (pointing to No Refilling)
- stress relaxation rate for excavation with refilling (pointing to w/ Refilling)
- delete the selected line (pointing to Delete)
- add a new line (pointing to Add)
- save the data in a text file (pointing to Save)
- read the data from a text file (pointing to Import)

Annotations for the stress distribution plots:

- pre-excitation stage
- stage elapse =0 (excavation stage)
- 40% stress relaxation (60 % unbalanced forces)
- stage elapse =1
- 70% stress relaxation (30 % unbalanced forces)
- stage elapse =2
- 100% stress relaxation (0 % unbalanced forces)

< Prescribing the stress relaxation rate >





## **Chapter 6**

## **Finite Element Processing**



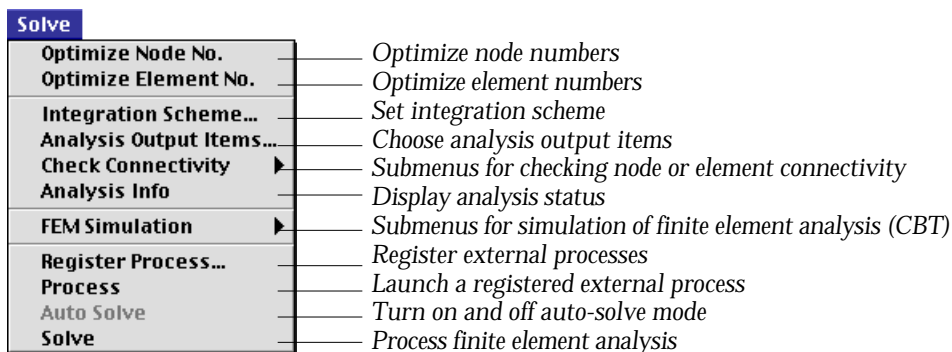
## Chapter 6 Finite Element Processing

Finite element analysis consists of several steps from creating geometric coordinate data to visualizing the analysis results.

The core of the finite element analysis is to form the system equations using the data prepared as described in the previous chapters, and to solve them. This procedure is called “processing of finite element analysis,” and termed here as “finite element processing.” Among all the procedures in finite element analysis, this is the one which requires minimum user interaction. Once the processing begins, it goes on continuously by itself to the end without requiring user intervention. However, the processing is the step which involves the most intensive computation. This usually demands huge computational resources in terms of computing time and memory space. The computing time for processing varies largely depending on the size of the problem: from less than a second to several hours or more. The computing time as well as the memory space required for finite element processing increases drastically as the size of the problem becomes larger, in going from 2-D to 3-D, linear to nonlinear, static to dynamic.

Computational efficiency in terms of computing time and memory space is an important issue in the finite element processing. There are a number of factors affecting computational efficiency. Either node numbering or element numbering depending on the solution scheme is one of the major factors determining the efficiency. Optimizing the node numbering or the element numbering is an essential step which should be taken prior to the processing. Accuracy as well as computational efficiency is affected also by integration schemes and element types employed in the processing. Therefore, they should be manipulated by the users for the best accuracy and efficiency.

This chapter describes the usage of functions related with finite element processing and its computational efficiency. These functions are arranged as items of **Solve** menu as shown in the figure below.



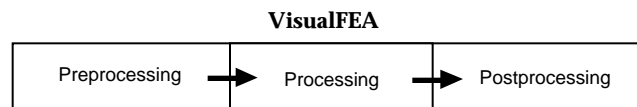
## Getting Finite Element Solution

Assembling and solving finite element equations involve many complicated steps and a huge amount of computation. For you as an end-user, however, it is usually a simple single step procedure. All you have to do is to choose an appropriate menu item. In spite of this simplicity, it is important for you to understand the mechanism of this procedure in order to get a proper solution with maximum efficiency.

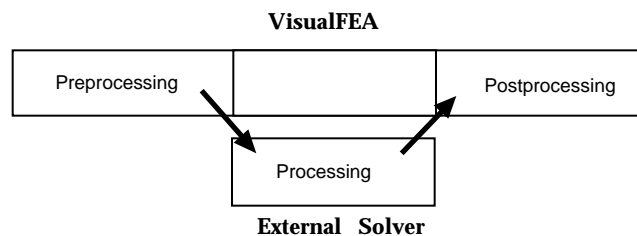
### Solver of finite element analysis

There is software specialized in finite element processing. They are commonly called “solver.” VisualFEA has the capability of such as solver embedded in itself. However, you may use VisualFEA together with other solvers provided by third party, or developed by yourself. In other words, you may use VisualFEA for creating a finite element model (preprocessing) and visualizing the analysis results (postprocessing), and use other solvers for processing. Such a finite element processor is termed here as “external solver.”

There are two paths for doing finite element analysis with VisualFEA: one is relying only on VisualFEA through the whole procedures, and the other combining an external solver for processing and VisualFEA for pre- and postprocessing.



Use of intuitiveFEM for whole procedures



Combined use of VisualFEA and an external solver

<Alternative procedures of FEA using VisualFEA>

The procedure of finite element analysis based only on VisualFEA is simpler, more straightforward and more efficient than that based on the combined usage of VisualFEA and an external solver. However, the solver embedded in VisualFEA may not cover the range of problem you want to solve. In such a circumstance,

you should employ an external solver.

It is your responsibility to make data interface between VisualFEA and external solvers, unless there exists software interfacing them. The data interface is described in Chapter 10.

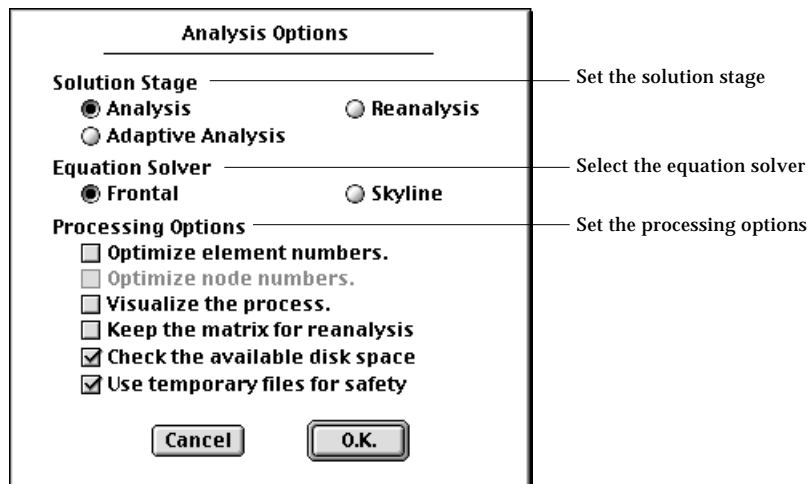
## Processing of structural analysis

The simplest method of getting a finite element solution is to use VisualFEA's own processing capability. The finite element solver's capabilities are embedded within VisualFEA. This makes VisualFEA much simpler to use than other finite element programs which has separate module for processing, preprocessing and postprocessing.

In order to get into the processing stage for solution, choose "Solve" item from **Solve** menu. Then, "Analysis Options" dialog appears on the screen. The contents of the dialog vary depending on the type of solution as will be described below. Set the dialog items as desired and click **O.K.** button. Then, the processing starts, and goes on up to the completion of all the necessary computation including assembling the system equations and solving them.

### ■ Setting analysis options for linear static analysis

Choosing "Solve" item from **Solve** menu will pop up "Analysis Options" dialog as shown below, if you have set the solution type as linear static. *The solution type is initially set as linear static, and remains as it is, unless you have checked any one of the check boxes under "Solution Type", i.e., "Material nonlinear", "Geometric nonlinear" and "Dynamic."*



The dialog has a few items setting the options related to the finite element solution procedure.

- **solution stage:** The solution stage may be set as one of “Analysis,” “Reanalysis” and “Adaptive.” If you turn on “Adaptive” radio button, the solution procedure turns into adaptive process. The analysis options for adaptive analysis are described in the next section. So, only “Analysis” and “Reanalysis” options are described in this section.

The initial default setting of the solution stage is “Analysis”, which leads to the normal procedure of processing based on assembling equations and solving them. Once the system equations are assembled and solved, they may be used for subsequent processing by setting the option as “Reanalysis”. The reanalysis option will eliminate most of the computing time for assembling and decomposing the system equations. The reanalysis option is enabled and valid only under the following conditions.

- The system equations should have been solved in the previous session.
- The files containing the system equations should exist in the same directory (folder) as the data file.

*In order to keep these file for reanalysis in next sessions, check “Keep the matrix for reanalysis” item in the “Analysis Options” dialog.*


- Geometry, element properties, and boundary conditions should not have been altered since the system equation files were created.

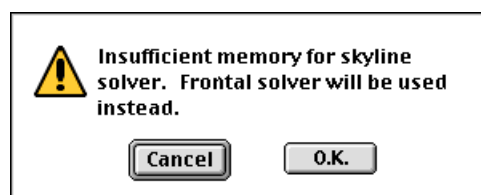
*Loads may be changed.*

- The analysis type is linear static, and not adaptive nor sequential.
- **equation solver:** You may choose one of two methods of assembling and solving equations: frontal method and skyline method. They are typical and the most widely used methods in finite element analysis solvers.

Skyline method uses only CPU memory, while frontal method relies much on auxiliary memory such as a hard disk. Accordingly skyline method demands much larger CPU memory space than frontal method. Skyline method usually works faster than the frontal method which requires frequent reading and writing with the auxiliary memory.

The default setting is “Frontal.” If your computer is equipped with huge CPU memory, and you want faster solution, choose “Skyline.” Otherwise, you should keep the option as “Frontal.”

In case you chose “Skyline,” but the memory space is not sufficient, the software will notify this by the following message box. If you click  button of the dialog, the software will automatically switch the solver to “Frontal.”



- processing options: You can turn on or off each of the processing options by clicking the check box in front of each item. These settings are applied during the processing stage.
  - “Optimize element number” : This item is enabled only when the equation solver is set as “Frontal”. If this option is turned on, optimization of element numbering is automatically done prior to assembling the system equations.
  - “Optimize node number” : This item is enabled only when the equation solver is set as “Skyline”. If this option is turned on, optimization of node numbering is automatically done prior to assembling the system equations.
  - “Visualize the process” : If this option is turned on, a graphical rendering of the model is provided along with the status of the element stiffness matrix assembly.
  - “Keep the matrix for reanalysis” : The system equation files are created at the start of matrix assemblage and removed at the end of the processing. In order to keep these files for reanalysis, this option should be turned on.
  - “Check the available disk space” : If this option is turned on, the available disk space is checked while processing is going on. If the disk space is not sufficient, the processing will pause with the following notice so that you may secure enough space and resume the processing.



- “Use temporary file for safety” : If the processing is abnormally interrupted due to system failure or any other reasons, the data file may be spoiled or lost. In order to avoid such risks, turn on this option. Then, a duplicate of the data file will be created temporarily and used during the processing, and it will replace the original file when the processing is successfully completed.
- The processing procedure is initiated when you click  button of the dialog, after setting all the appropriate items.

### ■ Setting analysis options for adaptive analysis

If you click "Adaptive analysis" radio button of "Analysis Options" dialog, the dialog expands with additional items as shown below. The upper part of the dialog has the original items, and the bottom half includes new items as follows:

- termination criterion: The adaptive iteration continues until one of the conditions set for its termination is satisfied. These conditions are termed here as the termination criterion. You may validate or invalidate each one of the following termination criteria by checking or unchecking the boxes in front of them. If the box is checked, the corresponding criterion is applied.
  - number of iteration cycles: The number of iteration cycles can be restricted

by checking this item, and setting the number. Iteration terminates when the number of cycles reaches the number.

- energy norm error: Iteration terminates when maximum energy norm error over the whole solution domain gets smaller than the criterion set as the limiting energy norm error.

If both of the termination criteria are checked, iteration terminates when any one of them is fulfilled.

The dialog box is titled "Analysis Options". It contains several sections with checkboxes and radio buttons. The "Termination Criterion" section has two checked items: "Number of iteration cycles" with a value of 2, and "Energy norm error" with a value of 5%. The "Handling of Intermediate Models" section has two radio buttons: "Keep all the intermediate models." (selected) and "Keep the original and the final models". The "Visualization of Adaptive Process" section has two checked items: "Display updated mesh." and "Display energy norm error.".

Analysis Options

Solution Stage

☐ Analysis ☐ Reanalysis

☒ Adaptive Analysis

Equation Solver

☒ Frontal ☐ Skyline

Processing Options

☐ Optimize element numbers.

☐ Optimize node numbers.

☐ Visualize the process.

☐ Keep the matrix for reanalysis

☒ Check the available disk space

☒ Use temporary files for safety

Adaptive Analysis Options

Termination Criterion

☒ Number of iteration cycles 2

☒ Energy norm error 5 %

Handling of Intermediate Models

☒ Keep all the intermediate models.

☐ Keep the original and the final models

Visualization of Adaptive Process

☒ Display updated mesh.

☒ Display energy norm error.

Cancel O.K.

Set the criteria for terminating the adaptive iterations.

Decide how to handle the data from intermediate iteration cycles.

Decide whether to display the intermediate adaptive process.

- handling of intermediate models: It is the option determining how to treat the model data created at the intermediate stages of iterations.
  - "Keep all the intermediate models": If this radio button is turned on, the modeling data and analysis results obtained during the intermediate cycles of adaptive iteration are saved, and can be retrieved for later use.
  - "Keep the original and the final models": If this radio button is turned on, the intermediate modeling and analysis data are discarded, and only the original and the final data are saved.
- visualization of adaptive process: It is the option related with visualizing the model and/or energy norm error while the adaptive iteration process is going on.
  - "Display updated mesh": If this radio button is turned on, the meshes generated at each step of adaptive iterations are plotted.
  - "Display energy norm error": If this radio button is turned on, the energy norm error distribution is displayed by contour at each step of adaptive



iterations. (This option may not work for the current version of VisualFEA.)

### ■ Setting analysis options for dynamic analysis

Choosing "Solve" item from **Solve** menu will pop up "Dynamic Analysis Options" dialog as shown below, if you have set the solution type as dynamic. The solution type can be set by using the "Project Setup" dialog.

The screenshot shows the 'Dynamic Analysis Options' dialog box. It is divided into several sections with various settings and checkboxes. Annotations with lines pointing to specific controls are provided on the right side of the dialog.

Section	Control	Value / Option	Description
Solution Method	Direct integration	<input type="radio"/>	Set the way of computing mass matrix
	Mode superposition	<input checked="" type="radio"/>	
	Modal analysis	<input type="radio"/>	
Integration Method	Central difference (explicit)	<input type="radio"/>	Set the time integration method
	Newmark	<input checked="" type="radio"/>	
Equation Solver	Frontal	<input type="radio"/>	Set the constants used with the integration method
	Skyline	<input checked="" type="radio"/>	
	Sparse	<input type="radio"/>	
Processing Options	Optimize element numbers.	<input type="checkbox"/>	Set the constants used with the integration method
	Optimize node numbers.	<input type="checkbox"/>	
	Visualize the process.	<input type="checkbox"/>	
	Keep the matrix for reanalysis	<input checked="" type="checkbox"/>	
	Check the available disk space	<input checked="" type="checkbox"/>	
Mass Matrix	Lumped	<input checked="" type="radio"/>	Set the way of computing mass matrix
	Consistent	<input type="radio"/>	
Dynamic Modes	Number of dynamic modes	10	Set the number of dynamic modes
	Number of time steps	200	
Time Step Size	Time step size	0.001 sec	Set the step size for time integration
	Rayleigh Damping	<input checked="" type="checkbox"/>	
Stiffness Damping	Stiffness damping ratio	0.01	Set the Rayleigh damping
	Mass damping ratio	0.01	
Modal Damping	Mode Equivalent Rayleigh Damping	<input type="checkbox"/>	Set the mode equivalent Rayleigh damping
	Modal Damping	<input type="checkbox"/>	
Newmark Parameters	Alpha	0.2500	Set the constants used with the integration method
	Delta	0.5000	

Buttons: Cancel, O.K.

- solution method : This option determines how to get the dynamic analysis results. VisualFEA supports 3 methods of performing dynamic analysis.
  - direct integration: No transformation is applied for integration in time. The nodal displacements are obtained directly at each time step.
  - mode superposition: Time integration is operated on the participation factors of dynamic modes. Thus, the dynamic modes are extracted first through eigenvalue analysis, and the system equations are formed in terms of participation factors of these dynamic modes. The nodal displacements are obtained by superposing the dynamic modes at each time step.
  - modal analysis: Only dynamic modes are extracted, No time integration is performed. Other analysis results including nodal displacements are not computed.
- integration method : This applies to the integration in time for both the direct integration method and the mode superposition method.
  - "Central difference (explicit)": A explicit integration method, in which the stiffness matrix is not decomposed. The mass matrix is decomposed only

when consistent mass matrix is used. The solution may diverge if the time step is larger than the critical value.

- "Newmark": An implicit method with linear acceleration controlled by parameters  $\gamma$  and  $\beta$ , which can be set by the user.
- "Wilson Theta": An implicit method with linear acceleration controlled by an input parameter  $\theta$ , which can be set by the user.
- mass matrix : There are following two options in computing the element mass matrix.
  - "Lumped": The mass matrix is computed by assuming that element mass is concentrated at nodal points.
  - "Consistent": The mass matrix is computed by interpolation consistent with that used for the stiffness matrix.
- number of dynamic modes : To specify the number of dynamic modes used for mode superposition. This item is valid only when the solution method is set as mode superposition.
- number of time steps : The number of steps included for time history analysis. The total duration of the analysis is determined by the number of steps and the step size which is the next input item.
- time step size : The length of time from one step to the next. Equal step size is assumed for the whole duration of the analysis.
- Rayleigh damping : If this item is checked, Rayleigh damping is assumed, which is the form of  $\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}$ . And the following 2 sub-items pop up.
  - "Stiffness damping ratio": This is the stiffness damping coefficient of the above equation.
  - "Mass damping ratio": This is the mass damping coefficient of the above equation.
- mode equivalent Rayleigh damping : If this item is checked, Rayleigh damping is assumed, but is represented by 2 modal damping ratios which appear as additional input items.
  - "Mode 1 damping ratio":  $\xi_1$
  - "Mode 2 damping ratio":  $\xi_2$

<input checked="" type="checkbox"/> Mode Equivalent Rayleigh Damping	
Mode 1 damping ratio	0.1
Mode 2 damping ratio	0.1

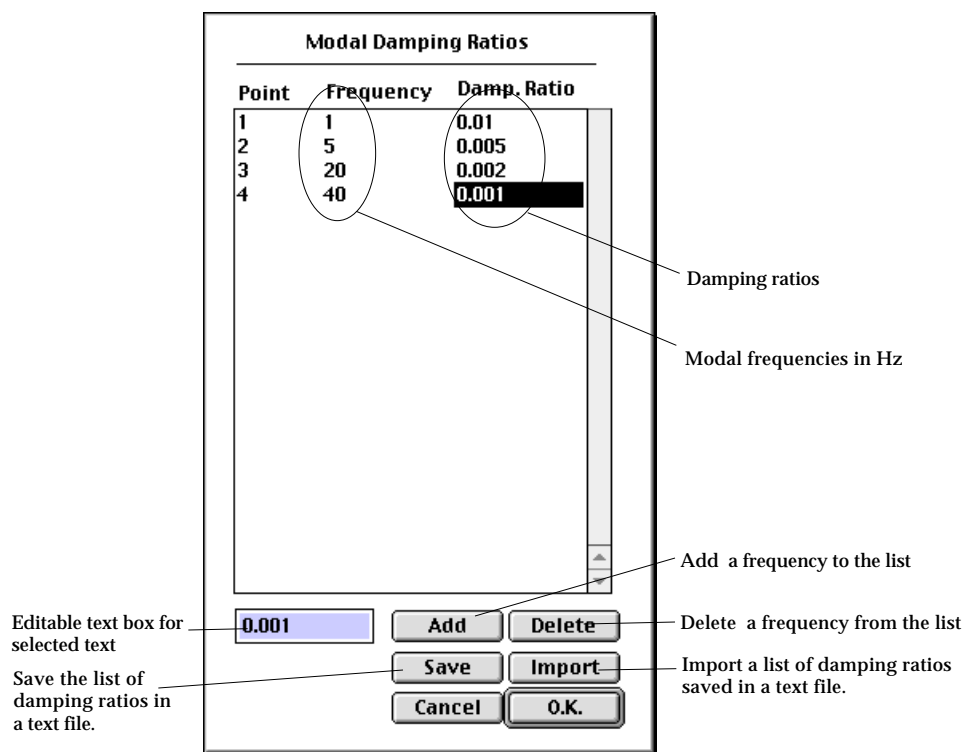
There is a following relationship between the Rayleigh damping coefficients( $\alpha$  and  $\beta$ ) and the modal damping ratios ( $\xi_1$  and  $\xi_2$ ).

$$\xi_1 = \frac{\alpha}{2\omega_1} + \frac{\beta\omega_1}{2}$$

$$\xi_2 = \frac{\alpha}{2\omega_2} + \frac{\beta\omega_2}{2}$$

- modal damping : One method of assigning damping characteristic is to assume an individual damping ratio for each dynamic mode. It is termed here as modal damping. However, information on dynamic modes is not available prior to completing the modal analysis. Thus, the damping ratio are specified as a function of modal frequency.

If you click this item, there appears  button which is used to launch "Modal Damping Ratio" dialog. A table of modal frequencies and paired damping ratio can be specified using this dialog. The damping ratio for a given frequency is estimated by interpolating the values given in this table.



- Acceleration parameters : parameter(s) for linear acceleration of time integration. There are different parameters depending on the method of time integration. The input items change as the method of integration changes.
  - central difference method: no input parameters for this method of integration.
  - Newmark: There are 2 parameters  $\alpha$  and  $\delta$ . The default values,  $\alpha=0.25$  and  $\delta=0.5$  are used for unconditionally stable solution.

#### Newmark Parameters

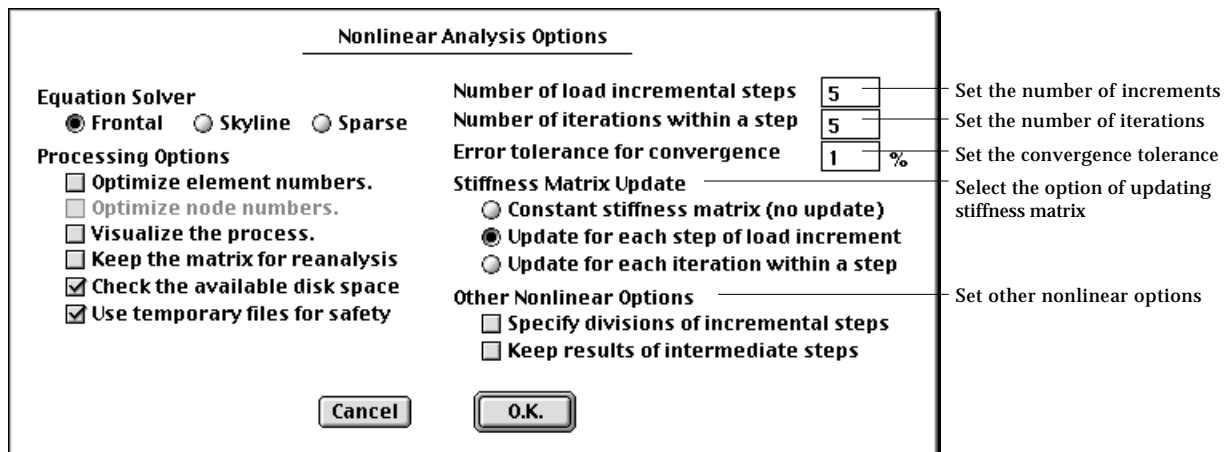
Alpha  Delta

- Wilson : There is a parameter . The default value is  $\theta = 1.4$ . The value of  $\theta$  should be 1.37 or greater for unconditionally stable solution.

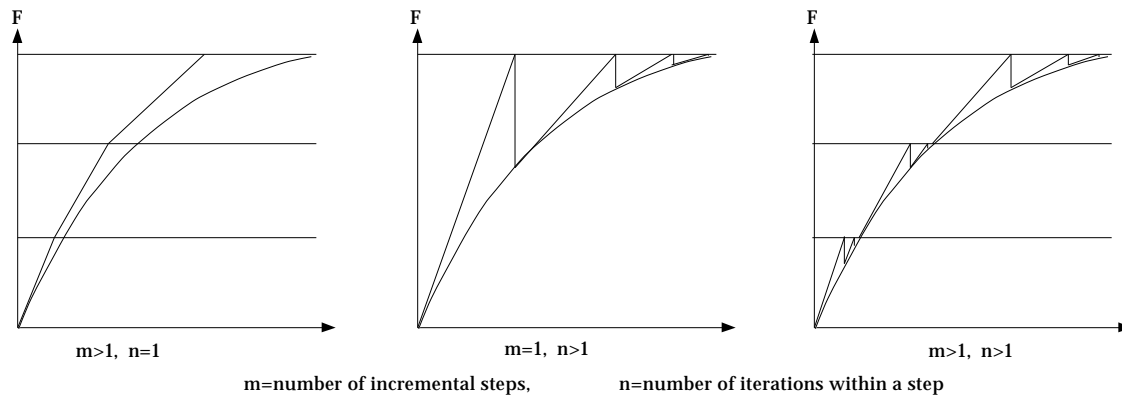
Wilson Theta  
 Theta

### ■ Setting analysis options for nonlinear analysis

Choosing "Solve" item from **Solve** menu will pop up "Nonlinear Analysis Options" dialog as shown below, if you have set the solution type as material nonlinear, or geometric nonlinear. The solution type can be set by using the "Project Setup" dialog.

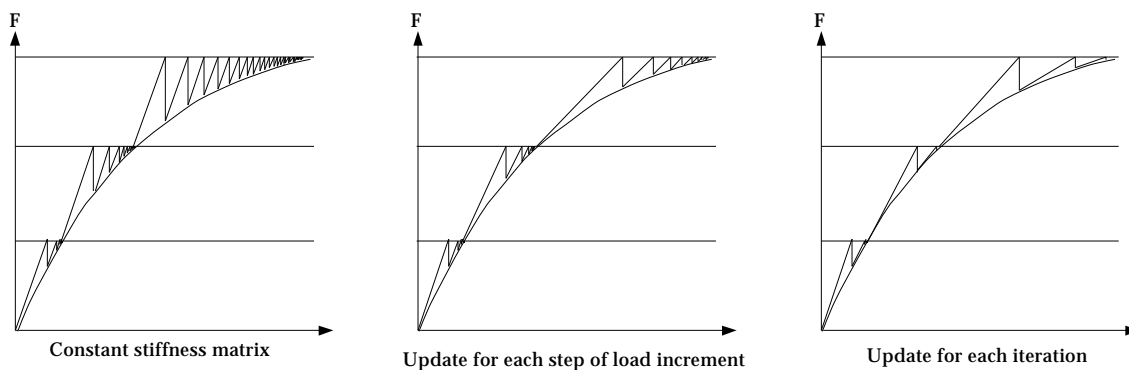


- number of load incremental steps : It is the number of steps for solution of a nonlinear problem by incremental method. If this value is 1, a simple iterative solution is applied. Otherwise, the total load is divided into as many segments as this number, and applied incrementally through the nonlinear solution process.
- number of iterations within a step : The maximum limit in the number of iterations within an incremental step for solution of nonlinear equation. If this value is 1, simple incremental procedure is applied.
- error tolerance for convergence : This convergence criterion applies to the iterative procedure within an incremental step. The iteration is terminated if either the percentage of the residual force falls below this level, or the number of iterations reaches the maximum limit.



< Number of load increments and number of iterations >

- options for stiffness matrix update: There are following 3 options of updating stiffness matrix throughout the incremental and iterative process.
  - "Constant stiffness matrix (no update)": If this button is turned on, the stiffness matrix is not updated throughout the whole incremental and iterative process. Thus, the stiffness matrix is computed only once and no additional time is required for its update. However, this option leads to large number of iterations as shown in the figure below.
  - "Update each step of load increment": If this button is turned on, the stiffness matrix is updated only for the first iteration of each load increment. The stiffness matrix remains constant for all iterations within a load incremental step.
  - "Update each iteration within a step": If this button is turned on, the stiffness matrix is updated for every iteration throughout the whole process. This option takes more time for updating stiffness matrix, but requires smaller number of iterations.



< Schemes of stiffness matrix update >

- other nonlinear option:
  - "Specify division of incremental steps" : This item is not checked by default, and the sizes of all load increments are equal. If you check this item, the following "Load Incremental Steps" dialog pops up. Initially the editable text boxes are filled with uniformly divided incremental portions of the load. These incremental proportions can be modified by editing the text boxes

Load Incremental Steps	
1	0.1
2	0.2
3	0.3
4	0.4
5	0.5
6	0.6
7	0.7
8	0.8
9	0.9
10	1
11	
12	
13	
14	
15	
16	
17	
18	
19	
20	

Set the incremental proportion of loads

Cancel O.K.

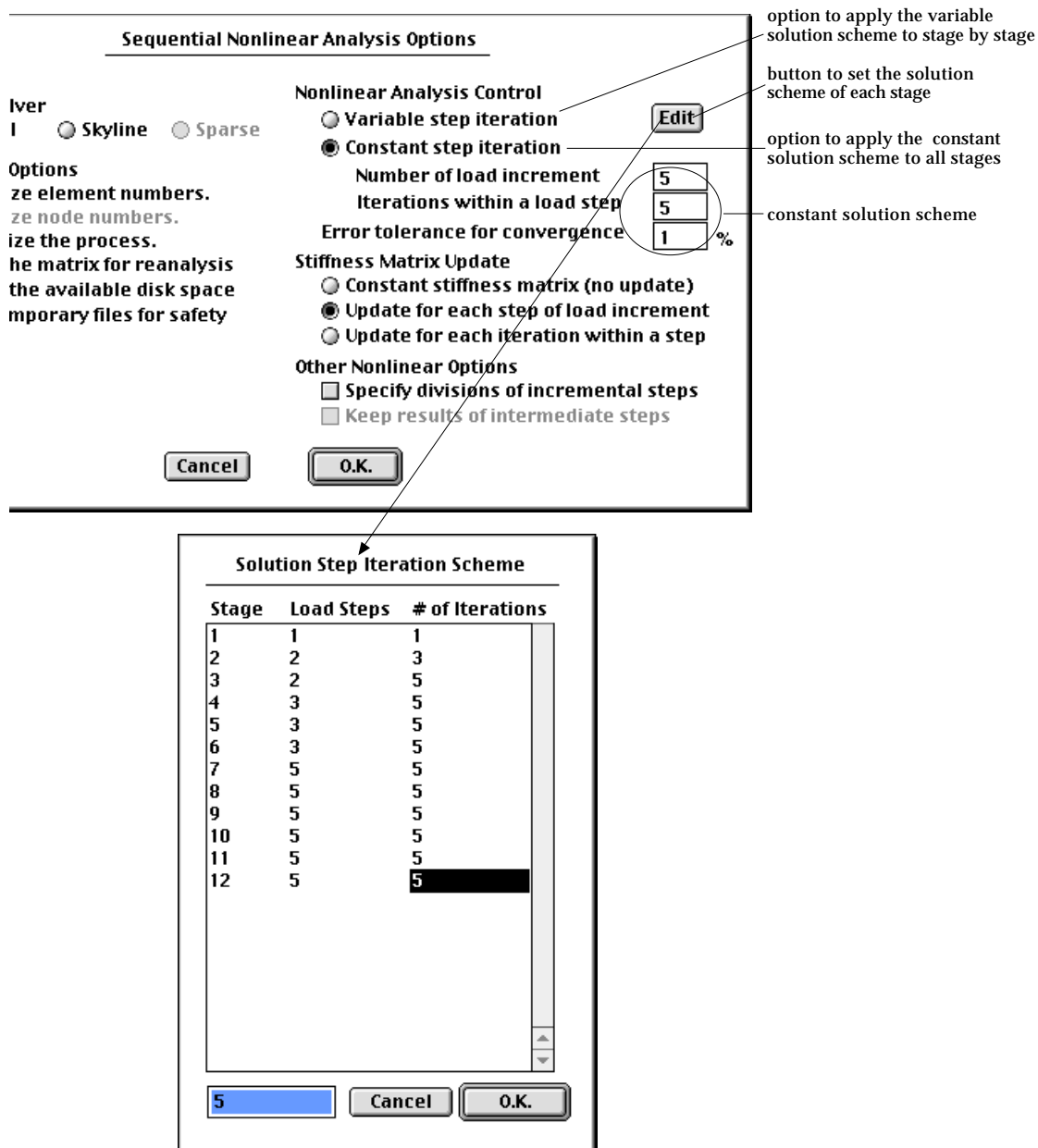
- "Keep results of intermediate steps" : If this item is checked, the solution data obtained at every incremental step is saved, and can be retrieved for later use.

### ■ Setting analysis options for sequentially staged modeling

As for linear analysis, the same dialog is used for both non-staged and staged modeling. In the case of nonlinear analysis, dialog for sequentially staged modeling has a few more items than the dialog for non-stage modeling. They are related to incremental and iterative solution scheme for each stage.

In order to apply constant number of incremental steps and iterations, turn on "Constant step iteration" radio button in the dialog and then insert the number of

incremental steps and iterations in the editable text box. In order to differentiate the incremental and iterative scheme from stage to stage, turn on "Variable step iteration" ratio button. And click **Edit** button. Then, "Solution Step Iteration Scheme" dialog appears. You may set the number of load steps and the number of iterations using this dialog. The dialog displays as many rows as the number of stages. At the beginning, each row is assigned with equal number of incremental steps and equal number of iterations. Set new values by editing the existing ones, and click **O.K.** button to complete the setting.



## Processing of heat conduction and seepage analysis

Processing of heat conduction analysis is basically the same as that of structural analysis. But a seepage analysis should go through iterative processes of determining the phreatic flow surface which is not necessary in a structural or a heat conduction analysis.

### ■ Setting analysis options for heat conduction analysis

A steady state analysis of a heat conduction problem requires only one cycle of equation assembly and solution process which is identical to the processing of a linear static analysis of a structural problem. This type of processing is described already in "Processing of structural analysis" section of this chapter, and is not repeated here.

### ■ Setting analysis options for seepage analysis

Although both the seepage analysis and the heat conduction analysis are based on the same governing equations, the processing of seepage analysis is different from that of the latter because it requires an iterative procedure determining the phreatic flow surface.

**Seepage Analysis Options**

☒ Skyline   ☐ Sparse  
 tions  
 element numbers.  
 node numbers.  
 the process.  
 matrix for reanalysis  
 e available disk space  
 orary files for safety

Datum (Coordinate of zero height)  datum level  $H_0$   
 Unit weight of fluid  unit weight of fluid  
 Number of time steps  number of time steps (only for transient analysis)  
 Time step size  sec length of a time step (only for transient analysis)  
 Number of iterations within a step  maximum number of iterations  
 Error tolerance for convergence  % convergence criterion for iteration (error percentage in head)  
 Open head increment  maximum head increment within an iteration cycle  
 Criterion for open head modification  
☒ By elevation   ☐ By pressure criterion for modifying the open head in each iteration

Other Analysis Options  
☐ Specify unequally spaced time steps  
☐ Keep results of intermediate steps

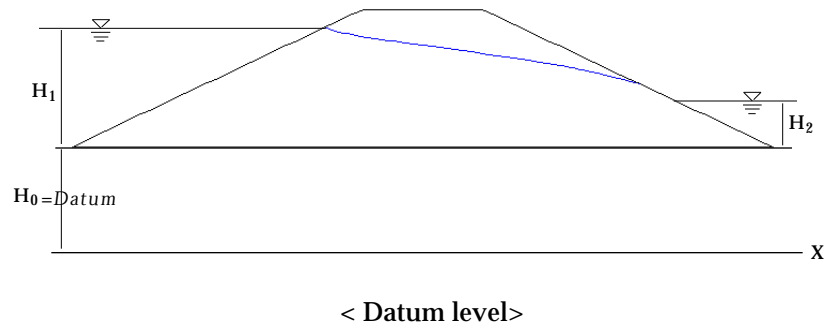
direction of elevation  
 (used only for 3-D models)  
☒ Y   ☐ Z

The editable text items of the dialog requires inputting the following values:

- Datum (Coordinate of zero height) : datum elevation. The hydraulic head is set to 0 at this level.





- Unit weight of fluid : The unit weight of fluid is used in converting the head to fluid pressure.
- Number of time steps : the number of time steps involved in the transient analysis. This value is ignored in a steady state analysis.
- Time step size : the length of time between two consecutive steps. This value is ignored in a steady state analysis.
- Number of iteration within a step : The phreatic surface is determined by iterative computation. This value specifies the maximum number of iterations determining phreatic surface in a steady state analysis. This value is also applied to a transient analysis as the number of iterations within each time step.
- Error tolerance for convergence : Another criterion for finishing the iterative process is base on the maximum difference of phreatic surface elevations between the current and the last iterations. One of this and the above criteria is met, the iterative processing is terminated.
- Open head increment : The limit of increment in modifying the open head in a iteration cycle.
- Criterion for open head modification : The open head is incremented on the basis of this criterion.
  - "By elevation": If this option is on, the node with lowest elevation is searched along the open head boundary, and the phreatic head is set to the head of node. If more than one node has the lowest elevation, only the node with the maximum pressure will be used to set the head
  - "By pressure": If this option is on, the node with maximum positive pressure is searched along the open head boundary, and the phreatic head is set to the head of node.

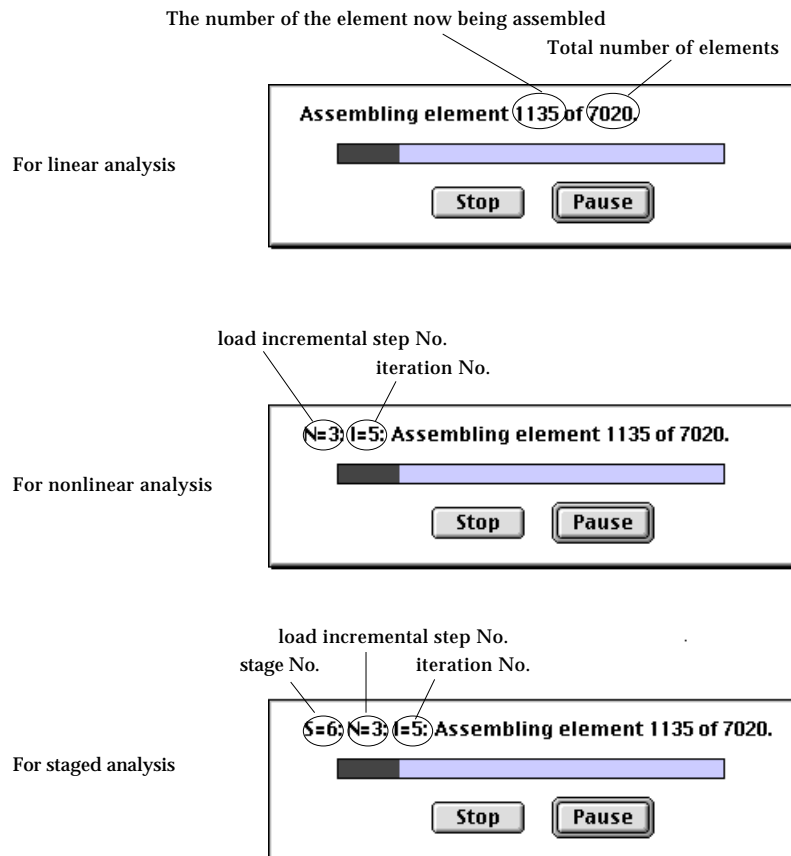
## Processing stages

Processing is divided into a few stages, and a message dialog shows the progress of each stage. Processing can be interrupted or resumed in the middle, if necessary. If there is any problem in solving the equations, appropriate message is displayed and processing aborts.

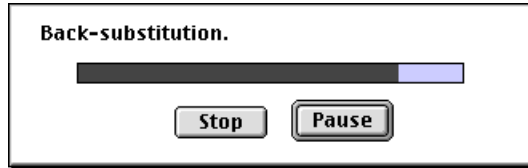
### ■ Progress of processing

While the processing is going on, its progress is indicated on a modal dialog as shown below. The progress is displayed in 3 stages. If the frontal solver is adopted, “Assembling element”, “Back substitution”, and “Stress recovery and smoothing” caption is posted on the dialog to indicate the processing stages. If skyline solver is used, “Matrix decomposition” is posted at the second stage.

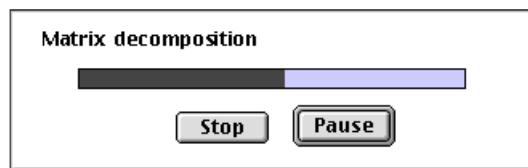
During the first stage, the element matrices are assembled. The dialog shows the total number of elements as well as the number of elements assembled so far. For nonlinear analysis or sequentially stage analysis, the stage No., the load incremental step No. and the iteration No. are also displayed in the dialog. In case of frontal solver, not only assembly but also decomposition of the equations are going on at this stage.



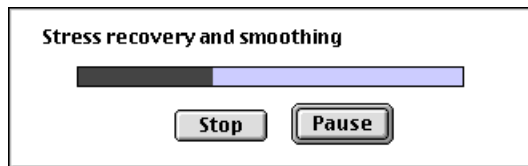
In the next stage, the decomposed equations back substituted, if the frontal solver is used. The progress bar starts from the beginning again.



If skyline solver is used, the system equations are decomposed at this stage. So, the caption of the dialog is different.



In the last stage, the strains (or gradient) and the stresses are computed at integration points of each element, and their values are smoothed at each node. The dialog appears as follows, and the progress bar starts from the beginning again.



If the execution of processing is successfully completed, a beep sounds and the following message dialog appears.



### ■ Interrupting the processing

While the processing is going on, execution can be interrupted at any time either permanently or temporarily by clicking **Stop** or **Pause** button. If you press **Stop** button, execution aborts and the following message box appears.



If you click **Pause** button of the progress dialog, processing does not abort but is temporarily suspended so that you may do other operations while the following dialog stays on the screen.



Execution is resumed and the status dialog moves to the front when you click **Resume** button of the dialog. Processing may be aborted at this stage by pressing **Stop** button.

### ■ Abnormal termination of the processing

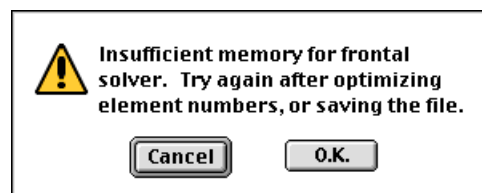
Processing may be terminated abnormally in the middle of execution due to one of the following reasons.

- **Matrix files for reanalysis are not found or are mismatching:** If the solution stage is set as "Reanalysis" but necessary files for reanalysis do not exist or do not match the data file, the processing cannot be executed. So, the operation will abort with the following message box.



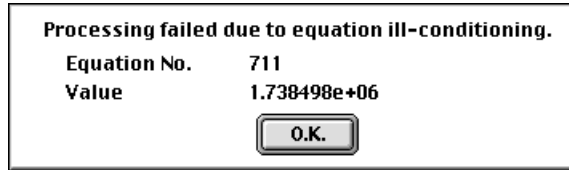
*The matrix files for reanalysis should reside in the same folder (or directory) as the data file. And their name and extension should be compatible with that of the data file. If the name of the data file is "mydata", for example, "mydata.mtx" and "mydata.fro" should exist in the same folder (or directory).*

- **Insufficient memory space:** If "Skyline" is chosen as solver option, but the CPU memory space is insufficient, then the software will let you to allow automatic switch to "Frontal" as already explained in the previous section. In case "Frontal" is chosen as solver option and the CPU memory space is insufficient, the processing aborts with the following message box.



The required memory space for “Frontal” solver may be reduced by optimizing element numbering. If a memory insufficiency problem persists even after element number optimization, more computer memory should be secured for VisualFEA.

- **Equation ill-conditioning:** If the assembled system equations are ill-conditioned, numerical difficulty arises in solving the equations, and thus the processing aborts in the middle with the following message.

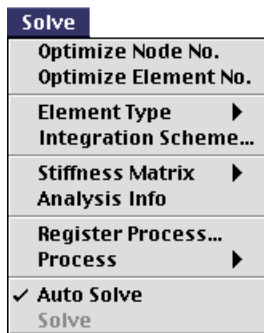


The possible causes of equation ill-conditioning are as follows.

- **Poor element shape or element connectivity :** The element stiffness matrix may have numerical singularity due to unacceptably distorted shape or due to improper connectivity. If such is the case, improve the element shape and connectivity by carefully regenerating the finite element mesh again.
- **Insufficient constraints :** The system equations may have rank deficiency due to insufficient constraints or boundary conditions. In this case, check the boundary conditions and add more constraints if necessary.
- **Irrelevant element properties:** Element stiffness matrices may have been spoiled by irrelevant values of element properties. To avoid such problem, check the contents of element property sets.
- **insufficient integration order:** The stiffness matrix may have spurious zero energy modes due to insufficient integration order. In order to remove spurious zero energy modes, adopt an integration scheme of higher order.

## Interactive real time processing

VisualFEA has the capability of interactive real time processing for truss and frame analysis. Interactive real time processing means that the cycle of data modification, subsequent processing, and graphical visualization of the analysis results is formed in real time. Thus, you can see the changing response as you add or modify the data.

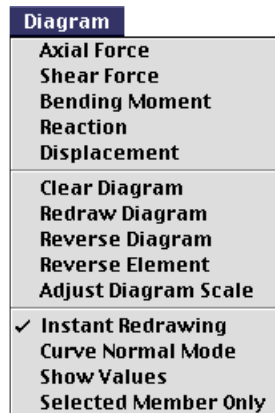


For analysis types of 2-D truss, 3-D truss, 2-D frame and 3-D frame, interactive real time processing is the default mode. “Auto Solve” item of **Solve** menu is enabled only for these analysis types. The interactive real time analysis mode can be turned on or off by checking the menu item. If the mode is turned off, the normal procedure of processing applies, as described in the previous sections.

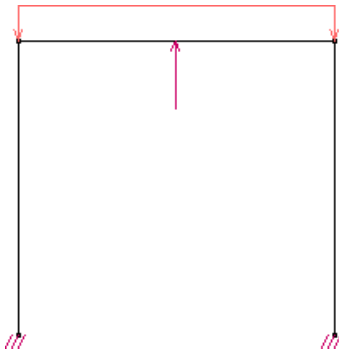
If the mode is on, you don’t have to select “Solve” item from **Solve** menu. Instead, you have to designate the data item to display from **Diagram** menu. Whenever you add or modify the data, the processing is executed automatically, and the display of the analysis results is updated immediately. The progress bar does not appear during processing.

Furthermore, if “Instant Redrawing” item of **Diagram** menu is checked, the diagrams representing the analysis results are automatically updated immediately after any change in the modeling data. Modification in geometries, attributes, boundary conditions, or load conditions of the model is reflected in the currently displayed diagram instantly.

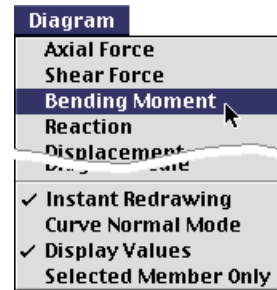
An example of interactive real time analysis is shown in the following figure. The example shows that the bending moment diagram is displayed directly by selecting the corresponding menu item. There is no need to invoke explicitly the processing stage. The diagram is updated at the moment the load condition is altered. Likewise, the diagram will be changed as you modify the boundary conditions or element properties. Thus, the responses to varying external effects or attributes can be examined interactively.



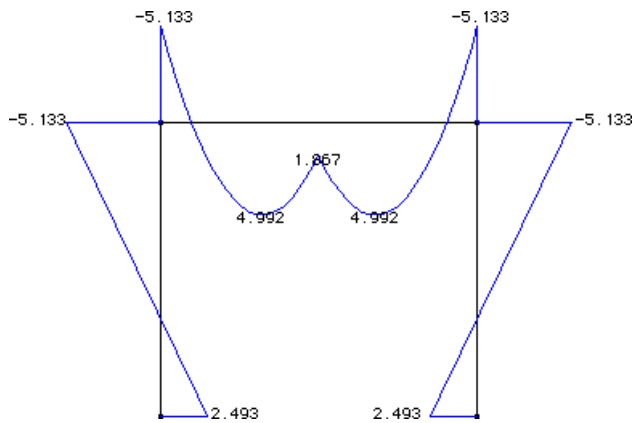
On the other hand, interactive real time analysis may sometimes hamper the responsiveness of the software, when the size of the problem is too large to get enough speed in processing one cycle. In such circumstances, you may suppress the options, by unchecking “Instant Redrawing” of **Diagram** menu. In addition, the progress bar can be shown while processing goes on, if “Auto Solve” item of **Solve** menu is unchecked as well.



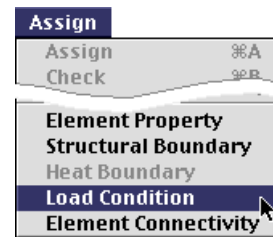
1. Model the structure, and assign attributes, boundary conditions and load conditions.



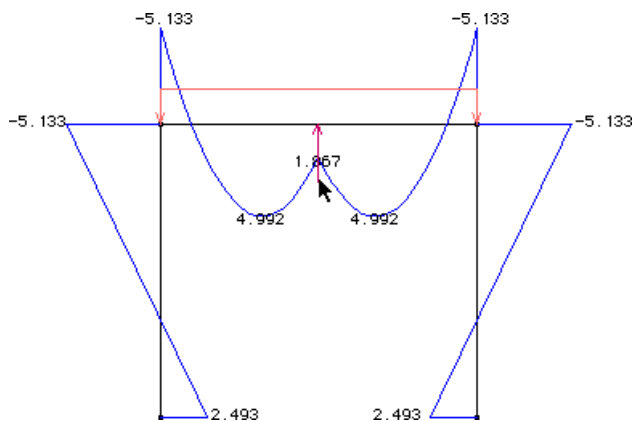
2. Choose "Bending Moment" menu item. Make sure that "Instant Redrawing" item is also checked.



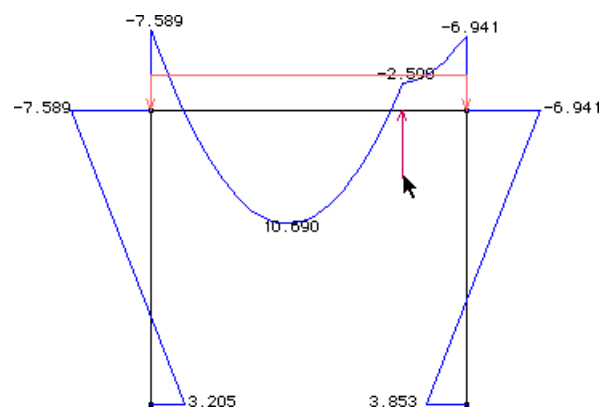
3. Bending moment diagram is displayed.



4. Choose "Load Condition" menu item.



5. Select the point load.



6. Drag the point load to the right. Then, the bending moment diagram changes subsequently.

< Example of interactive real time processing >

## Other Functions Related to Processing

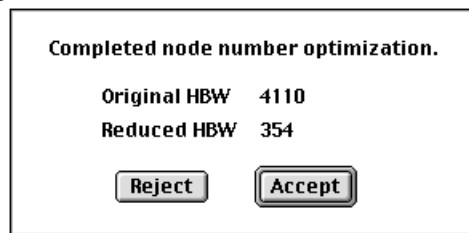
The computational efficiency of processing depends on a number of factors like node numbering, element numbering, integration scheme, and so on. They can be manipulated to improve the efficiency.

### Optimizing node numbering

If “Skyline” solver is used, the required memory and the computing time is directly related to the band width of the system equations. The band width is determined by the node numbering. Therefore, node numbering is very important. However, nodes are numbered initially in the order of their creation during the modeling stage. It is desirable to renumber the nodes before processing stage so that the band width is minimized. This procedure is called node number optimization.

Node number optimization can be dictated as an optional item to be done before processing as explained in the previous section. This option, however, may cause redundant repetition of the operation, if the same modeling data are used for multiple analysis. This redundant operation may be avoided by doing the node number optimization once and executing the processing multiple times without the optimization option.

Node number optimization can be invoked by choosing “Optimize Node Number” item from the menu. A dialog shows the intermediate state of progressive band width reduction and the final half band width (HBW) after the completion of optimization. If you click **Accept** button, the nodes will be renumbered as optimized. If you click **Reject** button, the optimization is ignored and the old node numbering will be retained. Save the file in order to keep this renumbering for future processing.



*It should be noted that the optimization does not mean absolute minimization of the band width, but means relative reduction of the band width. There may exist a node numbering with smaller band width than the optimized one. However, it is too time consuming or not feasible to find the numbering with this absolute minimum band width.*

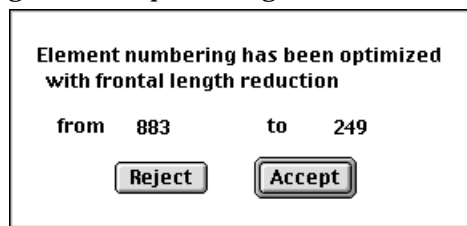


## Optimizing element numbering

If “Frontal” solver is used, the required memory and computing time are directly related to the critical frontal length of the system equations. The critical frontal length is determined by the element numbering. Similarly to node number optimization, it is desirable to renumber the elements before processing stage so that the critical frontal length becomes the minimum. This procedure is called element number optimization.

Element number optimization can be dictated as an optional item to be done before processing as explained in the previous section. This option, however, may cause redundant repetition of the operation, if the same modeling data are used for multiple analysis. This redundant operation may be avoided by doing the element number optimization once and executing the processing multiple times without optimization option.

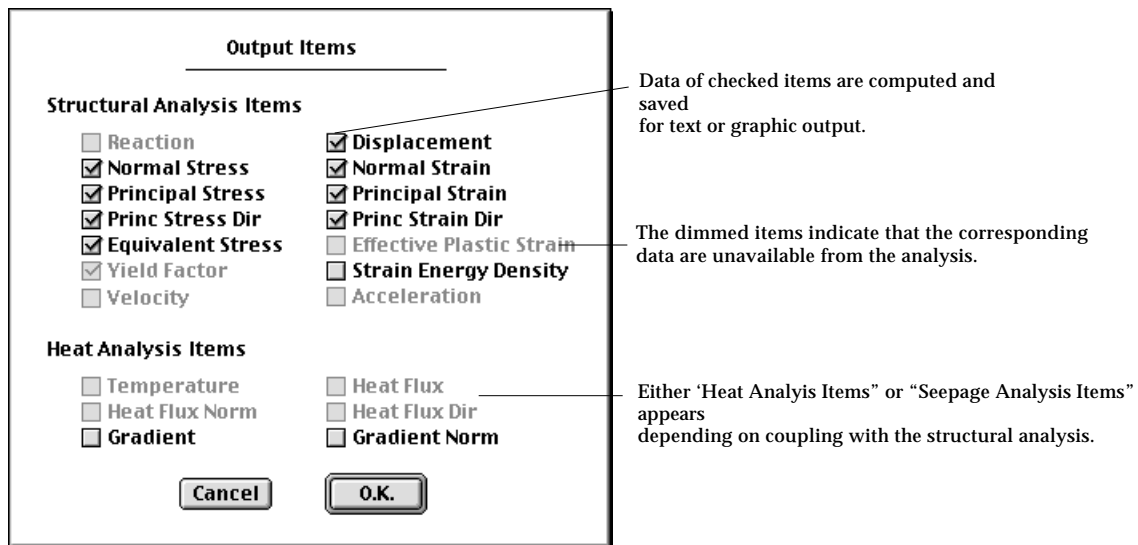
Element number optimization can be invoked by choosing “Optimize Element Number” item from menu. A dialog shows the intermediate state of progressive reduction of critical frontal length and the final critical frontal length after completion of optimization. If you click **Accept** button, the elements will be renumbered as optimized. If you click **Reject** button, the optimization is ignored and the old element numbering will be retained. Save the file in order to keep this renumbering for future processing.



*It should be noted that the optimization does not mean absolute minimization of the critical frontal length, but means relative reduction of the critical frontal length. There may exist an element numbering with smaller critical frontal length than the optimized one. However, it is too time consuming or not feasible to find the numbering with this absolute minimum critical frontal length*

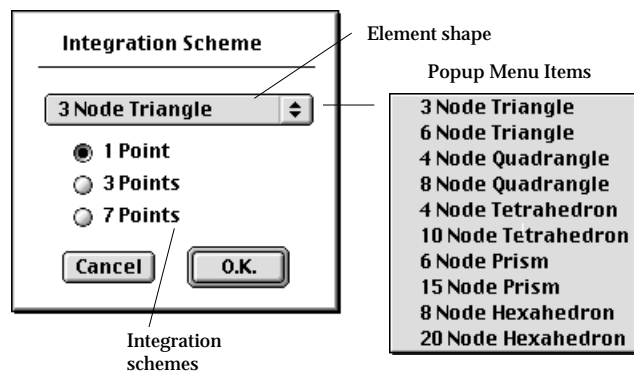
## Setting output items

- output items: Only the checked items are computed during the processing and accordingly are accessible for postprocessing or text output. Depending on the subject of analysis, either structural analysis related items or heat transfer related items are enabled and the others are disabled.



## Specifying integration scheme

In order to change the integration scheme, select "Integration Scheme..." item from **Solve** menu. Then "Integration Scheme" dialog box appears as shown below. Select the element shape (and order) from the popup menu of the dialog.

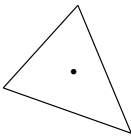
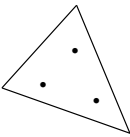
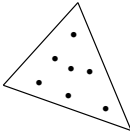
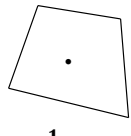
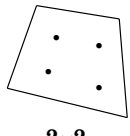
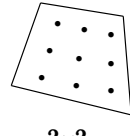
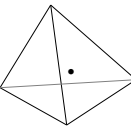
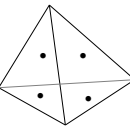
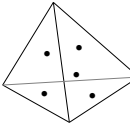
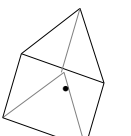
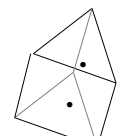
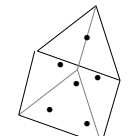
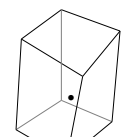
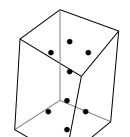
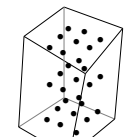


Then, alternative integration schemes for the element shape are displayed as radio button items. The radio button corresponding to the current setting is marked. The integration scheme can be changed simply by turning on the radio button of

the desired scheme. After changing the schemes of the relevant element shapes, click  button. The integration scheme of each element shape is applied in computing the element stiffness matrix of the respective element shape.

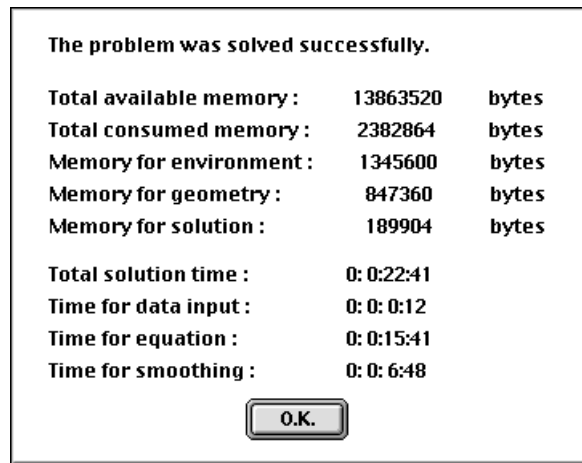
VisualFEA uses numerical integration(Gauss quadrature) in evaluating the stiffness matrix. The computing time, accuracy and stability of the system equations depend greatly on the integration scheme. There are default schemes preset for various element shapes as shown in the following table. These integration schemes can be altered if desired.

<Integration scheme>

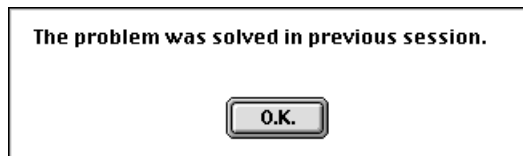
Element Shape	Integration Schemes	No. of Nodes	Default Scheme
Triangle	 1	3	1
	 3	6	3
	 7		
Quadrangle	 1	4	1
	 2×2	8	2×2
	 3×3		
Tetrahedron	 1	4	1
	 4	10	4
	 5		
Prism	 1	6	1
	 2×1	15	2×3
	 2×3		
Hexahedron	 1	8	2×2×2
	 2×2×2	20	2×2×2
	 3×3×3		

## Displaying analysis information

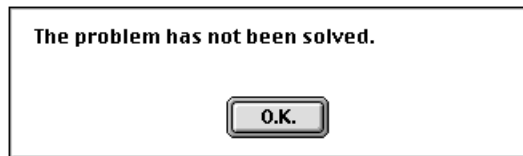
The state of analysis and related information is displayed as exemplified below by selecting “Analysis Info” item from **Solve** menu. The information includes the time and memory space required to solve the problem.



The above information is displayed only in case the problem was successfully solved in the current session. If it was solved in the previous session, the following message will be displayed.



If the problem has not gone through processing, the following message will be shown.



If the processing was canceled in the middle, the following message will be displayed .



# **Chapter 7**

## **Postprocessing of Continuum Analysis**



## Chapter 7 Postprocessing of Continuum Analysis

Various kinds of data are generated as the result of finite element analysis. They are displacements, stresses, strains, principal stresses and their directions, etc. in case of structural analysis. Temperatures, heat fluxes, heat flow directions and other data are obtained from heat transfer analysis. All of these analysis results are given in the form of numerical data. Their volume is huge and their content is very complex in most cases. Accordingly, it is extremely difficult to grasp the analysis results from these output data.

Postprocessing is the stage to process the analysis results and visualize them graphically so that their contents can easily be interpreted or understood. There are a number of functions related to visualizing data. They create two types of images. One is surface image type and the other is diagram type. The former type is applied for data in surface or volume space such as plane elasticity, plate, shell, or 3-D solid analysis. And the latter is applied to data on line or curve space such as truss or frame analysis. Thus, one of the two different menu titles, **Postpro** or **Diagram**, appears on the menu bar depending on the type of analysis. **Postpro** menu appears when the analysis type is one of the following:

- Plane stress
- Plane strain
- Axisymmetric
- Plate bending
- Shell
- 3D solid
- Plane heat
- Axisymmetric heat
- 3D volume heat
- Plane seepage
- Axisymmetric seepage
- 3D volume seepage

And, **Diagram** menu appears when the analysis type is one of the following:

- 2-D truss
- 3-D truss
- 2-D frame
- 3-D frame

Postprocessing for trusses and frames is detailed in Chapter 8. Only postprocessing of continuum analysis is described in this chapter.

Whenever the analysis type is altered by “Project Setup”, the menu title is changed to the relevant one. Both **Postpro** and **Diagram** menus appear on the menu bar, if

truss or frame members are included within an analysis of continuum structures. Each of the two menus has items appropriate for the given analysis type.

For analysis types other than truss and frame, the data distribution is expressed for points on surfaces or in 3-D volumes. These are the cases where **Postpro** menu appears on the menu bar, as described in the previous page.

The data obtained from continuum analysis can be divided into two types. One is scalar data, and the other is vector data. Scalar data have information of magnitude only. The following functions are provided as items under **Postpro** menu for visualization of scalar data:

- displaying contour image
- displaying iso-surface image
- curve plotting
- surface plotting

Vector data have information of magnitude and spatial direction. There are the following functions under **Postpro** menu for visualization of scalar data:

- displaying vector image
- displaying deformed shape
- nodal resultant

There are also other auxiliary functions related with postprocessing. VisualFEA has the capability of image handling and animation. Related commands are provided under **Postpro** menu.

<b>Postpro</b>	
Contour...	Display contour image of the data
Iso-surface...	Display isosurface image of the data
Curve Plot...	Start curve plotting mode
Surface Plot...	Display the data by surface plotting image
Vector...	Display vector image of the data
Deformed Shape	Display the deformed shape of the model
Seepage	Display the seepage analysis results
Nodal Values	Display the numerical values of the nodal resultants
Multi-step View	Start multi-step view mode
Dynamic Response	Start displaying dynamic response
Mode Shape	Start displaying dynamic mode shapes
Open Image...	Open an image file and display the image
Save Image...	Save the screen image in a file
Capture Image	Capture the screen image
Play Animation...	Open an animation file and play the animation
Make Animation...	Create an animation and save it in a file
Show Scale	Show or hide scale bar



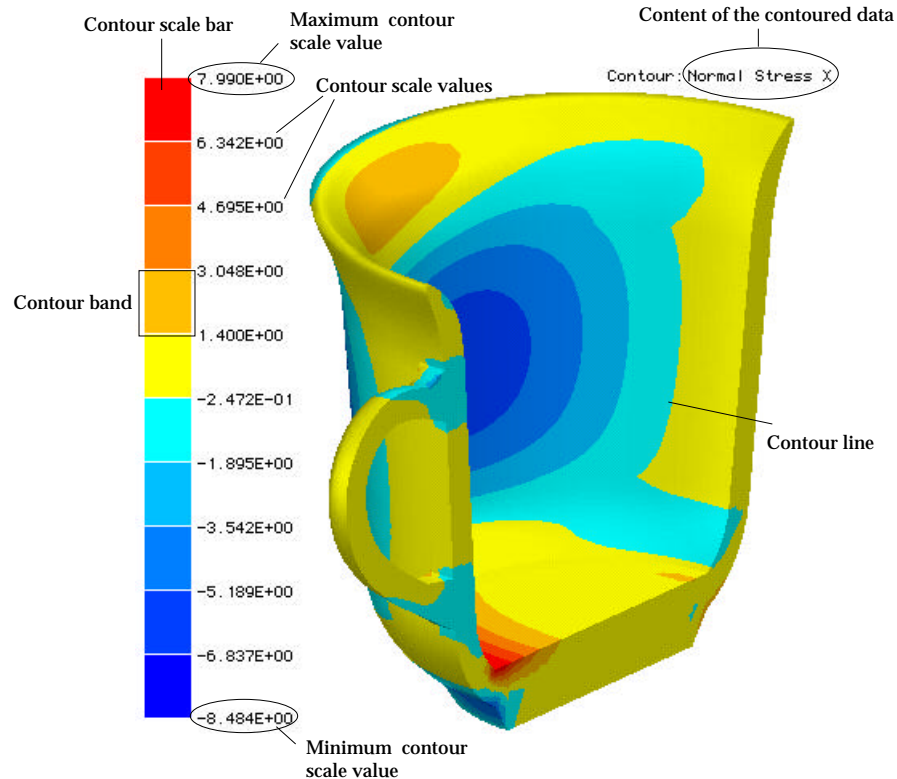
## Visualizing Scalar Data by Contours



Contouring data is the most frequently used method of visualizing finite element analysis results. This method is appropriate for visualizing the distribution of scalar data, such as stress, strain or temperature, on planes or surfaces. Each contour represents a continuous curve on which the value of the scalar data is uniform. Thus, gradual variation of data on the planes or surfaces is scaled by a number of contours.

In order to improve the visual effect, VisualFEA renders colored contour bands instead of contour lines. The entire span of the scalar value is divided into a number of sub-spans. Each sub-span is represented by a contour band. The numerical values represented by border lines of contour bands are indicated in the contour scale bar which appears together with a contour image.

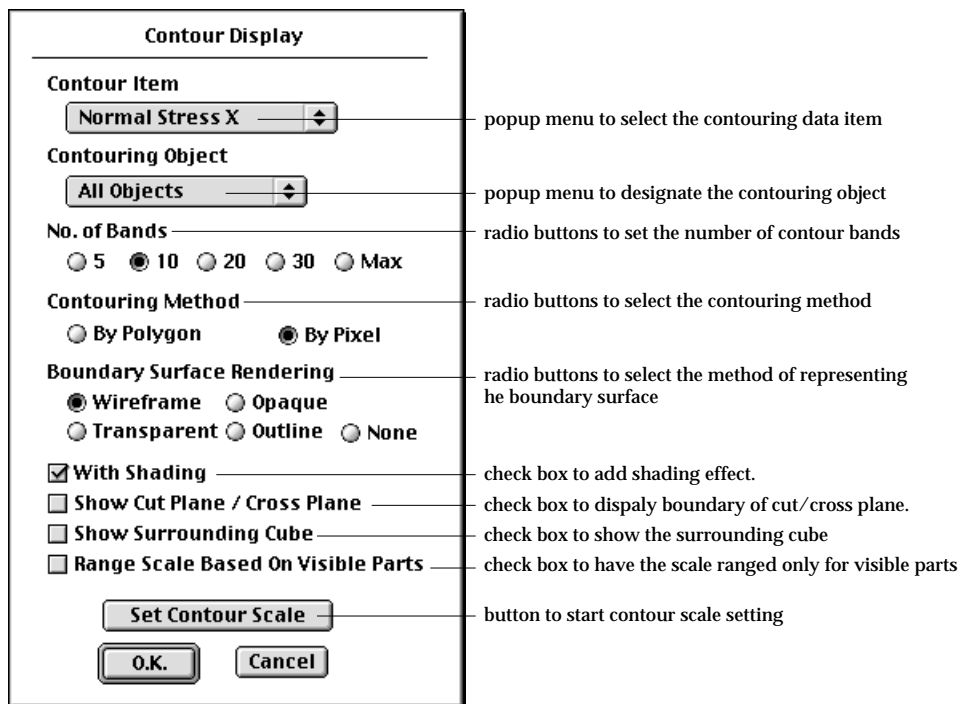
One weakness of contouring is its incapability of visualizing the data distribution inside of 3 dimensional volume. Auxiliary planes like cut plane, cross planes and parallel planes are used as volume visualization aids to make up for this limitation.



< Contour image >

## Setting contouring options

In order to get contour image of a data set, first select “Contour...” item from **Postpro** menu. Then, “Contour Display” dialog appears on the screen. There are a number of items in this dialog. Each item has a default setting. Change the setting if necessary. You may also specify the scale of contour bands. Click **O.K.** button if every item is set as desired. Then, the contour image will be displayed with a scale bar which indicates the ranges of data values represented by contour bands.



<“Contour Display” dialog >

### ■ Selecting the data item

The popup menu in “Contour Display” dialog has the list of data items which can be displayed by contouring. One of the items should be selected from this popup menu. The data associated with the currently selected popup menu item is rendered in contour. The first item is always selected initially when the “Contour Display” dialog is first opened. But the item used for last contouring will be selected automatically when the dialog is opened next.

The popup menu items vary depending on the analysis type. Only those output items appropriate for contouring appear on the popup menu. They are shown below.

All the items are not always shown in the popup menu. If you exclude any of the output items in “Analysis Options” dialog described in Chapter 6, the




corresponding item(s) will not be shown.

If you are using an external solver, the menu items may be different from those shown below, because the popup menu items can be customized by the external

<table><tr><td>Normal Stress X</td></tr><tr><td>Normal Stress Y</td></tr><tr><td>Shear Stress XY</td></tr></table> <table><tr><td>Normal Strain X</td></tr><tr><td>Normal Strain Y</td></tr><tr><td>Shear Strain XY</td></tr></table> <table><tr><td>Princ Stress 1</td></tr><tr><td>Princ Stress 2</td></tr><tr><td>Von Mises Stress</td></tr></table> <table><tr><td>Princ Strain 1</td></tr><tr><td>Princ Strain 2</td></tr><tr><td>Von Mises Strain</td></tr></table> <table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Norm</td></tr></table>	Normal Stress X	Normal Stress Y	Shear Stress XY	Normal Strain X	Normal Strain Y	Shear Strain XY	Princ Stress 1	Princ Stress 2	Von Mises Stress	Princ Strain 1	Princ Strain 2	Von Mises Strain	Displacement X	Displacement Y	Displacement Norm	<table><tr><td>Moment X</td></tr><tr><td>Moment Y</td></tr><tr><td>Moment XY</td></tr></table> <table><tr><td>Curvature X</td></tr><tr><td>Curvature Y</td></tr><tr><td>Curvature XY</td></tr></table> <table><tr><td>Princ Moment 1</td></tr><tr><td>Princ Moment 2</td></tr></table> <table><tr><td>Princ Curvature 1</td></tr><tr><td>Princ Curvature 2</td></tr></table> <table><tr><td>Displacement Z</td></tr><tr><td>Rotation X</td></tr><tr><td>Rotation Y</td></tr><tr><td>Displacement Norm</td></tr></table>	Moment X	Moment Y	Moment XY	Curvature X	Curvature Y	Curvature XY	Princ Moment 1	Princ Moment 2	Princ Curvature 1	Princ Curvature 2	Displacement Z	Rotation X	Rotation Y	Displacement Norm	<table><tr><td>Membrane Stress X</td></tr><tr><td>Membrane Stress Y</td></tr><tr><td>Membrane Stress XY</td></tr><tr><td>Membrane Stress YZ</td></tr><tr><td>Membrane Stress ZX</td></tr><tr><td>Moment X</td></tr><tr><td>Moment Y</td></tr><tr><td>Moment XY</td></tr></table> <table><tr><td>Membrane Strain X</td></tr><tr><td>Membrane Strain Y</td></tr><tr><td>Membrane Strain XY</td></tr><tr><td>Membrane Strain YZ</td></tr><tr><td>Membrane Strain ZX</td></tr><tr><td>Curvature X</td></tr><tr><td>Curvature Y</td></tr><tr><td>Curvature XY</td></tr></table> <table><tr><td>Princ Stress 1</td></tr><tr><td>Princ Stress 2</td></tr><tr><td>Princ Stress 3</td></tr><tr><td>Princ Moment 1</td></tr><tr><td>Princ Moment 2</td></tr><tr><td>Von Mises Stress</td></tr></table> <table><tr><td>Princ Strain 1</td></tr><tr><td>Princ Strain 2</td></tr><tr><td>Princ Strain 3</td></tr><tr><td>Princ Curvature 1</td></tr><tr><td>Princ Curvature 2</td></tr><tr><td>Von Mises Strain</td></tr></table> <table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Z</td></tr><tr><td>Rotation X</td></tr><tr><td>Rotation Y</td></tr><tr><td>Displacement Norm</td></tr></table>	Membrane Stress X	Membrane Stress Y	Membrane Stress XY	Membrane Stress YZ	Membrane Stress ZX	Moment X	Moment Y	Moment XY	Membrane Strain X	Membrane Strain Y	Membrane Strain XY	Membrane Strain YZ	Membrane Strain ZX	Curvature X	Curvature Y	Curvature XY	Princ Stress 1	Princ Stress 2	Princ Stress 3	Princ Moment 1	Princ Moment 2	Von Mises Stress	Princ Strain 1	Princ Strain 2	Princ Strain 3	Princ Curvature 1	Princ Curvature 2	Von Mises Strain	Displacement X	Displacement Y	Displacement Z	Rotation X	Rotation Y	Displacement Norm	<table><tr><td>Normal Stress X</td></tr><tr><td>Normal Stress Y</td></tr><tr><td>Normal Stress Z</td></tr><tr><td>Shear Stress XY</td></tr><tr><td>Shear Stress YZ</td></tr><tr><td>Shear Stress ZX</td></tr></table> <table><tr><td>Normal Strain X</td></tr><tr><td>Normal Strain Y</td></tr><tr><td>Normal Strain Z</td></tr><tr><td>Shear Strain XY</td></tr><tr><td>Shear Strain YZ</td></tr><tr><td>Shear Strain ZX</td></tr></table> <table><tr><td>Princ Stress 1</td></tr><tr><td>Princ Stress 2</td></tr><tr><td>Princ Stress 3</td></tr><tr><td>Von Mises Stress</td></tr></table> <table><tr><td>Princ Strain 1</td></tr><tr><td>Princ Strain 2</td></tr><tr><td>Princ Strain 3</td></tr><tr><td>Von Mises Strain</td></tr></table> <table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Z</td></tr><tr><td>Displacement Norm</td></tr></table>	Normal Stress X	Normal Stress Y	Normal Stress Z	Shear Stress XY	Shear Stress YZ	Shear Stress ZX	Normal Strain X	Normal Strain Y	Normal Strain Z	Shear Strain XY	Shear Strain YZ	Shear Strain ZX	Princ Stress 1	Princ Stress 2	Princ Stress 3	Von Mises Stress	Princ Strain 1	Princ Strain 2	Princ Strain 3	Von Mises Strain	Displacement X	Displacement Y	Displacement Z	Displacement Norm
Normal Stress X																																																																																										
Normal Stress Y																																																																																										
Shear Stress XY																																																																																										
Normal Strain X																																																																																										
Normal Strain Y																																																																																										
Shear Strain XY																																																																																										
Princ Stress 1																																																																																										
Princ Stress 2																																																																																										
Von Mises Stress																																																																																										
Princ Strain 1																																																																																										
Princ Strain 2																																																																																										
Von Mises Strain																																																																																										
Displacement X																																																																																										
Displacement Y																																																																																										
Displacement Norm																																																																																										
Moment X																																																																																										
Moment Y																																																																																										
Moment XY																																																																																										
Curvature X																																																																																										
Curvature Y																																																																																										
Curvature XY																																																																																										
Princ Moment 1																																																																																										
Princ Moment 2																																																																																										
Princ Curvature 1																																																																																										
Princ Curvature 2																																																																																										
Displacement Z																																																																																										
Rotation X																																																																																										
Rotation Y																																																																																										
Displacement Norm																																																																																										
Membrane Stress X																																																																																										
Membrane Stress Y																																																																																										
Membrane Stress XY																																																																																										
Membrane Stress YZ																																																																																										
Membrane Stress ZX																																																																																										
Moment X																																																																																										
Moment Y																																																																																										
Moment XY																																																																																										
Membrane Strain X																																																																																										
Membrane Strain Y																																																																																										
Membrane Strain XY																																																																																										
Membrane Strain YZ																																																																																										
Membrane Strain ZX																																																																																										
Curvature X																																																																																										
Curvature Y																																																																																										
Curvature XY																																																																																										
Princ Stress 1																																																																																										
Princ Stress 2																																																																																										
Princ Stress 3																																																																																										
Princ Moment 1																																																																																										
Princ Moment 2																																																																																										
Von Mises Stress																																																																																										
Princ Strain 1																																																																																										
Princ Strain 2																																																																																										
Princ Strain 3																																																																																										
Princ Curvature 1																																																																																										
Princ Curvature 2																																																																																										
Von Mises Strain																																																																																										
Displacement X																																																																																										
Displacement Y																																																																																										
Displacement Z																																																																																										
Rotation X																																																																																										
Rotation Y																																																																																										
Displacement Norm																																																																																										
Normal Stress X																																																																																										
Normal Stress Y																																																																																										
Normal Stress Z																																																																																										
Shear Stress XY																																																																																										
Shear Stress YZ																																																																																										
Shear Stress ZX																																																																																										
Normal Strain X																																																																																										
Normal Strain Y																																																																																										
Normal Strain Z																																																																																										
Shear Strain XY																																																																																										
Shear Strain YZ																																																																																										
Shear Strain ZX																																																																																										
Princ Stress 1																																																																																										
Princ Stress 2																																																																																										
Princ Stress 3																																																																																										
Von Mises Stress																																																																																										
Princ Strain 1																																																																																										
Princ Strain 2																																																																																										
Princ Strain 3																																																																																										
Von Mises Strain																																																																																										
Displacement X																																																																																										
Displacement Y																																																																																										
Displacement Z																																																																																										
Displacement Norm																																																																																										
Plane stress/strain	Plate bending																																																																																									
<table><tr><td>Normal Stress X</td></tr><tr><td>Normal Stress Y</td></tr><tr><td>Circumferential Stre</td></tr><tr><td>Shear Stress XY</td></tr></table> <table><tr><td>Normal Strain X</td></tr><tr><td>Normal Strain Y</td></tr><tr><td>Circumferential Stra</td></tr><tr><td>Shear Strain XY</td></tr></table> <table><tr><td>Princ Stress 1</td></tr><tr><td>Princ Stress 2</td></tr><tr><td>Von Mises Stress</td></tr></table> <table><tr><td>Princ Strain 1</td></tr><tr><td>Princ Strain 2</td></tr><tr><td>Von Mises Strain</td></tr></table> <table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Norm</td></tr></table>	Normal Stress X	Normal Stress Y	Circumferential Stre	Shear Stress XY	Normal Strain X	Normal Strain Y	Circumferential Stra	Shear Strain XY	Princ Stress 1	Princ Stress 2	Von Mises Stress	Princ Strain 1	Princ Strain 2	Von Mises Strain	Displacement X	Displacement Y	Displacement Norm	<table><tr><td>Temperature</td></tr><tr><td>Heat Flux Norm</td></tr><tr><td>Flux in X Dir</td></tr><tr><td>Flux in Y Dir</td></tr></table>	Temperature	Heat Flux Norm	Flux in X Dir	Flux in Y Dir																																																																				
Normal Stress X																																																																																										
Normal Stress Y																																																																																										
Circumferential Stre																																																																																										
Shear Stress XY																																																																																										
Normal Strain X																																																																																										
Normal Strain Y																																																																																										
Circumferential Stra																																																																																										
Shear Strain XY																																																																																										
Princ Stress 1																																																																																										
Princ Stress 2																																																																																										
Von Mises Stress																																																																																										
Princ Strain 1																																																																																										
Princ Strain 2																																																																																										
Von Mises Strain																																																																																										
Displacement X																																																																																										
Displacement Y																																																																																										
Displacement Norm																																																																																										
Temperature																																																																																										
Heat Flux Norm																																																																																										
Flux in X Dir																																																																																										
Flux in Y Dir																																																																																										
Axisymmetric	Plane/axisymmetric heat																																																																																									
	<table><tr><td>Temperature</td></tr><tr><td>Heat Flux Norm</td></tr><tr><td>Flux in X Dir</td></tr><tr><td>Flux in Y Dir</td></tr><tr><td>Flux in Z Dir</td></tr></table>	Temperature	Heat Flux Norm	Flux in X Dir	Flux in Y Dir	Flux in Z Dir	Shell																																																																																			
Temperature																																																																																										
Heat Flux Norm																																																																																										
Flux in X Dir																																																																																										
Flux in Y Dir																																																																																										
Flux in Z Dir																																																																																										
	3-D volume heat																																																																																									

<Popup menu selecting the contouring data item >






## ■ Designating the contouring object

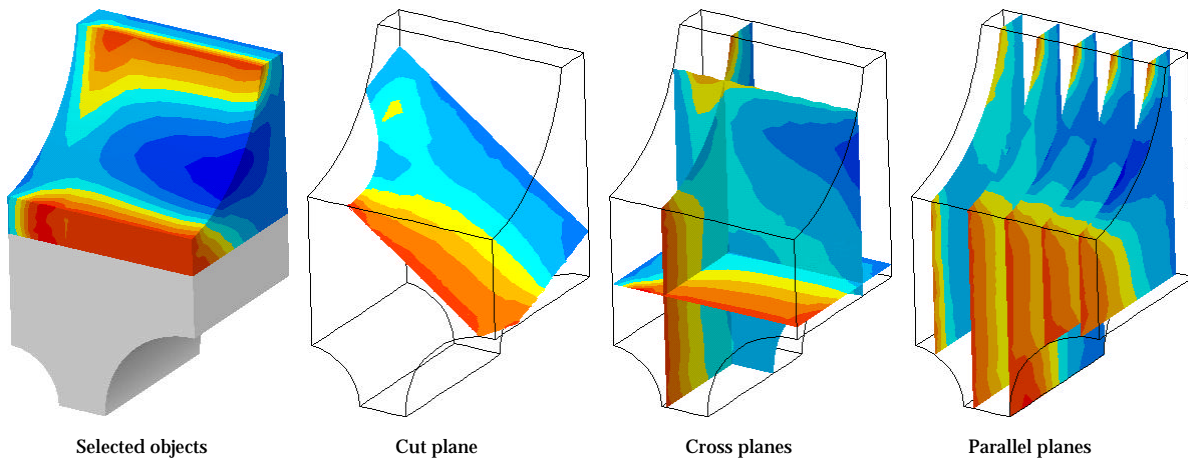
The object of a contour image is not necessarily the entire model. Contours may be drawn only on the selected part of the model, or on a pre-defined object such as a cut plane, cross planes, or parallel planes. These planes are defined using respective tools, , , and .

Contouring Object

All Objects

All Objects  
Selected Objects  
Cut Plane  
Cross Planes  
Parallel Planes

- “All Objects”: Contours are rendered on all objects, that is, on all the surfaces included in the model. In case of a 3-D solid structure or volume heat model, all boundary surfaces are used for contouring.
- “Selected Objects”: Contours are rendered only on the selected surfaces or volumes. In the case of a volume, its boundary surfaces are used for contouring. This popup menu item is active only when at least one or more objects are selected using  or  tool, prior to opening “Contour Display” dialog.
- “Cut Plane”: Contours are rendered on the cut plane. This popup menu item is active only when the cut plane is defined using  tool, prior to opening “Contour Display” dialog. The method of defining the cut plane is described in the latter part of this chapter.
- “Cross Planes”: Contours are rendered on the cross plane. This popup menu item is active only when the cross planes are defined using  tool, prior to opening “Contour Display” dialog. The method of defining the cross planes is described in the latter part of this chapter.
- “Parallel Planes”: Contours are rendered on the parallel planes. This popup menu item is active only when the parallel planes are defined using  tool, prior to opening “Contour Display” dialog. The method of defining the parallel planes is described in the latter part of this chapter.

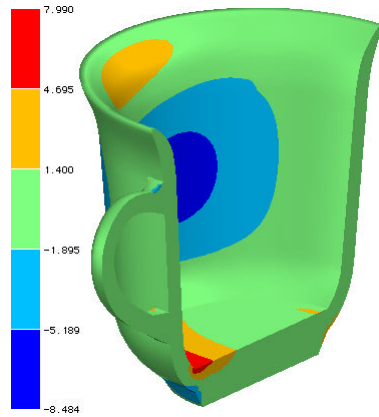


<Various contouring objects>

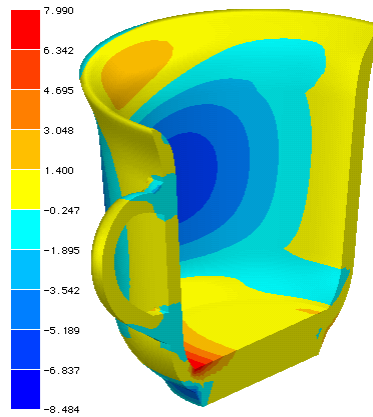
### ■ Setting the number of contour bands

The contour image has a number of contour bands, each of which represents a certain range of data value. The number of bands is initially set as 10 by default. This default setting can be changed by clicking the radio button labeled with the desired number of bands, which should be one of 5, 10, 20, 30 and Max. If you

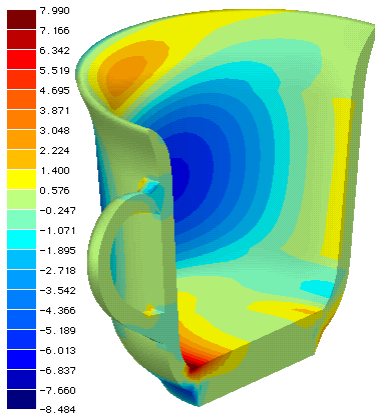
choose “Max”, the contour image will be rendered with as many bands as possible using the available colors. In this case, the color variation is so smooth that the boundaries between contour bands may not be recognizable.



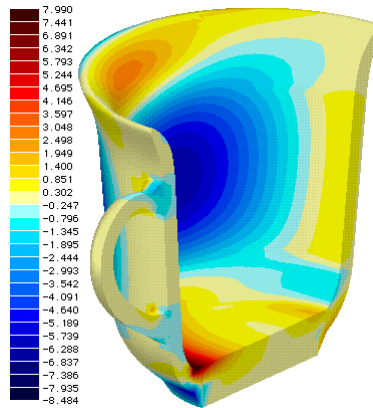
Number of contour bands = 5



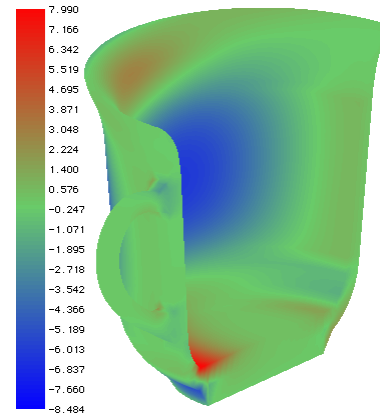
Number of contour bands = 10



Number of contour bands = 20



Number of contour bands = 30



Number of contour bands = Max

<Comparison of number of contour bands>

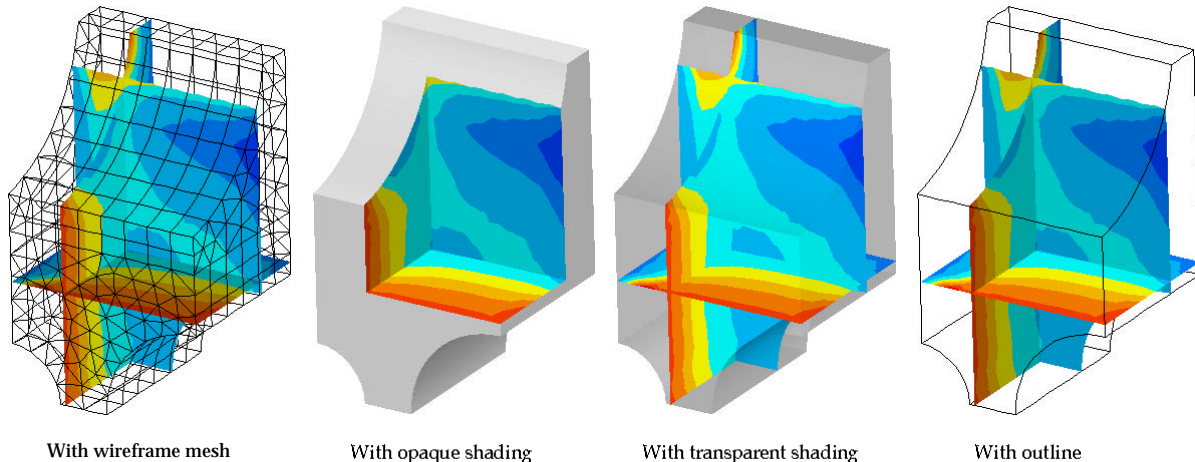
### ■ Selecting the contouring method

VisualFEA creates contour images either by polygon fill or by pixel painting. Contouring by polygon fill is faster, while pixel painting produces better quality image. You can select either one of the two using the radio buttons in the dialog. The initial default setting is “By polygon”, but can be changed by the preference setting.

### ■ Selecting the style of boundary surface rendering

Boundary surface rendering is applicable only in case the contouring object is set as a cut plane, cross planes or parallel planes defined prior to starting “Contour Display” dialog.

- “Wireframe”: The boundary surfaces are rendered in the form of wireframe. The hidden lines are removed from the wireframe rendering.
- “Opaque”: The boundary surfaces are rendered by opaque shading. The front side of the cut plane, cross planes or parallel planes are rendered together with the boundary surfaces in the back sides.
- “Transparent”: The boundary surfaces are rendered in transparency shading. The cut plane, cross planes or parallel planes are rendered as opaque objects surrounded by transparent boundary surfaces.
- “Outline”: The outlines of the boundary surfaces are extracted, and represented together with the contour image on cut plane, cross planes or parallel planes.
- “None”: The contour image is displayed without boundary surface rendering.



<Styles of boundary surface rendering>

### ■ Turning the shading effect on or off

It is desirable to add shading effect on contour images of 3 dimensional objects, in order to maintain the 3-D view after contouring. Shading effect can be turned on or off by checking or unchecking the check box labeled “With Shading”. Addition of the shading effect will require more time for contouring. For 2 dimensional planar models, the shading effect will not be applied regardless of this option.

### ■ Displaying the boundary of cut or cross planes

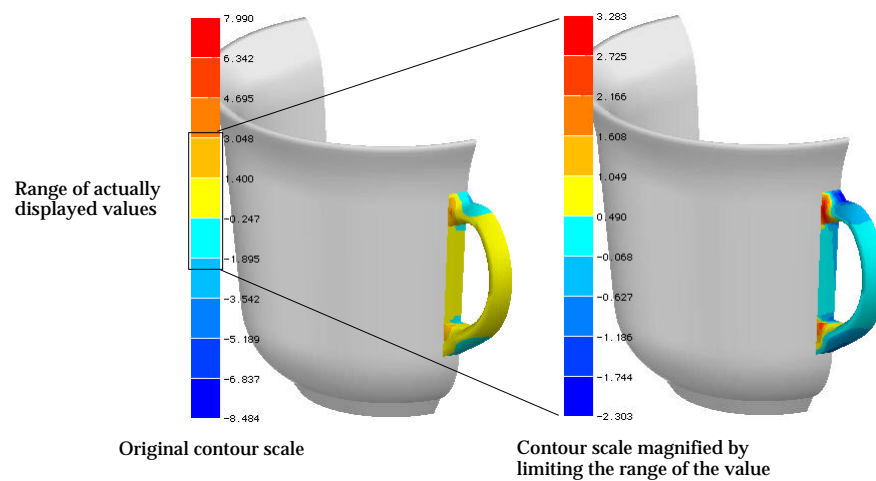
The boundary lines of a cut plane or cross planes are displayed together with the contour image, if “Show Cut Plane/Cross Plane” box is checked. This option is valid only when the contour is drawn over the cut plane or the cross planes defined prior to starting “Contour Display” dialog.

### ■ Displaying the cube surrounding the entire model

The cube surrounding the model is displayed together with the contour image, if “Show Surrounding Cube” box is checked. This option is valid only when the contour is drawn over the cut plane or the cross planes defined prior to starting “Contour Display” dialog.

### ■ Limiting the range of contour scale by actually displayed values

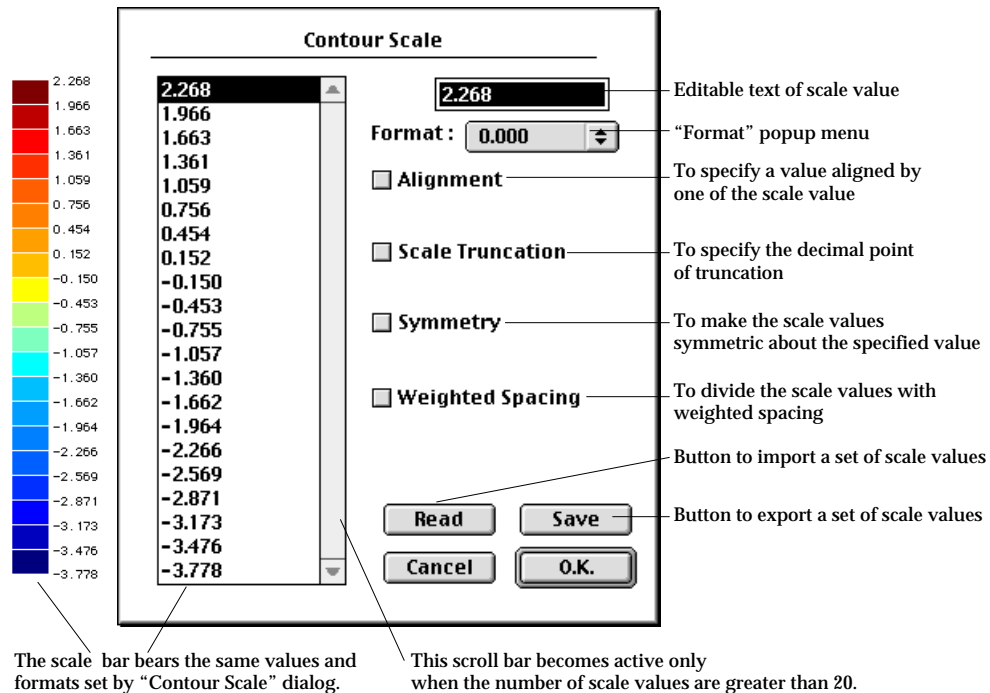
The contour image is not always drawn for the entire model. A contour image may be drawn only on the selected parts or on the visible parts. In such cases, the range of the actually contoured data may be much narrower than that of the whole data. The scale of the contour can be adjusted to the actual range of the displayed part rather than that of the entire model, by checking the box labeled “Range Scale Based On Visible Parts”.



<Limiting the range of the contour scale>

## Setting the contour scale

The contour scale is determined automatically on the basis of the number of contour bands, and the minimum and the maximum values of the data to be contoured. The gap between the maximum and minimum values is equally divided by the number of contour bands so that the intervals of adjacent two scale values are uniform. However, the scale values can be adjusted as desired. In order to adjust the contour scale, first click **Set Contour Scale** button in “Contour Display” dialog. Then, “Contour Scale” dialog will appear on the screen. The contour scale values can be modified using this dialog.



<Scale bar and “Contour Scale” dialog >

### ■ Editing the scale values

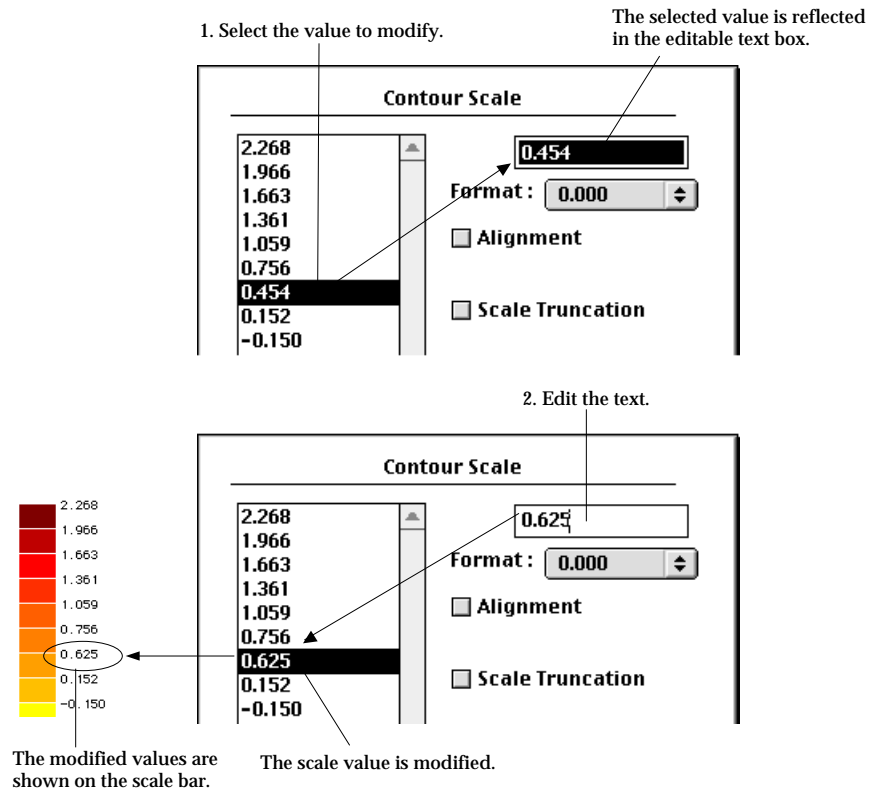
The lower and the upper limits of each contour band are represented by the values marked on the contour scale. These values may be modified by editing the corresponding text individually. The text can be edited by the following steps.

- 1) Select the scale value for modification by clicking it.  
The selected value is highlighted with reversed background, and reflected in the editable text box.
- 2) Edit the text in the editable text box.  
The scale value is modified to reflect the text editing.



3) Repeat step 1) and 2) for all scale values to modify.

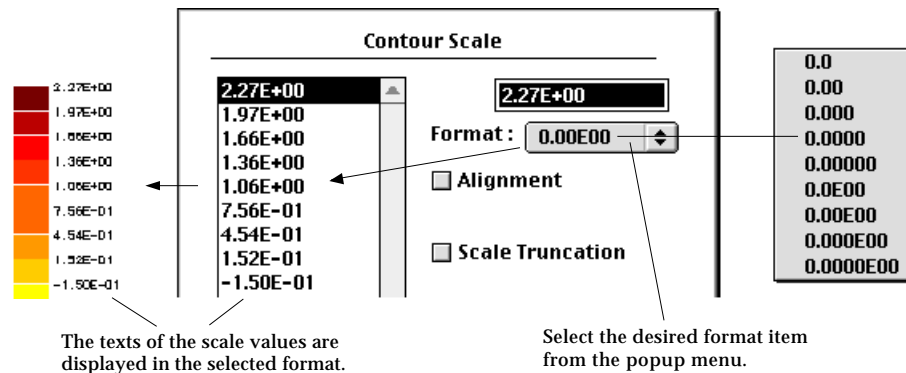
This modification mode is terminated by clicking any other buttons, check boxes or editable texts.



<Editing scale values >

### ■ Setting the format of the contour scale

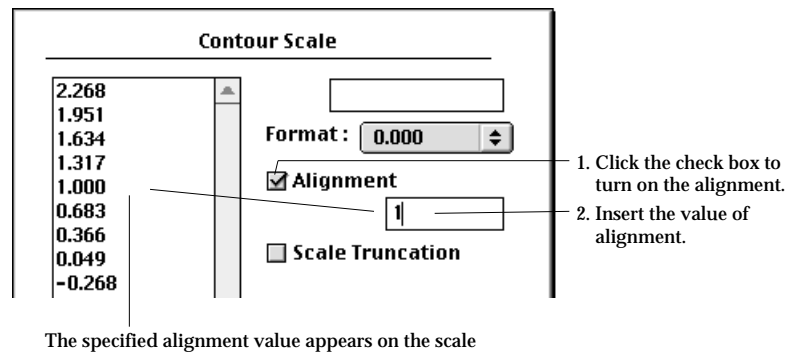
The contour scale values may be displayed in a number of different formats. The format can be set using "Format" popup menu in the dialog. The popup menu includes the various format items shown below. Select the desired item from the popup menu. Then, the text of the scale values shown in the dialog will be changed immediately. At the same time, that format will apply for future display.



<Setting the scale format >

### ■ Aligning a contour to the specified value

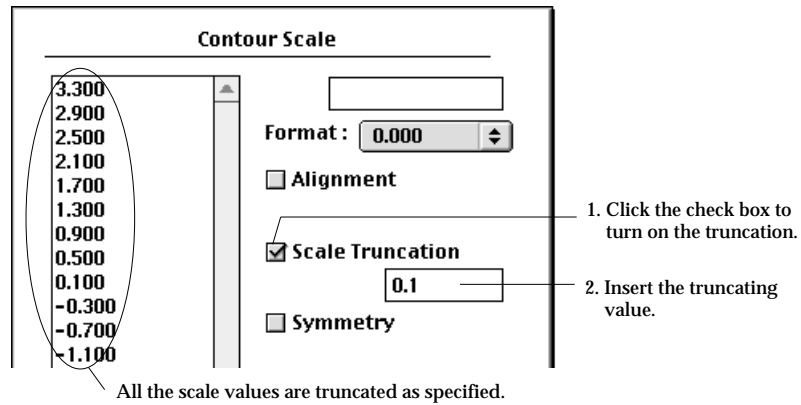
It is sometimes necessary to display a specific value by a contour line. This can be achieved by forcing one of the scale values to be aligned to the specified value. Check "Alignment" box by clicking it. Then, an editable text item appears below the check box. Insert the value specifying the alignment into this text box. The scale values are rearranged so that one of the values matches the specified value.



<Specifying the alignment value>

### ■ Truncating the scale values at the specified decimal point

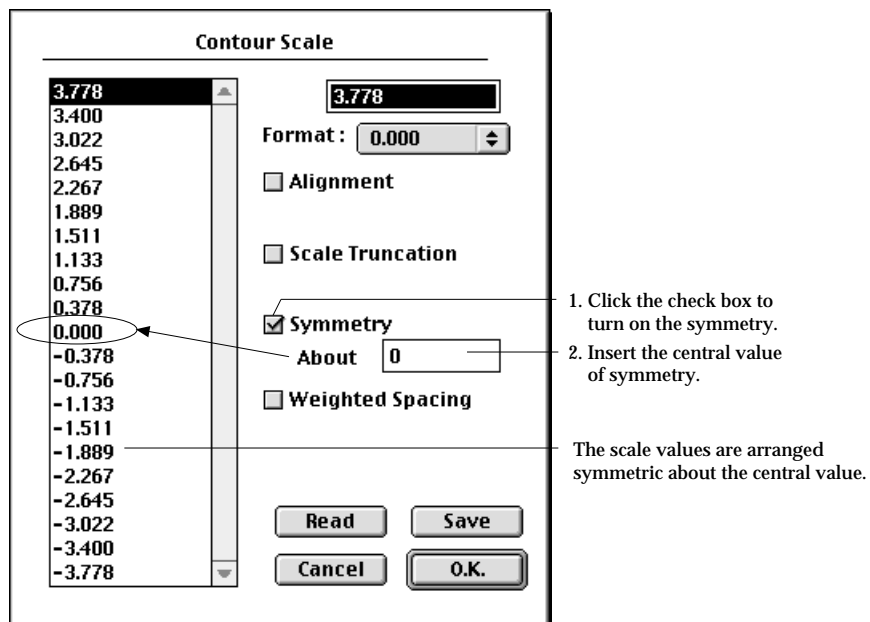
The scale values may be determined so that each of them is truncated at the specified decimal point. Check "Scale Truncation" box by clicking it. Then, an editable text item appears below the check box. Insert into this text box the value indicating the decimal point for truncation. The scale values are computed again with the specified truncation. The other rules of computing the scale values continue to be applied in determining them.



<Truncating the scale values>

### ■ Getting symmetrically arranged scale values

It is sometimes useful to have the scale values arranged symmetrically about a specified value. Check "Symmetry" box by clicking it. Then, an editable text item labeled "About" appears below the check box. Insert the value specifying the central value of the symmetry into this text box. The scale values are reset so that the scale values are arranged symmetrically about the specified value.



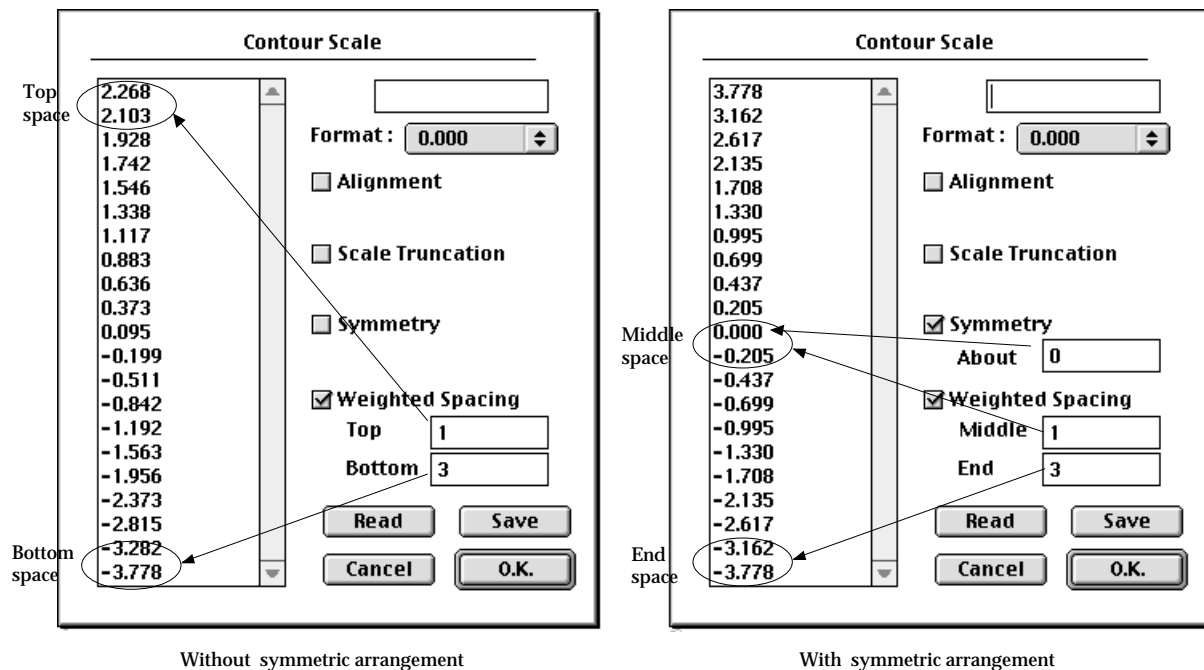
<Arranging the scale values symmetrically>

### ■ Spacing the scale values with weight

The scale values are determined initially such that they are spread with uniform spacing between the minimum and the maximum data values. This uniform spacing may be altered to a weighted spacing. In other words, the spacing can be made to vary gradually from top to bottom with given weight. The weight is either the ratio between the space of top band and the bottom band, or the ratio between the central and the end bands, depending on whether the arrangement of the scale values is symmetric or not.

If the arrangement is symmetric, editable text boxes labeled respectively as “Center” and “End” appear under the “Weighted Spacing” check box, when this box is checked.

If the arrangement is symmetric, editable text boxes labeled respectively as “Top” and “Bottom” appear under the “Weighted Spacing” check box, when this box is checked.



<Spacing the scale values with weight>

### ■ Possible combination of contour scale options

As described above, there are options for adjusting the contour scale values, namely “Alignment”, “Scale Truncation”, “Symmetry” and “Weighted Spacing”. An option can or cannot be used in combination with others. The allowable combinations are summarized below.


If two options are not compatible, one option is automatically turned off when the other option is turned on. For example, “Alignment” and “Symmetry” cannot be

used at the same time as shown in the table. Thus, checking “Symmetry” option will automatically uncheck “Alignment” option, if it is turned on.


<Possible combinations of contour scale options>

Possible combinations							Options
<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<b>Alignment</b>
<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<b>Scale Truncation</b>
<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<b>Symmetry</b>
<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<b>Weighted Spacing</b>

#### ■ Saving contour scale values

A set of contour scale values can be saved in a file for future use. Click  button in the dialog. Then, a standard file saving dialog appears. The file is initially named as “untitled.cnt”, but can be substituted by any other name. The extension “.cnt” is attached simply to facilitate the identification of contour scale files, but not always required. The number of contour bands as well as all the contour scale values are saved in the designated file.

#### ■ Reading contour scale values

A set of contour scale values can be retrieved from a file with previously saved contour scales. Click  button in the dialog. Then, a standard file opening dialog appears. Browse the dialog and find the desired file with contour scale values. Only files with contour data values will appear in the browser. Opening the file will immediately substitute the current contour scale values by the set in the file.

Saving and reading contour scale values are useful functions when a common contour scale should be applied to different data sets. In such case, the contour scale should be determined wide enough to cover the range of all the data sets.




## Setting a cut plane for contouring

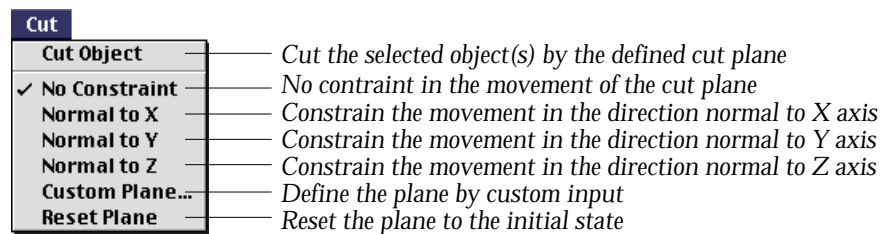
As described in the previous sections, the contours may be displayed on user-defined auxiliary planes such as a cut plane, cross planes, or parallel planes. These planes should be defined prior to contouring on them. “Cut plane”, “Cross Planes” and “Parallel planes” items in the object type popup menu of “Contour Display” dialog become enabled only when the corresponding auxiliary planes are defined before opening the dialog.

A cut plane is a kind of volume visualization aid and can be used either as an auxiliary plane for contouring, or as a plane splitting the selected object(s). This section describes how to define a cut plane and how to use it as an auxiliary plane for contouring.

### ■ Activating the cut plane setting mode

The cut plane setting mode should be activated in order to start defining a cut plane. This can be done by the following steps.

- 1) Start the volume selection tool by pressing  button.  
The program is now ready to select volume object(s) in the model.
- 2) Select object(s) in which the cut plane is to be defined.  
Select one or more object(s) which form a continuous volume which will contain the cut plane.
- 3) Start the cut plane setting tool by pressing  button.  
Get into the cut plane setting mode by clicking  button in the tool palette. Then, a rectangular box will surround the selected object(s). The initial state of the cut plane is also shown within the box. At the same time, **Cut** menu appears on the menu bar. This menu has items for cut plane setting and other related functions.



The cut plane setting mode is deactivated simply by activating other tool in the tool palette.

### ■ Setting a cut plane

The initial or the previous setting is shown, when the cut plane setting mode is activated. There are 2 different ways of setting the position and the orientation of a cut plane to the desired state.

- Interactive setting by dragging the vertices, edges or face of the cut plane.  
The position and orientation of a cut plane can be modified by interactively moving the vertexes, the edges, or the face of the cut plane. Click one of the vertexes, edges or face, and drag the selected part. The clicked part moves along with the cursor. Repeat the above steps many times until the plane is set at the desired position and orientation.  
While setting the cut plane interactively, the movement of vertexes, edges or face can be constrained by setting appropriate options in menu. The currently effective option is indicated by a check mark in front of the corresponding menu item.
  - “No Constraint” : free movement without constraint.
  - “Normal to X” : Movement is constrained to the direction normal to X.
  - “Normal to Y” : Movement is constrained to the direction normal to Y.
  - “Normal to Z” : Movement is constrained to the direction normal to Z.
- Custom setting by using “Cut Plane Setting” dialog.

Select “Custom Plane...” item from **Cut** menu. Then, “Cut Plane Setting” dialog appears. The dialog has editable text items for the coordinates of 3 points which are on the cut plane. The position and the orientation of the cut plane are defined by inserting the coordinates in the text boxes.

	Point 1	Point 2	Point 3
X=	14.5	14.5	14.5
Y=	9	9	23
Z=	7	-5.25	-5.25

In order to reset the cut plane setting to the initial state, select “Reset Plane” menu item from **Cut** menu.

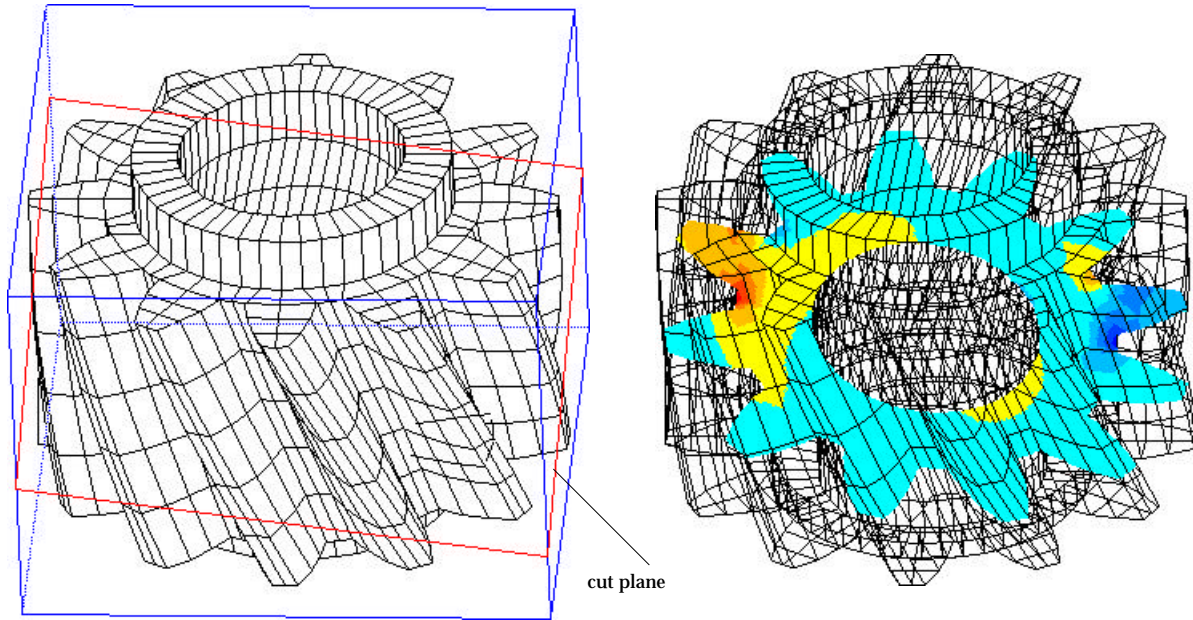
### ■ Contouring on a cut plane

As explained above, the cut plane may be used for contouring. If a cut plane is set immediately prior to opening “Contour Display” dialog, “Cut Plane” item of the contouring object popup menu is enabled and is selected. (Refer to the previous section, “Setting contouring options”.) If the selection of this popup menu item is not altered, the contour will be drawn on the cut plane.

Contouring Object



There are options for setting the style of the surrounding boundary surface rendering. Refer to “Selecting the style of boundary surface rendering” for more detailed description on this subject.





Setting the cut plane

Contour image rendered on the cut plane


### Setting the cut plane and contouring on the plane

#### ■ Splitting objects using the cut plane

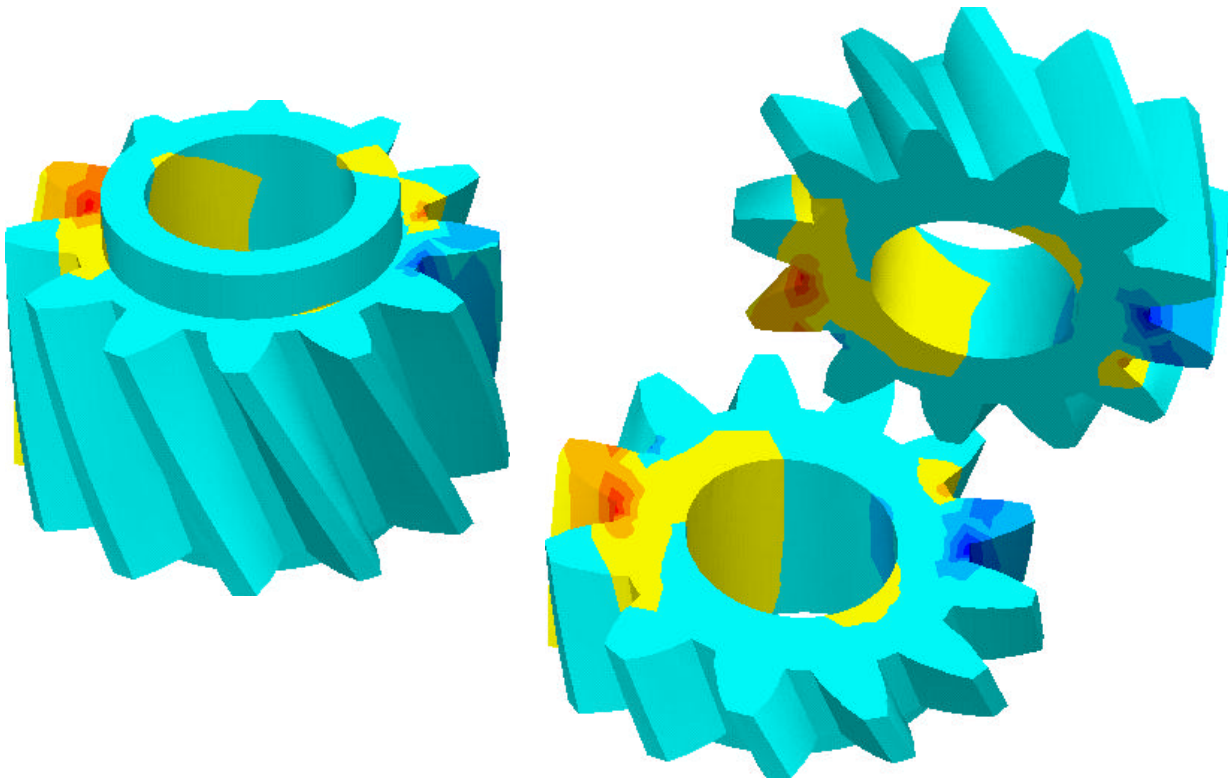
The cut plane may also be used as a plane splitting the selected objects. The split parts can be separated later for improved visualization of 3-dimensional volume data. Splitting and separating of volume objects can be achieved by the following steps:

- 1) Set the cut plane as explained above.  
The cut plane is defined within the selected object.
- 2) Split the object by choosing “Cut Object” item from **Cut** menu.  
The menu is now the right most on the menu bar. Choose “Cut Object” item from the menu. Then, the selected objects are split by the cut plane. If the objects are rendered in shading or contouring, the rendered image is maintained, as before, after splitting. If the objects are represented by wireframe meshes, newly added wireframe meshes will be shown after splitting.
- 3) Select one part of the split objects.  
Start the volume selection tool by pressing  button, and select one of the split parts.
- 3) Move or rotate the selected part.  
Start the object movement tool or the object rotation tool by pressing  or



 tool button respectively. Move or rotate the selected part using the tool. By repeating the movement or rotation of the selected part, the split objects can be separated as desired.

If the objects are rendered in shading or contouring, rendering is updated while moving or rotating the selected part.



Contour image rendered before the objects are split

Contour image rendered after the objects are split and separated




Splitting and separating objects using cut plane

## Setting parallel planes for contouring

The data distribution inside a 3-dimensional volume can be visualized effectively using another volume visualization aid, i.e., parallel planes as auxiliary planes for contouring. Parallel planes are a number of planes defined within selected volumes, normal to the specified coordinate axis. The direction and the location of the planes as well as the number of planes can be set as described below.

### ■ Activating the parallel plane setting mode

The parallel plane setting mode should be activated in order to start defining parallel planes. This can be done by the following steps.

- 1) Start the volume selection tool by pressing  button.  
One can now select volume object(s) in the model.
- 2) Select object(s) in which the parallel planes are to be defined.  
Select one or more object(s) which form a continuous volume and which will contain the parallel planes.
- 3) Start the parallel plane setting tool by pressing  button.  
Get into the parallel plane setting mode by clicking  button in the tool palette. Then, a rectangular box will surround the selected object(s). The initial state of the parallel planes is also shown within the box. At the same time, **Parallel** menu appears on the menu bar as shown below.

Parallel	
<input checked="" type="checkbox"/> Normal to X	Set the parallel planes normal to X axis
Normal to Y	Set the parallel planes normal to Y axis
Normal to Z	Set the parallel planes normal to Z axis
1 Plane	1 parallel plane
2 Planes	2 parallel planes
3 Planes	3 parallel planes
4 Planes	4 parallel planes
<input checked="" type="checkbox"/> 5 Planes	5 parallel planes
10 Planes	10 parallel planes
Others...	Set the number of parallel planes
Custom Planes...	Define the parallel planes by custom input
<input checked="" type="checkbox"/> By Coordinates	Represent the location of a parallel plane by coordinates
By Ratio	Represent the location of a parallel plane by ratio

The parallel plane setting mode is deactivated simply by activating any other tool in the tool palette.

### ■ Setting parallel planes interactively

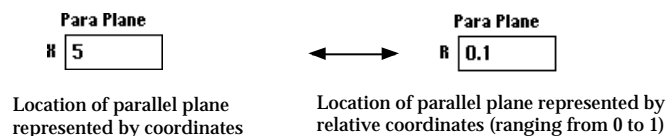
The initial or the previous setting is shown, when the parallel plane setting mode is activated. This setting can be altered interactively as explained below.

- Setting the orientation of the parallel planes : The orientation of the parallel planes can be set normal to X, Y or Z axis by choosing respectively “Normal to X”, “Normal to Y” or “Normal to Z” item from **Parallel** menu.

- Determining the number of parallel planes : The number of parallel planes can be specified simply by choosing “1 Plane” through “10 Planes” item from **Parallel** menu, or choosing “Others..,” item. In the last case, any number of planes can be set by inserting the desired number in the editable text box of the following dialog.



- Locating the parallel planes by mouse dragging : The parallel planes can be located at desired points by dragging each of the planes using mouse. Place the screen cursor over the image of a parallel plane, and drag the plane to the desired point. The plane moves along with mouse movement.
  - 1) Select the parallel plane to move.  
Place the cursor over the image of the plane, and click the mouse button.
  - 2) Drag the plane with the mouse button pressed.  
The selected parallel plane moves along with the cursor.
  - 3) Release the mouse button.  
The parallel plane settles at the position where the button is released.
  - 4) Repeat step 1) 2) and 3) until all the parallel planes are positioned as desired.
- Placing the parallel planes by keyboard input : While the parallel plane setting mode is active, the editable text box labeled “Para Plane” is shown at the lower part of the tool palette. The location of the parallel planes can be specified by direct keyboard input in this text box by the following steps:
  - 1) Select the parallel plane to be located.  
Place the screen cursor over the image of the plane, and click the mouse button. The location of the plane is given in the text box.
  - 2) Edit the text.  
Change the location by editing the text of location represented either by coordinates or by ratio.
  - 2) Press **return** key (Windows : **Enter** key).  
The edited text of the location is applied by pressing **return** key (Windows : **Enter** key), and the parallel plane moves to the specified location.



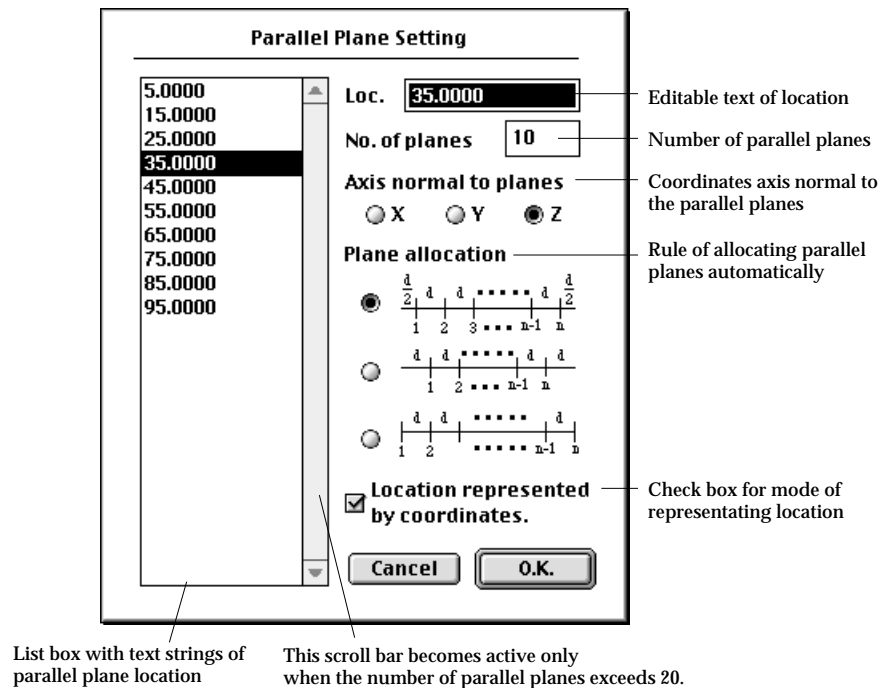
< Editable text item for keyboard input of parallel plane location >

In the above procedure, the location can be specified either by the coordinates or as the ratio depending on whether “By Coordinates” or “By Ratio” item is checked in **Parallel** menu.

In case of “By Coordinates”, one of X, Y and Z coordinates applies depending on which coordinates axis the parallel plane is normal to.

### ■ Setting parallel planes by custom input

There is another method of setting the parallel planes. That is to use “Parallel Plane Setting” dialog. In order to start the dialog, choose “Custom Planes...” item in **Parallel** menu. The parallel planes are newly defined and displayed by clicking **O.K.** button after setting the relevant items in the dialog as detailed below.

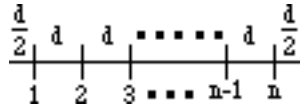
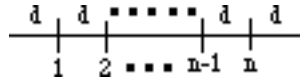
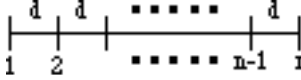


< Parallel plane setting dialog >

- Setting the number of parallel planes : Insert the number of parallel planes in the editable text box labeled as “No. of planes”.
- Setting the orientation of the parallel planes : Set the orientation of the parallel planes by turning one of the radio buttons labeled “X”, “Y” and “Z”, which represent direction normal to X, Y and Z coordinates axis, respectively.
- Deciding the rule of plane allocation : The parallel planes are initially located automatically at the points evenly dividing the total length of the model in the direction normal to the planes. There are 3 different rules of determining the division points. One of these 3 rules should be chosen by turning on the

corresponding radio button. The example in the following table shows how the 10 parallel planes are located over the distance 100 by applying each one of the rules.

<Rules of allocating the parallel plane>

Allocation rule options	Principle	Example divisions
	The total length is evenly divided into $n$ segments, and one plane is allocated in the middle of each segment.	5.00, 15.00, 25.00, 35.00, 45.00, 55.00, 65.00, 75.00, 85.00, 95.00
	The total length is evenly divided into $(n+1)$ segments, and planes are allocated at ends of segments excluding the first and the last end points.	9.09, 18.18, 27.27, 36.36, 45.45, 54.55, 63.64, 72.73, 81.82, 90.91
	The total length is evenly divided into $(n-1)$ segments, and planes are allocated at ends of segments including the first and the last end points.	0.00, 11.11, 22.22, 33.33, 44.44, 55.56, 66.67, 77.78, 88.89, 100.00

- Setting the mode of representing the location of a parallel plane : The location of a parallel plane may be represented either by coordinates or by ratio. If the mode is set to “ by coordinates”, the location of a parallel plane should be defined by the actual coordinates along the axis normal to the plane. Otherwise, location should be defined by the ratio scaled from 0 to 1 along the total length of the model in the direction normal to the plane. Check the box labeled “Location represented by coordinates” in order to set the mode to “by coordinates”, or uncheck the box to set mode to “by ratio”.
- Editing the text string of parallel plane locations: The locations of all the parallel planes are displayed as text strings in the list box of the dialog. The initial locations are determined automatically by the size of the selected parts, number of planes, etc. The parallel plane location may be altered by editing the text strings of the initial setting by the following steps:

1) Click the text string to modify.

The clicked text is highlighted, and an identical text string is placed in the

editable text box in the dialog.

- 2) Edit the text string in the editable text box.

The selected text string in the list box is updated while the string in the editable text box is being edited. The edited string represents the modified location of the parallel plane.

- 3) Repeat step 1) and 2) for all the parallel plane locations to modify.

Edit the text strings one after another until all the relevant text strings are modified as desired.

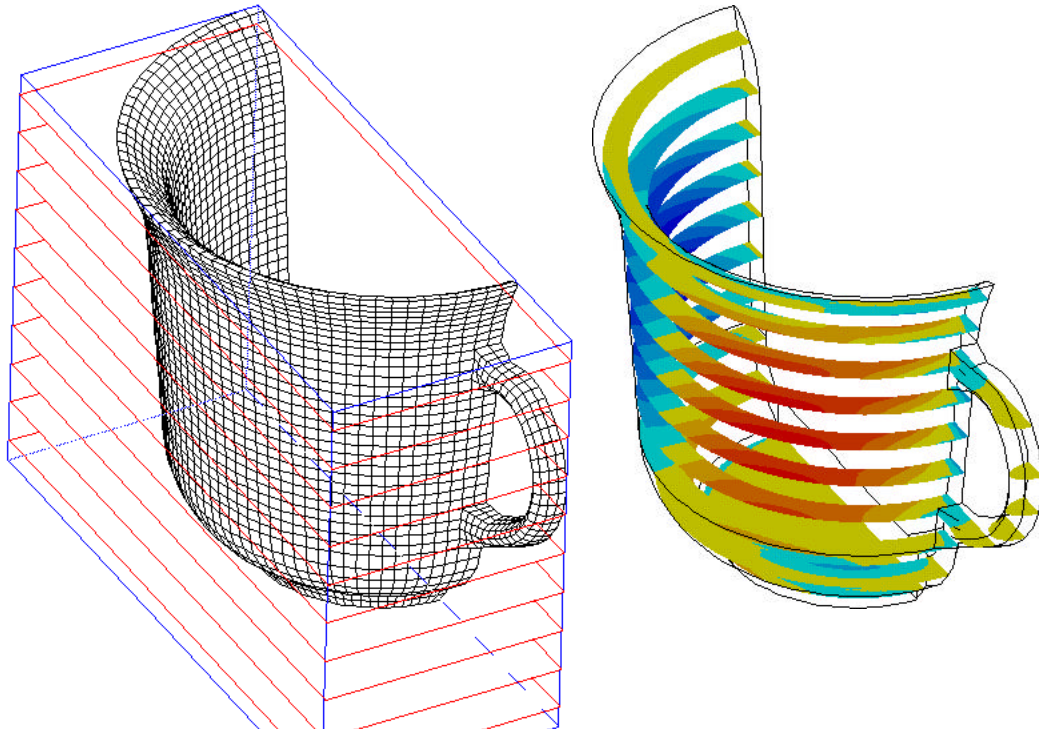
### ■ Contouring on parallel planes

The parallel planes are used for contouring data inside 3-dimensional volumes. If parallel planes are set immediately prior to opening “Contour..” dialog, “Parallel Planes” item of the contouring object popup menu is enabled and is selected. If the selection of this popup menu item is not altered, the contour will be drawn on the parallel planes.

Contouring Object

Parallel Planes

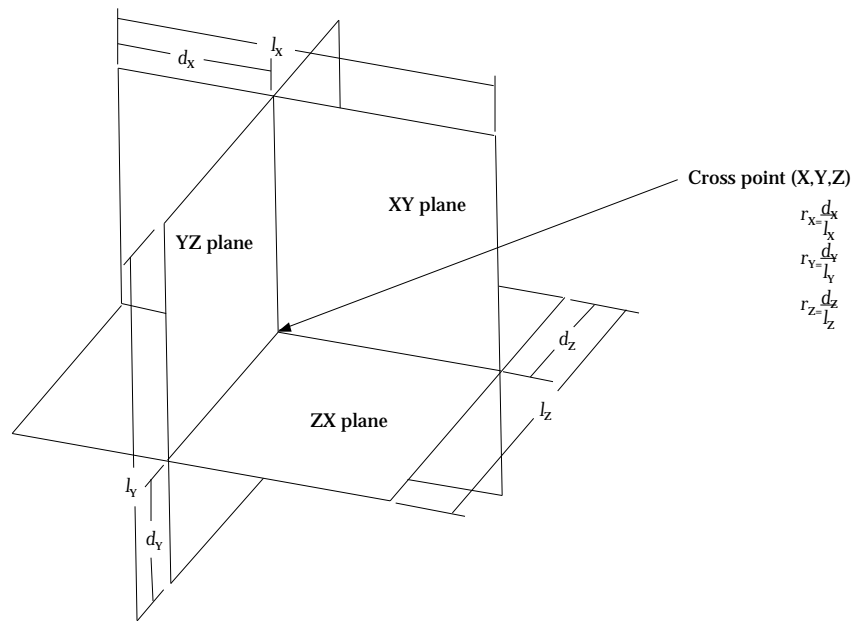
There are options setting the style of the surrounding boundary surface rendering. Refer to "Selecting the style of boundary surface rendering" for more detailed description on this subject.



Setting the parallel plane and contouring on the plane

## Setting cross planes for contouring




The data distribution around a specific point within a 3-dimensional volume can be visualized effectively using cross planes as auxiliary planes for contouring. Cross planes are also volume visualization aid and consist of 3 orthogonal planes, XY plane, YZ plane and ZX plane, crossing at a specified point, namely the cross point. Once the cross planes are defined, the contour images are drawn on these planes. This method is especially suitable to represent the 3 dimensional variation of data in the vicinity of the cross point. Each of the cross planes can be turned on or off, and the location of the cross point can be specified as described in the following.



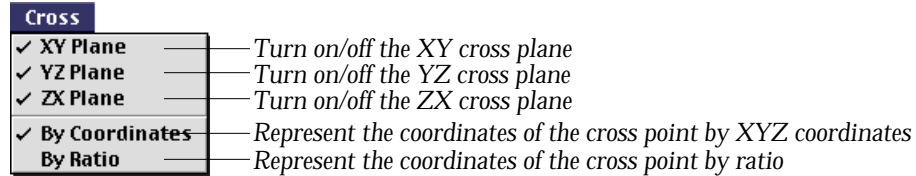
< Cross planes and cross point >

### ■ Activating the cross plane setting mode

The cross plane setting mode should be activated in order to start define the cross planes. This can be done by the following steps.

- 1) Start the volume selection tool by pressing  button.  
One can now select volume object(s) in the model.
- 2) Select object(s) in which the cross planes are to be defined.  
Select one or more object(s) which form a continuous volume which will contain the cross planes. The cross point is not necessarily within the volume.
- 3) Start the cross plane setting tool by pressing  button.  
Get into the cross plane setting mode by clicking  button in the tool palette. Then a rectangular box will surround the selected object(s). The

initial state of the cross planes is also shown within the box. At the same time, **Cross** menu appears on the menu bar. This menu has items related to cross plane setting.



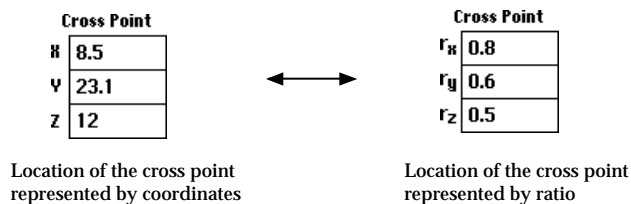
The cross plane setting mode is deactivated simply by activating other tool in the tool palette.

### ■ Setting cross planes

The initial or the previous setting is shown, when the parallel plane setting mode is activated. This setting can be altered interactively as explained below.

- Turning on or off the cross planes : Each one of the cross planes can be turned on or off individually by choosing “XY Plane”, “YZ Plane” or “ZX Plane” item from **Cross** menu. If the item is checked, the corresponding cross plane is turned on. Otherwise, it is turned off. Only the turned on planes are used for contouring.
- Setting the mode of representing the location of the cross point : The positions of the cross planes are determined by that of the cross point. While the cross plane setting mode is active, the editable text box labeled “Cross Point” is shown at the lower part of the tool palette. The text strings display the current location of the cross point, and can also be used to input the location. The location of the cross point can be expressed either by XYZ coordinates or by the ratios in X, Y and Z axis directions.

The mode of expressing the location can be selected by choosing “By Coordinates” or “By Ratio” item in **Cross** menu.



< Editable text item for coordinates of cross point >

The ratios  $r_x$ ,  $r_y$  and  $r_z$  represent the relative location of the cross point with respect to the total length of the model in X, Y and Z directions, and has value between 0 and 1. (Refer to the figure <Cross planes and cross point>)



- Placing the cross planes by mouse dragging : Each one of the cross planes can





be moved to the desired position by mouse dragging by the following steps.

- 1) Select the cross plane to move.  
Place the screen cursor over the image of the plane, and click the mouse button. The coordinates of the cross point are given in the text box.
- 2) Drag the plane with the mouse button pressed.  
The selected cross plane moves along with the cursor.
- 3) Release the mouse button.  
The cross plane settles at the position where the button is released.
- 4) Repeat step 1) 2) and 3) until all the cross planes are positioned as desired.  
Place all the parallel planes one by one, as explained above.

- Locating the cross point by keyboard input: The coordinates or the ratios of the cross point can be inputted directly in the text box at the lower part of the tool palette by the following steps:

- 1) Set the appropriate mode of representing the location of the cross point.  
The mode can be set by choosing “By Coordinates” or “By Ratio” item in **Cross** menu.
- 2) Edit the text.  
Change the location by editing the text of location represented either by coordinates or by ratio.
- 2) Press  key (Windows :  key).

The edited text of the location is applied by pressing  key (Windows :  key), and the cross planes move to form the cross point at the new location.

## ■ Contouring on cross planes

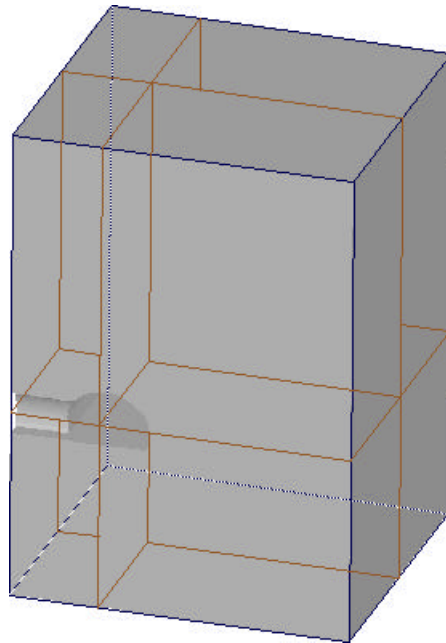
The cross planes are used for contouring data inside 3-dimensional volumes. If cross planes are set immediately prior to opening “Contour” dialog, “Cross Planes” item of the contouring object popup menu is enabled and is selected. (Refer to the previous section, “Setting contouring options”.) If the selection of this popup menu item is not altered, the contour will be drawn on the cross planes.

Contouring Object

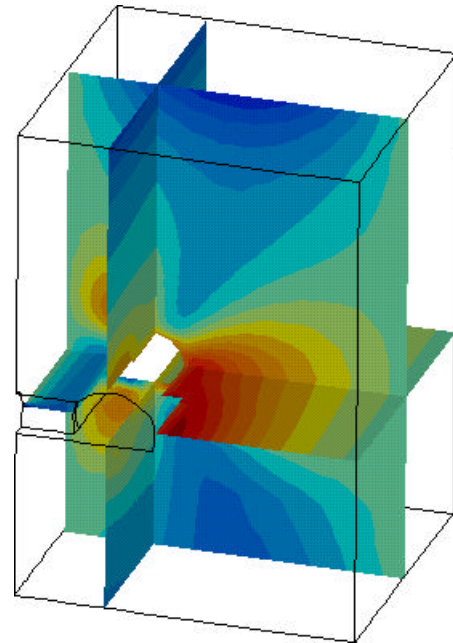


There are options setting the style of the surrounding boundary surface rendering. Refer to (...) for more detailed description on this subject.

Each one of the cross planes can be turned on or off as described earlier this section. A contour image will be drawn only on cross planes which are turned on.



Cross plane setting



Contour image on the cross plane

< Setting the cross plane and contouring on the plane >


## Other functions related to contouring

Various functions related to contouring are described in the previous sections. There are other functions which are devised to supplement contouring. They are contour marking, sampling contour value and turning on/off the contour scale, and described below.



### ■ Contour marking over a contour image


A contour image is rendered by a limited number of contour bands as specified in “Contour Display” dialog, and accordingly does not give information on minute variation of data within a contour band. This weak point of contour image is supplemented by the function of marking a contour line over the image. Contour marking makes up for the wide gap of data values that can be represented by contour bands, and thus gives more detailed information on the data distribution. This function can proceed interactively by the following steps:

- 1) Start the contour marking tool by pressing the tool button  , if it is not yet activated.

The contour marking tool is automatically activated whenever a new contour

image is drawn. Activation of the tool is not necessary in this case. But, other tool may have been activated after creating a contour image, and thus the contour marking tool is deactivated. In such case, the tool button should be pressed in order to activate the tool again.

- 2) Place the screen cursor either over the contour image or over the contour scale bar.

As the cursor moves into the VisualFEA window, its shape turns into . A data value may be sampled either at the point of the cursor on the contour image or on the contour scale bar. Place the cursor over the point from which the data value is to be sampled.

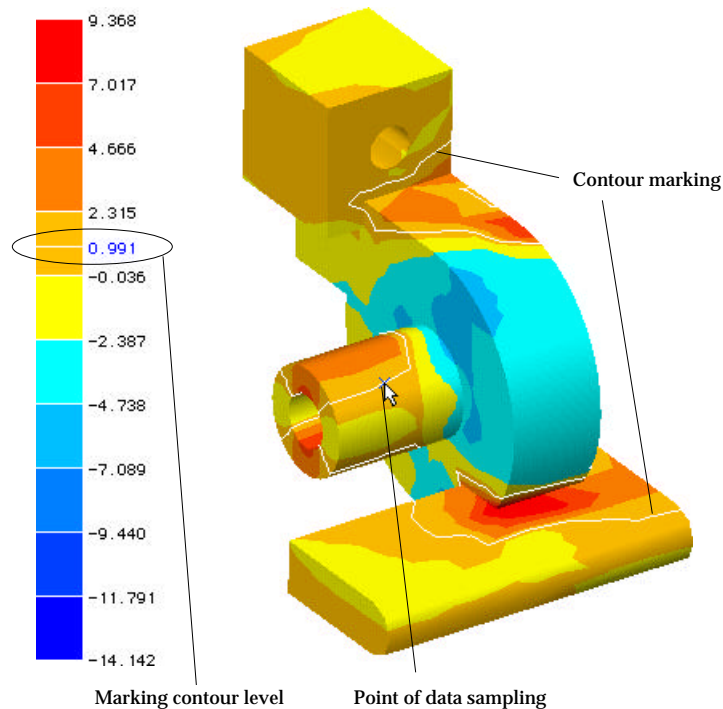
- 3) Sample the data value by clicking the point.

The data value is sampled at the point of mouse click, and a contour line representing the sampled value is marked over the contour image. The level of the marking contour is displayed over the contour scale bar, and in the editable text box at the lower part of the tool palette.



- 4) Move the mouse with button pressed.

The contour marking is updated in real time as the screen cursor moves along with the mouse movement. At the same time, the scale value is also continuously updated.

Contour Level  
f 0.99112




< Contour marking >

The above step 2), 3) and 4) may be repeated as long as the contour marking tool is active. The editable text box at the lower part of the tool palette shows the level of the marked contour. The contour marking may also be renewed by inserting a value in the box and pressing  key (Windows :  key).

The precision of the marking contour line is determined by the viewing scale of the model and the screen resolution. Therefore, you can obtain more precise contour line by zooming in the screen view. However, the analysis results themselves have limited accuracy, and zooming beyond a certain limit may not have any significance.

### ■ Sampling contour value by specifying the coordinates

Contour marking is a method to sample the contour value pointed by screen cursor. Another method of contour value sampling is specifying the coordinates of the sampling point. The advantage of this method is that any point within a 3-dimensional volume can be sampled, while contour marking is only possible for a point on outer surfaces of volume. The coordinates of the sampling point can be entered by the following steps:

- 1) Start the contour marking tool by pressing the tool button  , if it is not yet activated.

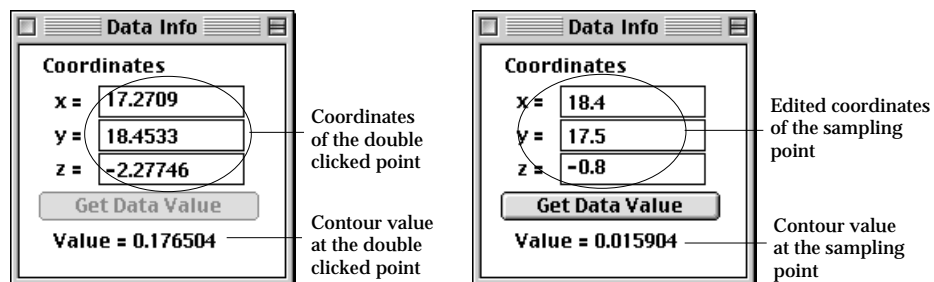
Refer to the previous section.

- 2) Place the screen cursor over the contour image.

Place the cursor around the point of sampling. It is not necessary to place the cursor precisely at the point of sampling, because the location of the point is to be specified more precisely by its coordinates afterward.

- 3) Double click the point

Double clicking will bring up “Data Info” dialog. The dialog has editable text items for the coordinates of the sampling point.



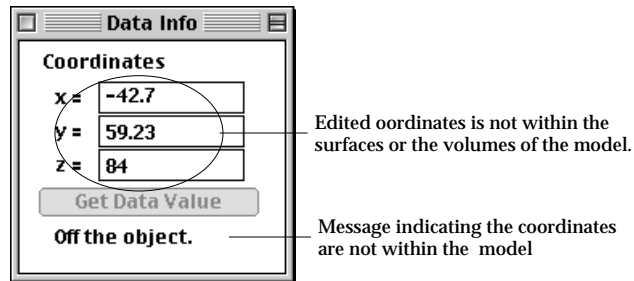
< “Data Info” dialog >

- 4) Edit the text string of the coordinates

Enter the coordinates of the desired sampling point by editing the strings in the editable text boxes.

- 5) Press **Get Data Value** button.

The button is dimmed initially when the dialog is open, and becomes enabled when any of the text strings of the coordinates is edited. Press the button. Then the data value at the sampling point is displayed below the button. However, the specified sampling point may not be within the volume. If so, the data value cannot be sampled. Instead, a message indicating that the coordinates are not within the model, will appear on the dialog.



< “Off the object” message >

#### ■ Turning on and off the contour scale bar



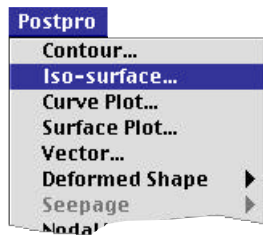
The contour scale bar is automatically displayed at the left edge of VisualFEA window, when the contour image is rendered. If desired, this scale bar can be made hidden by selecting “Hide Scale” item from **Postpro** menu. The menu item is enabled only when a contour image is rendered. Selecting the item will remove the scale bar from the screen, and alter the menu string into “Show Scale.” The scale bar can be shown again by simply selecting the “Show Scale” item.



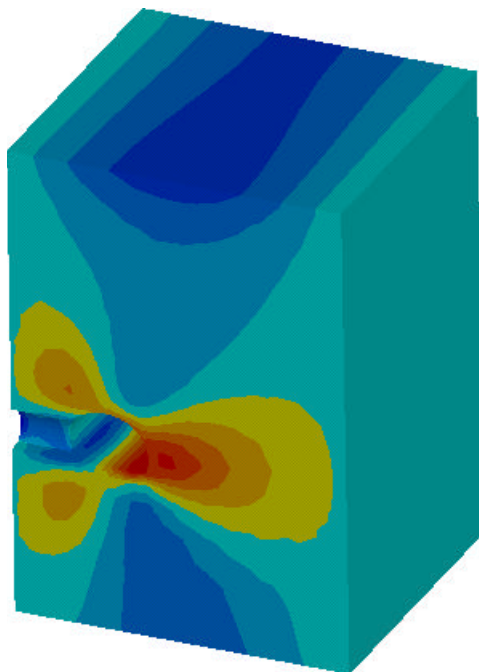
## Visualizing Scalar Data by Iso-surface and others

Besides contouring, VisualFEA supports other popular methods of visualizing scalar data in 2-D or 3-D space: iso-surface rendering, curve plotting and surface plotting. They are not so widely used as contouring in other finite element analysis software. But they are as efficient and convenient as contouring, and may be used in lieu of, or in conjunction with contouring.

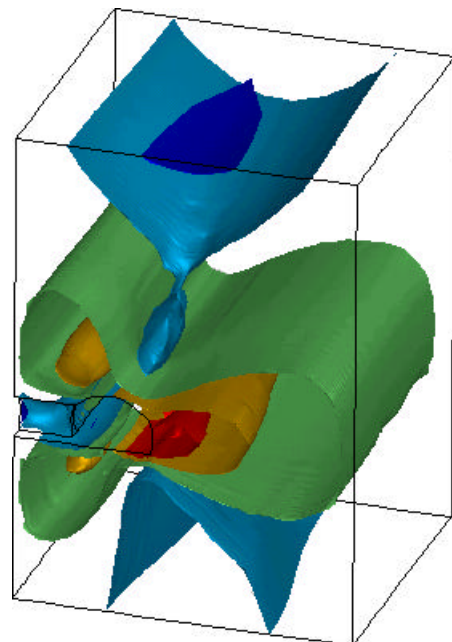
### Visualizing scalar data using iso-surfaces



Iso-surface representation is an efficient method of volume visualization, i.e., visualizing data distribution in 3-D space. Iso-surfaces within a 3-D volume are analogous to contours on surface. An iso-surface represents a surface in a 3-D volume, while a contour line represents a curve on plane or surface, on which all the points have equal data value. The 3-D model space is split by a number of iso-surfaces, and each of the split parts represents a region of the model with certain range of the data value. Thus, the overall data distribution in 3-D space can be grasped easily from the iso-surface image.



Contour image



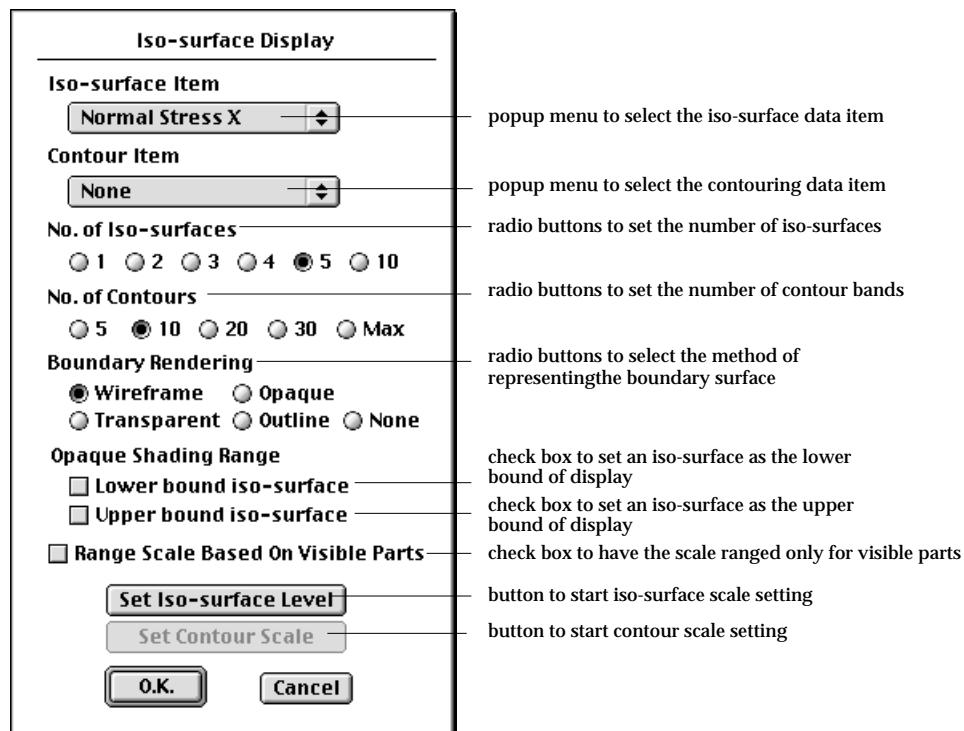
Isosurface image

<Comparison of contour and iso-surface images>

### ■ Setting the iso-surface display options

In order to get an iso-surface image of a data set, first select “Iso-surface...” items from **Postpro** menu. Then, “Iso-surface Display” dialog appears on the screen. This dialog is similar to “Contour Display” dialog described in the preceding chapters. However, “Iso-surface Display” dialog has more items than “Contour Display” dialog. There are a number of items in this dialog.

Each item has default setting. Change the setting if necessary. You may also specify the scale of contour bands as well as iso-surfaces. Click **O.K.** button if every item is set as desired. Then, the iso-surface image will be displayed with a scale bar which indicates the data value represented by each one of the iso-surfaces.



<"Iso-surface Display" dialog >

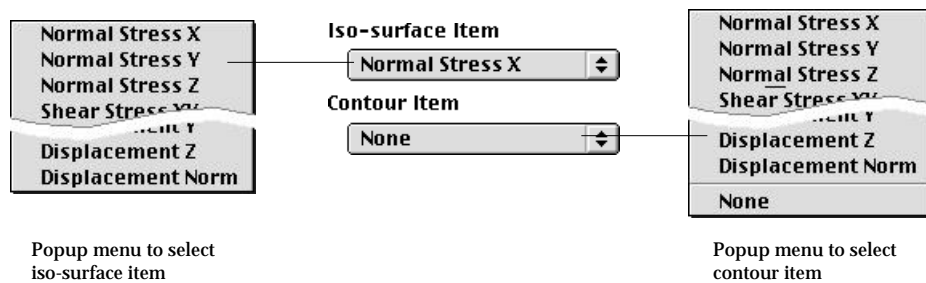
### ■ Selecting the data item to be represented by iso-surfaces

There are two popup menus in “Iso-surface Display” dialog. The data item for iso-surface representation is selected using the first popup menu, which has the same menu items as the one in “Contour Display” dialog. (Refer to “Visualizing Scalar Data by Contours” section of this chapter.) Only available data items are listed in the menu.

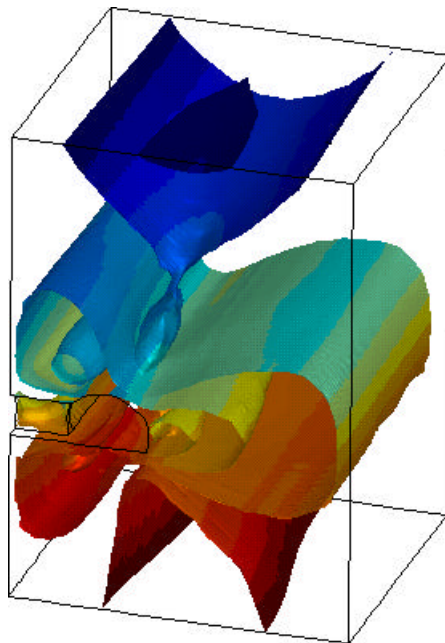
### ■ Selecting the data item to be represented by contours

Iso-surfaces may be painted with contour bands which represent distribution of another data item. For example, iso-surfaces represent the distribution of normal stress  $\sigma_x$ , and contours represent the distribution of shear stress  $\tau_{xy}$ .

The second popup menu has the menu items to select the contour data item, and has one more item “None” than the first popup menu for iso-surfaces. The popup menu is initially set as “None”, which designates no contour on the iso-surfaces. Thus each of the iso-surfaces is rendered in one color corresponding to the iso-surface level.



< Popup menus in “Iso-surface Display” dialog >



<Iso-surface image with contour>



### ■ Setting the number of iso-surfaces

The number of iso-surfaces can be set by clicking the radio button labeled with the desired number. There are 6 radio buttons under “No. of Iso-surfaces”: “1”, “2”, “3”, “4”, “5” and “10”. The number of iso-surfaces is initially set as 5 by default. The rendering time depends on the number of surfaces, and increases significantly as the number of iso-surfaces increases.

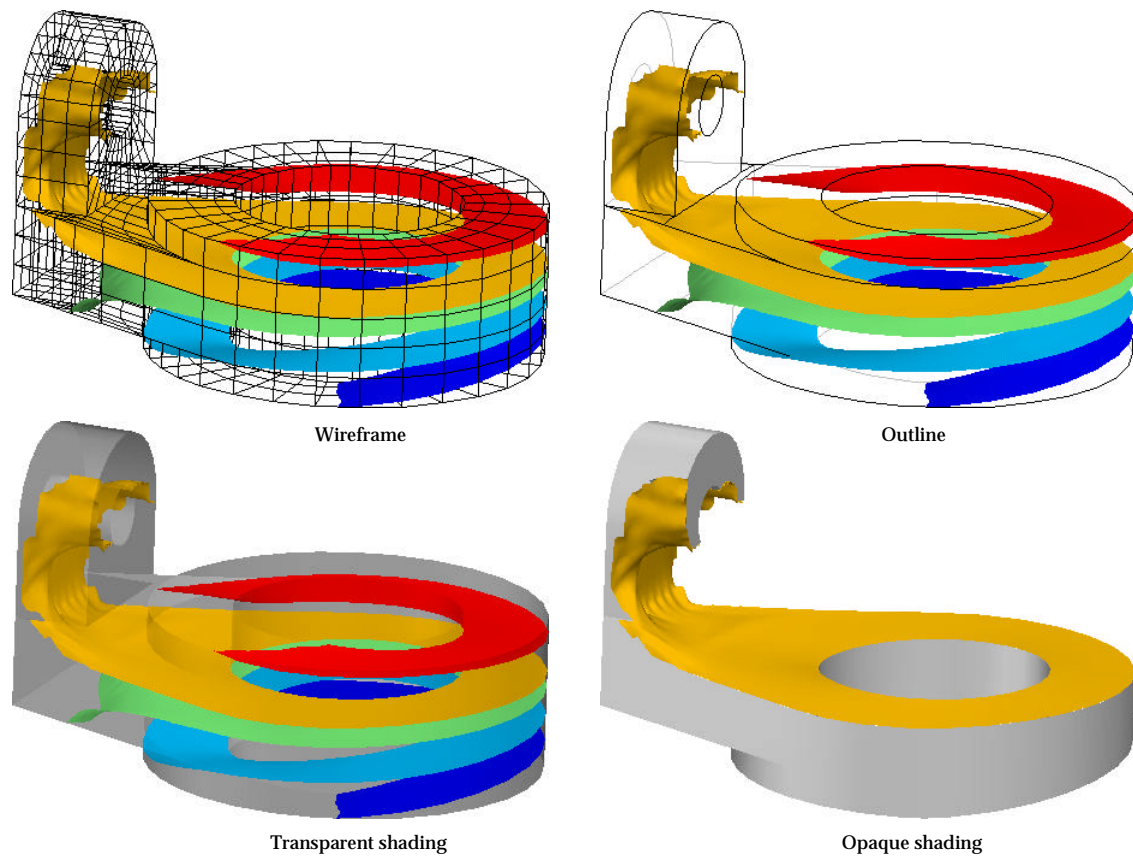
### ■ Setting the number of contour bands

In case contours are to be drawn on the iso-surfaces, the number of contours should be determined. The number of contour bands is initially set as 10 by default. This default setting can be changed by clicking the radio button labeled with the desired number of bands, which should be one of 5, 10, 20, 30 and Max. If you choose “Max”, the contour image will be rendered with as many bands as possible using the available colors. The number of contours has no effect on the rendering time.

### ■ Selecting the type of boundary surface rendering

Iso-surfaces are contained within 3-D volumes surrounded by boundary surfaces. In order to enhance the visual understanding, it is desirable to display the surrounding boundary surfaces together with iso-surfaces. There are a few different types of rendering the boundary surface, one of which should be chosen:

- “Wireframe”: The boundary surfaces are rendered in the form of wireframe. The hidden lines are removed from the wireframe rendering.
- “Opaque”: The boundary surfaces are rendered in the form of the volume surrounded by the specified upper bound or/and lower bound iso-surfaces. If this option is on, the check boxes under “Opaque shading range” are automatically checked. On the other hand, this option is automatically on if the check boxes are checked.
- “Transparent”: The boundary surfaces are rendered in transparency shading. Iso-surfaces are rendered as opaque objects surrounded by transparent boundary surfaces.
- “Outline”: The outlines of the boundary surfaces are extracted, and represented together with the iso-surfaces.
- “None”: The iso-surface image is displayed without boundary surface rendering.



< Boundary surface rendering with iso-surfaces >

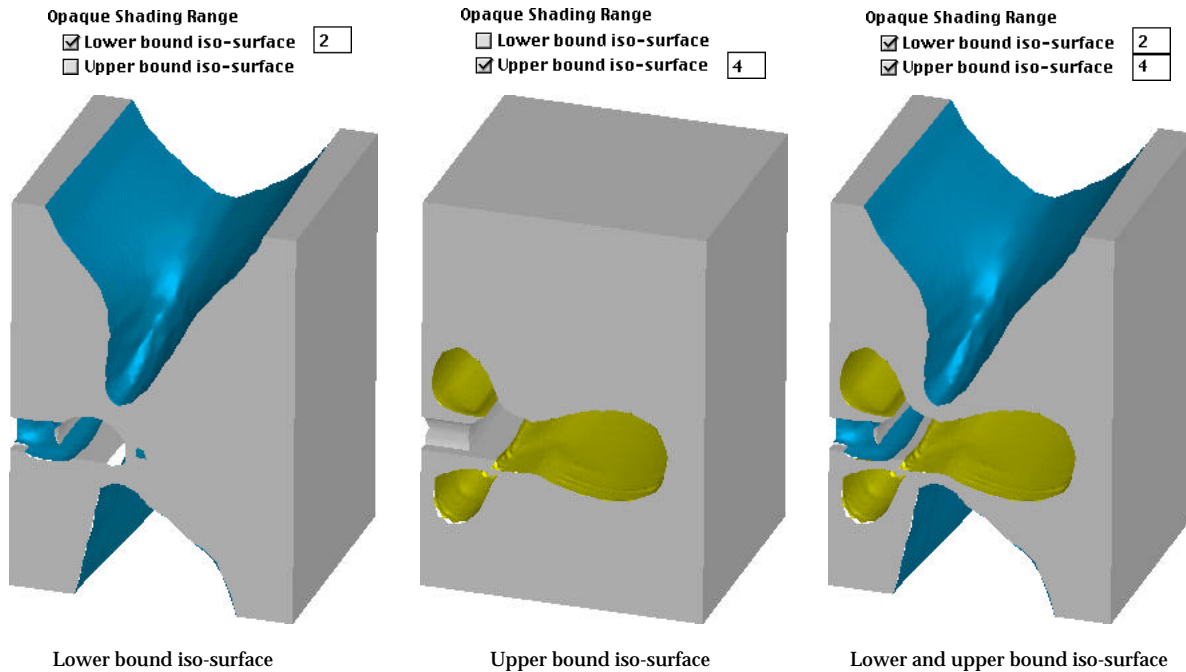
#### ■ Designating the iso-surfaces as the boundary of a truncated model

Iso-surfaces may also be used as boundary surfaces isolating a part of the model with a certain range of data values. There are two check boxes under the heading of “Opaque shading range”: “Lower bound iso-surface” and “Upper bound iso-surface”. Any one or both of the two may be checked. The checked item will be followed by editable text items to enter an iso-surface by its number as the upper bound or the lower bound surface of the truncated model. If any one of the boxes is checked, the option for boundary surface rendering will be automatically switched to “Opaque” if it is not.

There are following 3 possibilities of truncating the model by use of:

- lower bound iso-surface only.
- upper bound iso-surface only.
- both lower and upper bound iso-surfaces.

Model rendering in the above 3 cases are 1 in the figure below.



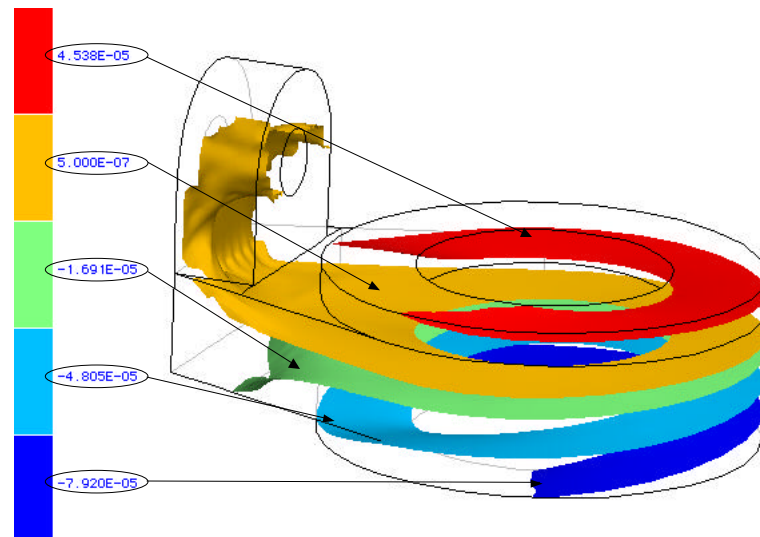
< Model truncated by iso-surfaces >

### ■ Limiting the range of iso-surface scale by actually displayed values

Levels of iso-surfaces are determined by the differences between the maximum and the minimum values of the data to be visualized. In case iso-surfaces are defined only within the selected parts or within the visible parts of the model, it is better to determine the iso-surface level based on the range of data values in the visualized parts only rather than that of the whole model. This adjustment of the data range can be achieved by checking the box labeled “Range Scale Based On Visible Parts”.

### ■ Setting the iso-surface level

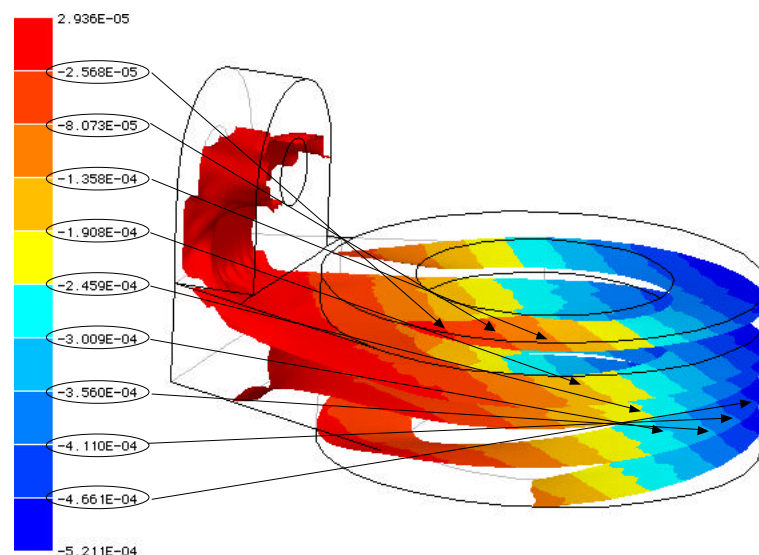
An iso-surface splits the model into two regions, one with data value just above and the other just under the data value represented by the iso-surface. This data value is termed here as iso-surface level. Each of the iso-surface levels has a value between the maximum and minimum values of the data to be visualized. The iso-surface levels are initially determined by the software, but can be altered as desired in the same manner as changing the contour scales described in the previous section. In order to start setting the iso-surface levels, first click **Set Iso-surface Level** button in “Iso-surface Display” dialog. Then, “Contour Scale” dialog will appear on the screen. This is the identical dialog used for setting the contour scale.



&lt; Scale bar for iso-surfaces &gt;

### ■ Setting the contour scale

In case the iso-surfaces are displayed with contours, the data represented by the contours are scaled independently from that of the iso-surface. In order to adjust the contour scale, first click **Set Contour Scale** button in “Iso-surface Display” dialog. Then, “Contour Scale” dialog will appear on the screen. The contour scale values can be set by using this dialog. **Set Contour Scale** button is enabled only when the second popup menu of “Iso-surface Display” dialog is set to a data item other than “None”. Otherwise, this button is dimmed.

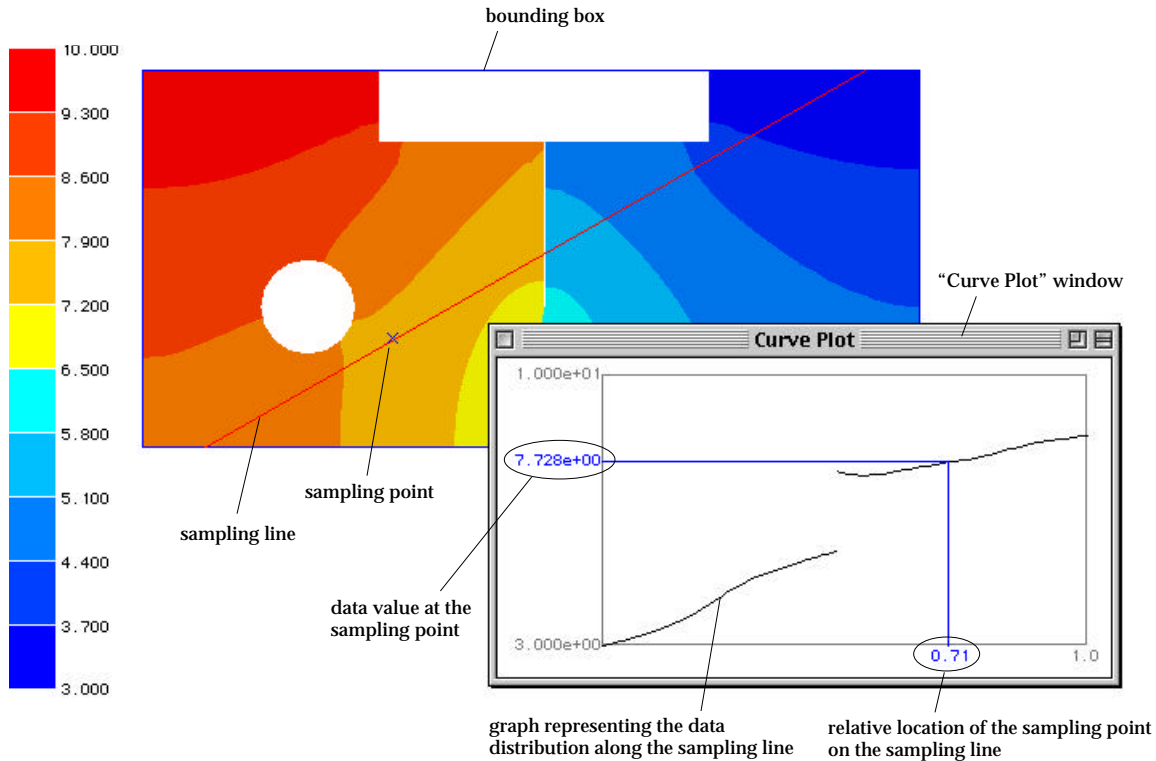


&lt; Scale bar for contours on iso-surfaces &gt;

## Curve plotting of scalar data



Curve plotting is a method of representing the data distribution along a specified line in the form of graph. This method is supplementary to contouring, and should be preceded by contouring. Curve plotting converts a contour image into graphical representation. The data values along a specified line are extracted and plotted in the form of graph, which gives, in some cases, more descriptive and more detailed information than contour lines do. Curve plotting can be applied to contour images on any plane or surface including cut plane and parallel planes. Numerical data can also be viewed using the curve plotting. This function is available for both 2-D surfaces and 3-D volumes.



< Curve plotting >

### ■ Initiating curve plotting

Curve plotting is initiated by choosing "Curve Plot" item from **Postpro** menu. Then, "Curve Plot" window appears on the screen, and **Plot** menu is attached on the menu bar. A bounding box surrounds the model rendered in the main window, and the initial state of the sampling line is drawn across the bounding box. The curve plot is displayed on "Curve Plot" window, for the initial setting.

Prior to initiating curve plotting, a contour image should be displayed. Data of the contour image becomes the subject of the curve plotting.

### ■ Modifying curve plotting

At the time curve plotting is initiated, “Curve Plot” window contains a graph representing the magnitude of the data value along the sampling line initially set by the software. This curve plot can be modified or adjusted by further interaction in the main window and “Plot Window”. Most importantly, a new plot can be obtained by moving the sampling line. The sampling line can be moved and slanted by using mouse click and drag by the following steps:

- 1) Set the **Plot** menu items as desired.  
The **Plot** menu has items related with controlling of sampling line and plotting update. This is explained in detail in the later part of this section.
- 2) Place the screen cursor on one end of the sampling line, and press the mouse button.  
The color of the sampling line is altered, indicating that the sampling line is selected.
- 3) Drag the end of the line with the mouse button pressed.  
The end of the line moves along with the cursor, and accordingly the position and the direction of the line are changed. The graph on “Curve Plot” window is updated as soon as the sampling line is modified.
- 4) Double click an edge of the bounding box to change the direction of the sampling line quickly.  
Double clicking an edge of the bounding box induces one end of the sampling line positioned on the center of the edge, and the other end on the center of the opposite edge.

### ■ Displaying the numerical value at the sampling point

A point on the sampling line can be designated as the sampling point. Click one point on the sampling line. Then, the sampling point is set at the point, and indicated by  $\times$  mark. The numerical value of the data on the sampling point is displayed in a character string on “Curve Plot” window. The relative location of the sampling point over the sampling line is indicated by hair lines and a character string on “Curve Plot” window as shown in the above figure. The sampling point can be moved along the sampling line interactively by using mouse drag. While the point is being moved, the numerical value is updated continuously together with the mark of the newly positioned point.

## ■ Resizing the graph

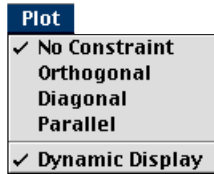
The graph is drawn to fit the plotting window. Thus, the size of the graph can be changed by resizing the window. Click the bottom right corner of the window, and drag. Or, click the zoom box of the window. Then the graph is resized to fit the new window frame.

## ■ Setting the options for curve plotting

When curve plotting is initiated, **Plot** menu appears on the menu bar. The menu has items related to controlling the sampling line and updating “Curve Plot” window.

- Controlling the movement of the sampling line.

The movement of the sampling line is controlled in the manner set by the menu items in **Plot** menu.



- “No Constraint” : The sampling line moves without constraint. While one end of the sampling line moves, the position of the other end is not altered. This is the default setting.
- “Orthogonal” : The sampling line is always positioned orthogonal to the edges of the bounding box, while one end of the line is repositioned.
- “Diagonal” : If one end of the sampling line moves along an edge of the bounding box, the other end moves in the opposite direction, so that the line rotates.
- “Parallel” : The sampling line moves in parallel. If one end of the line moves, the other end also moves the same distance.

- Setting the update of “Curve Plot” window.

The graph on “Curve Plot” window may be updated dynamically while the sampling line is being moved by dragging. Or, the graph may be updated only when the movement of the sampling line is completed by releasing the mouse button. This option can be set by checking or unchecking “Dynamic Display” item in **Plot** menu.

## ■ Terminating curve plotting

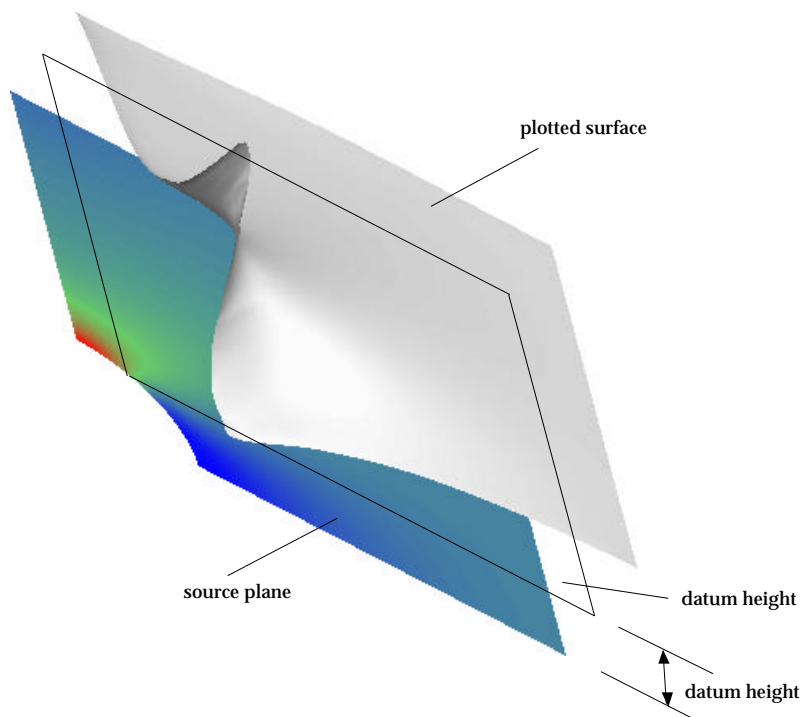
Curve plotting is terminated by clicking the close box at the top left corner of the “Curve Plot” window, or by initiating any other function. The bounding box and the sampling line disappear from the main window at the moment the “Curve Plot” window goes away. **Plot** menu also disappears from the menu bar.

## Surface plotting of scalar data



Surface plotting is a method of visualizing 2-dimensional scalar data distribution by a 3-dimensionally rendered surface over the data plane. The magnitude of the data at a point is represented by the height of the surface at that point. Surface plotting involves two surfaces, the source plane and the plotted surface. The source plane is the actual plane, the data on which are to be visualized. The plotted surface is the numerically constructed surface representing the data distribution.

The datum of the plotted surface and the scale of the height can be set as desired. The surface is rendered as a smooth and continuous surface in a few different forms. This method is also supplementary to contouring, but not necessarily preceded by contouring. This function is appropriate only for visualizing data on 2-D planes.

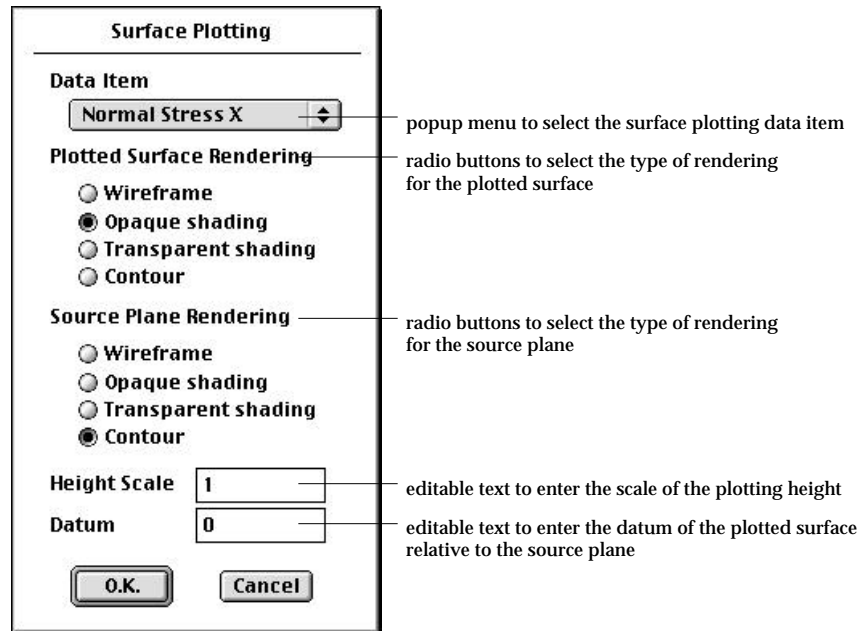


< Surface plotting >

### ■ Setting the surface plotting options

In order to get surface plotting image for a scalar data set, first select “Surface Plot...” items from **Postpro** menu. Then, “Surface Plotting” dialog appears on the screen as shown in the following figure. There are a number of items in this dialog. Each item has default setting. Change the setting if necessary. Click **O.K.** button if every item is set as desired. Then, the surface plotting image will be displayed.





&lt; “Surface Plotting” dialog &gt;

### ■ Selecting the data item

The popup menu in “Surface Plotting” dialog has the list of data items which can be displayed by surface plotting. They are identical to the popup menu item in “Contour Display” dialog. One of the items should be selected from this popup menu. The data associated with the currently selected popup menu item are displayed by surface plotting.

### ■ Selecting the type of rendering for the plotted surface

The plotted surface can be rendered in one of the following 4 different forms. Select the type of the rendering by turning on one of the following radio buttons under the heading, “Plotting Surface Rendering”:

- “Wireframe”: The plotted surface is rendered in the form of a wireframe. Hidden lines are removed from the wireframe rendering.
- “Opaque shading”: The plotted surface is rendered by opaque shading.
- “Transparent shading”: The plotted surface is rendered by transparent shading.
- “Contour”: The plotted surface is rendered by a contoured surface.

### ■ Selecting the type of rendering for the source plane

The source plane can also be rendered in one of the following 5 different forms. Select the type of the rendering by turning on one of the following radio buttons

under the heading, “Source Plane Rendering”:

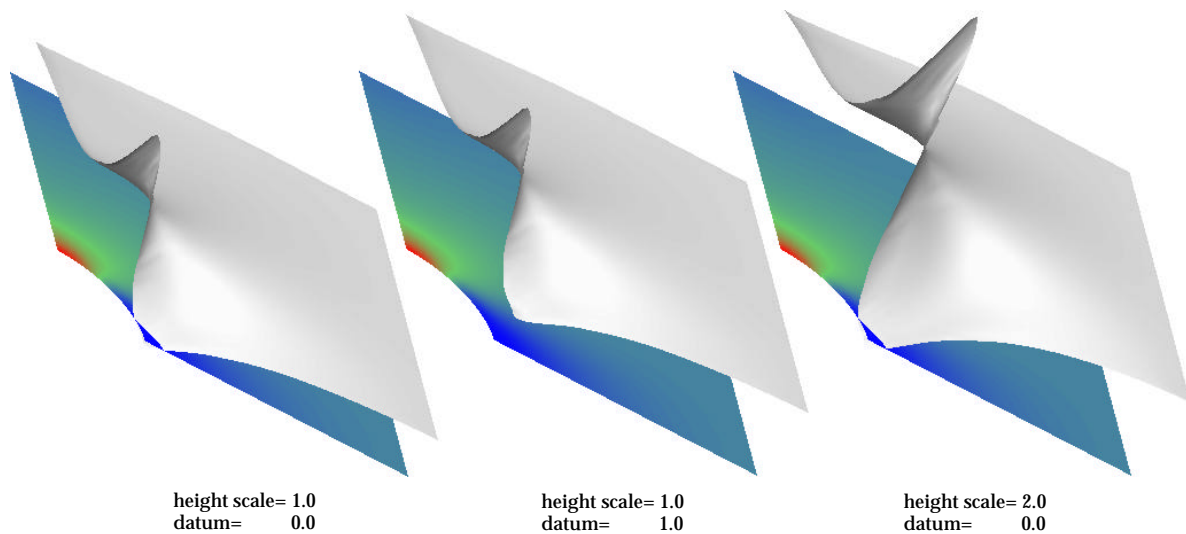
- “Wireframe”: The source plane is rendered in the form of wireframe.
- “Opaque shading”: The source plane is rendered by opaque shading.
- “Transparent shading”: The source plane is rendered by transparent shading.
- “Contour”: The source plane is rendered by a contoured surface.
- “None”: The source plane is not displayed.

#### ■ Setting the scale of height

In surface plotting, the magnitude of a data value is represented by the height of the plotted surface over its datum. The scale of height implies the height representing a unit magnitude, and is entered in the editable text box labeled as “Height Scale”.

#### ■ Setting the datum of the plotted surface

The datum of the plotted surface implies the reference from which the height of the plotted surface is measured. The datum is entered in the editable text box labeled as “Datum”. The default value of the datum is 0, and thus, the plotted surface touches the source plane at the point of zero value.



< Comparison of surface plotting with various height scale and datum >

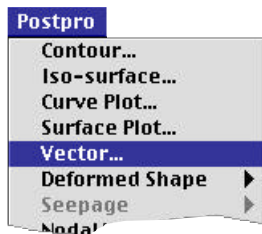
## Visualizing Vector Data

Vector data is defined as quantity with both magnitude and spatial direction.

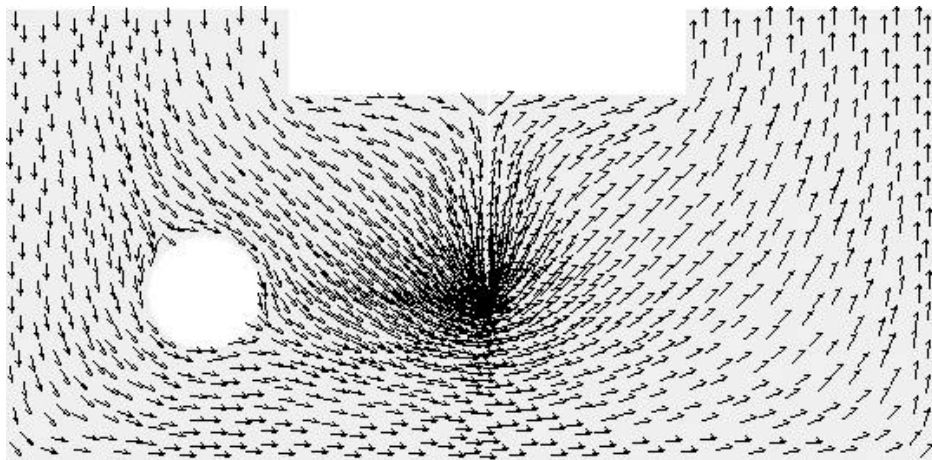
Displacement, principal direction of stresses in structural analysis, and flux direction in heat conduction analysis are typical examples of vector data.

Vector data can be visualized in various forms like arrow images, deformed shapes and force symbols, appropriate for the characteristics of the data.

### Visualizing vector data by arrows



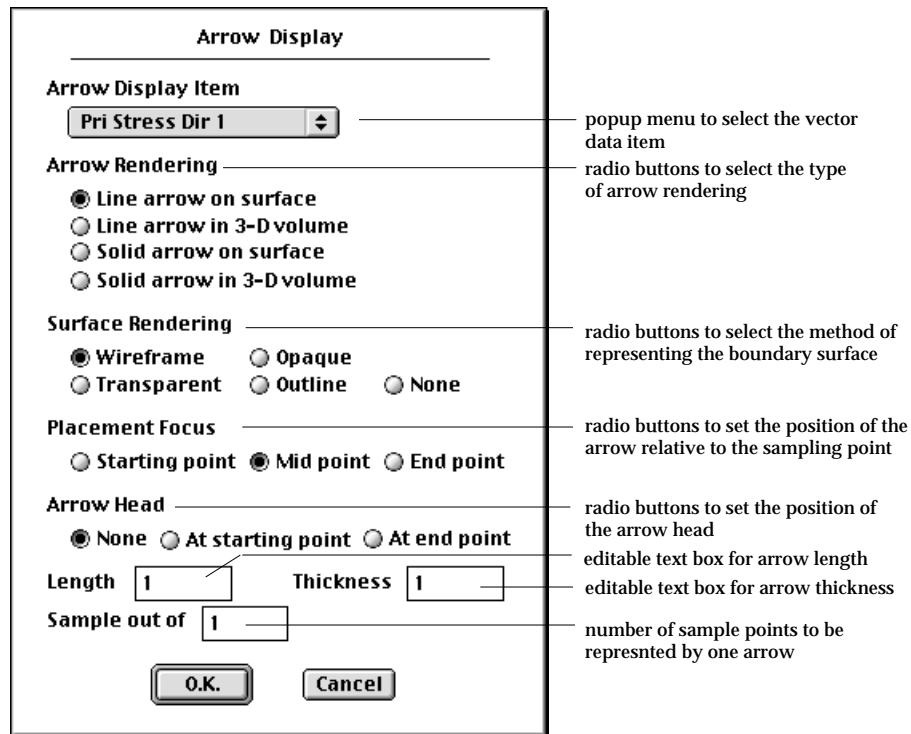
Arrows are usually used to visualize vector data. The magnitude of the data is represented by their lengths, and the direction of the data by arrow-head directions. In case of 3-dimensional solid analysis, arrows may be rendered as 3-dimensional solid objects. The following figure shows an example of ground water flow analysis, in which the flow directions are represented by arrows. In this example, an arrow indicates only the direction of the flow at the point, but not magnitude. So, the arrow length has no significance.



< Ground water flow directions represented by arrows >

### ■ Setting the arrow display options

In order to get arrow image of a vector data set, first select "Vector..." items from **Postpro** menu. Then, "Arrow Display" dialog appears on the screen as shown in the following figure. There are a number of items in this dialog. Each item has default setting. Change the setting if necessary. Click **O.K.** button if every item is set as desired. Then, the arrow image will be displayed.



&lt;"Vector Display" dialog &gt;

### ■ Selecting the data item

The popup menu in "Vector Display" dialog has the list of data items which can be displayed by arrow representation. One of the items should be selected from this popup menu. The data associated with the currently selected popup menu item is displayed by arrow image. The first item is always selected initially when the "Arrow Display" dialog is first opened. But the item used for the last arrow display will be selected automatically when the dialog is opened next.

The popup menu items vary depending on the analysis type. Only those output items appropriate for vector display appear on the popup menu. They are shown below.

Not all the items are always shown in the popup menu. If you exclude any of the output items in "Analysis Options" dialog described in Chapter 6, the corresponding item(s) will not be shown.

If you are using an external solver, the menu items may be different from those shown below, because the popup menu items can be customized by the external solvers.

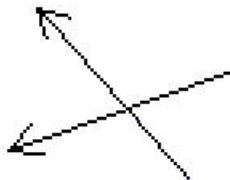
<table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Norm</td></tr><tr><td>Pri Stress Dir 1</td></tr><tr><td>Pri Stress Dir 2</td></tr><tr><td>Pri Strain Dir 1</td></tr><tr><td>Pri Strain Dir 2</td></tr></table> <p>Plane stress/strain &amp; axisymmetric</p> <table><tr><td>Flux Direction</td></tr></table> <p>2-D plane heat axisymmetric heat 3-D volume heat</p>	Displacement X	Displacement Y	Displacement Norm	Pri Stress Dir 1	Pri Stress Dir 2	Pri Strain Dir 1	Pri Strain Dir 2	Flux Direction	<table><tr><td>Displacement Z</td></tr><tr><td>Rotation X</td></tr><tr><td>Rotation Y</td></tr><tr><td>Displacement Norm</td></tr><tr><td>Princ Moment Axis 1</td></tr><tr><td>Princ Moment Axis 2</td></tr><tr><td>Princ Curvat Axis 1</td></tr><tr><td>Princ Curvat Axis 2</td></tr></table> <p>Plate bending</p>	Displacement Z	Rotation X	Rotation Y	Displacement Norm	Princ Moment Axis 1	Princ Moment Axis 2	Princ Curvat Axis 1	Princ Curvat Axis 2	<table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Z</td></tr><tr><td>Rotation X</td></tr><tr><td>Rotation Y</td></tr><tr><td>Displacement Norm</td></tr><tr><td>Pri Stress Dir 1</td></tr><tr><td>Pri Stress Dir 2</td></tr><tr><td>Pri Stress Dir 3</td></tr><tr><td>Princ Moment Axis 1</td></tr><tr><td>Princ Moment Axis 2</td></tr><tr><td>Pri Strain Dir 1</td></tr><tr><td>Pri Strain Dir 2</td></tr><tr><td>Pri Strain Dir 3</td></tr><tr><td>Princ Curvat Axis 1</td></tr><tr><td>Princ Curvat Axis 2</td></tr></table> <p>Shell</p>	Displacement X	Displacement Y	Displacement Z	Rotation X	Rotation Y	Displacement Norm	Pri Stress Dir 1	Pri Stress Dir 2	Pri Stress Dir 3	Princ Moment Axis 1	Princ Moment Axis 2	Pri Strain Dir 1	Pri Strain Dir 2	Pri Strain Dir 3	Princ Curvat Axis 1	Princ Curvat Axis 2	<table><tr><td>Displacement X</td></tr><tr><td>Displacement Y</td></tr><tr><td>Displacement Z</td></tr><tr><td>Displacement Norm</td></tr><tr><td>Pri Stress Dir 1</td></tr><tr><td>Pri Stress Dir 2</td></tr><tr><td>Pri Stress Dir 3</td></tr><tr><td>Pri Strain Dir 1</td></tr><tr><td>Pri Strain Dir 2</td></tr><tr><td>Pri Strain Dir 3</td></tr></table> <p>3-D solid</p>	Displacement X	Displacement Y	Displacement Z	Displacement Norm	Pri Stress Dir 1	Pri Stress Dir 2	Pri Stress Dir 3	Pri Strain Dir 1	Pri Strain Dir 2	Pri Strain Dir 3
Displacement X																																													
Displacement Y																																													
Displacement Norm																																													
Pri Stress Dir 1																																													
Pri Stress Dir 2																																													
Pri Strain Dir 1																																													
Pri Strain Dir 2																																													
Flux Direction																																													
Displacement Z																																													
Rotation X																																													
Rotation Y																																													
Displacement Norm																																													
Princ Moment Axis 1																																													
Princ Moment Axis 2																																													
Princ Curvat Axis 1																																													
Princ Curvat Axis 2																																													
Displacement X																																													
Displacement Y																																													
Displacement Z																																													
Rotation X																																													
Rotation Y																																													
Displacement Norm																																													
Pri Stress Dir 1																																													
Pri Stress Dir 2																																													
Pri Stress Dir 3																																													
Princ Moment Axis 1																																													
Princ Moment Axis 2																																													
Pri Strain Dir 1																																													
Pri Strain Dir 2																																													
Pri Strain Dir 3																																													
Princ Curvat Axis 1																																													
Princ Curvat Axis 2																																													
Displacement X																																													
Displacement Y																																													
Displacement Z																																													
Displacement Norm																																													
Pri Stress Dir 1																																													
Pri Stress Dir 2																																													
Pri Stress Dir 3																																													
Pri Strain Dir 1																																													
Pri Strain Dir 2																																													
Pri Strain Dir 3																																													

<Popup menu selecting the arrow display data item >

## ■ Selecting the type of arrow rendering

Arrows may be rendered in 3 different types. Select one of the types by turning on the corresponding radio button.

- “Line arrow on surface”: Arrows are drawn as lines with or without arrow head. Arrows represent the vector data on planes or surfaces.
- “Line arrow in 3-D volume”: Arrows are drawn as lines, but represent the vector data with 3-D volumes. This type is applicable only to 3-D volume models.
- “Solid arrow on surface”: Arrows are rendered as 3-D solid objects on plane or surface. Arrows represent the vector data on planes or surfaces.
- “Solid arrow in 3-D volume”: Arrows are rendered as 3-D solid objects within 3-D volume. Arrows represent the vector data on planes or surfaces. This type is applicable only to 3-D volume models.



Line arrow



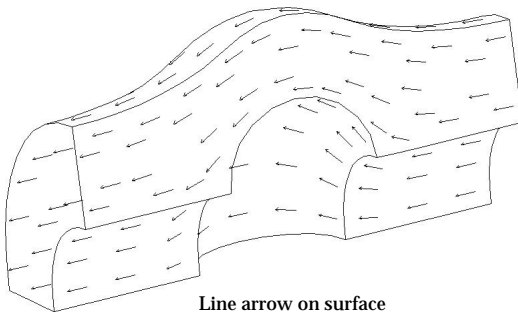
Solid arrow

< Popup menu selecting the arrow display data item >

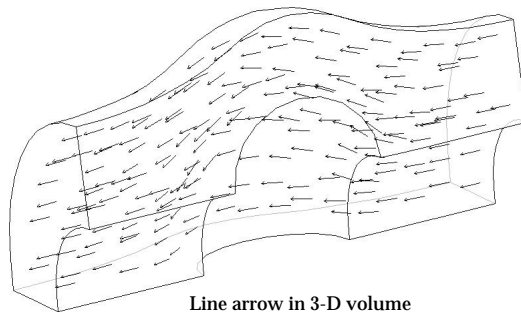
### ■ Selecting the style of surface or boundary surface rendering

The arrows are usually rendered together with the surfaces on which the arrows lie, or the boundary surfaces of the volumes containing the arrows. Select the style of the surface rendering by turning on one of the following radio buttons in “Vector Display” dialog:

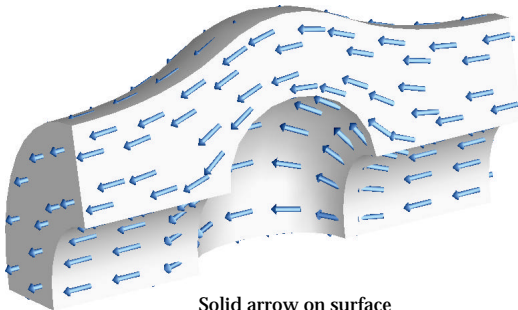
- “Wireframe”: The boundary surfaces are rendered in the form of wireframe. The hidden lines are removed from the wireframe rendering.
- “Opaque”: The boundary surfaces are rendered by opaque shading. If the type of arrow rendering is set to “Solid arrow in 3-D volume”, this option is not applicable, and thus “Opaque” button is disabled.
- “Transparent”: The boundary surfaces are rendered in transparency shading. This option is useful if the type of arrow rendering is set to “Line arrow in 3-D volume” or “Solid arrow in 3-D volume”.
- “Outline”: The outlines of the boundary surfaces are extracted, and represented together with the arrow image.
- “None”: The arrow image is displayed without boundary surface rendering. This option may be used to overlay the arrow image over the existing screen image.



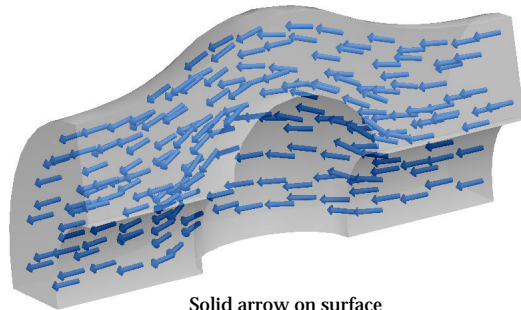
Line arrow on surface  
with outline rendering



Line arrow in 3-D volume  
with outline rendering



Solid arrow on surface  
with opaque shading



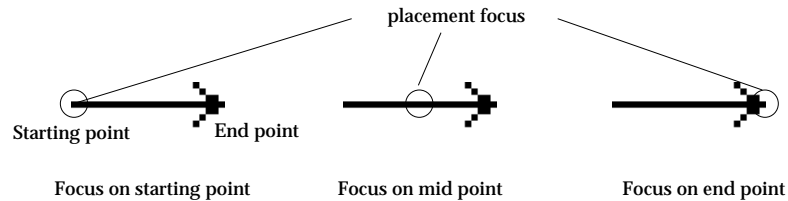
Solid arrow on surface  
with transparent shading

< Type of arrow and boundary surface rendering >

### ■ Setting the placement focus of arrows

An arrow may be placed at various positions relative to its data point. The placement focus of an arrow implies the position of the arrow, which matches with the data point, and can be set as one of the following options:

- “Starting point”: The starting point of an arrow is set as the placement focus.
- “Mid point”: The midpoint of an arrow is set as the placement focus.
- “End point”: The end point of an arrow is set as the placement focus.

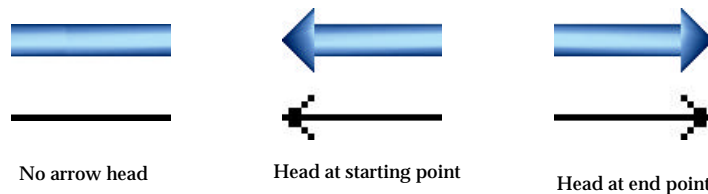


< Arrow placement >

### ■ Setting the position of arrow heads

An arrow can be drawn either with or without arrow head. At the same time, the arrow head may be attached at the starting point or the ending point of the arrow. Thus, there are the following 3 options related to arrow heads:

- “None” : Arrow heads are not drawn. An arrow is represented by a straight line. This option is appropriate for the case in which the sign of the direction is not important. For example, the directions of principal stresses are usually drawn without arrow heads.
- “At starting point” : The arrow head is attached to the starting point of the arrow. The arrow directs from the end point toward the starting point. This option is used to draw the arrow in the directions reverse to the actually computed ones.
- “At end point” : The arrow head is attached to the end point of the arrow. The arrow directs from the starting point toward the end point. The vector directions are normally represented by this option.

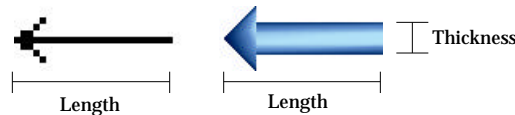


< Arrow head position >

### ■ Setting the length of arrows

The length of an arrow may or may not represent the magnitude of its corresponding data value, depending on whether the corresponding vector data have magnitude or not. For example, displacements in structural analysis have magnitudes as well as directions. But, there are such vector data as principal stress directions for example, in which the data has only direction, not magnitude. In either case, the length of the arrow is determined on the basis of the predefined length factor. The length factor is defined by entering its value in the editable text box labeled “Length” in the dialog. The entered value should be greater than 0, and represents a factor determining the arrow length.

The length is independent of the view scale of the model. Thus, the length of arrow remains unchanged even if the screen scale is altered.



< Length and thickness of arrow >

### ■ Setting the thickness of arrows

This item is effective only for “Solid arrow in 3-D volume” option. The length of the arrow rendered as 3-D solid is determined on the basis of the predefined thickness factor. The thickness factor is defined by entering its value in the editable text box labeled “Thickness” in the dialog. The entered value should be greater than 0, and represents a factor determining the arrow thickness.

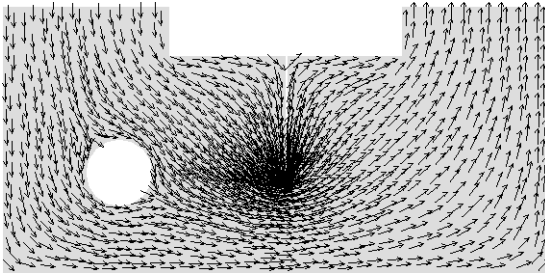
The thickness is independent of the view scale of the model. Thus, the thickness of arrow remains unchanged even if the screen scale is altered.

### ■ Displaying selectively sampled arrows

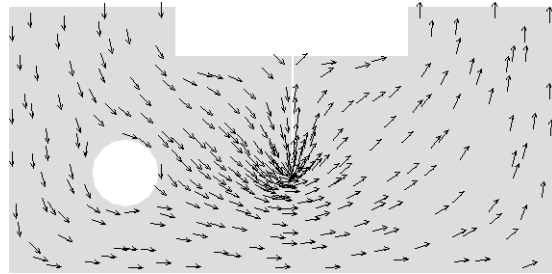
One arrow is usually displayed at every data point, which may be a node or a point in an element. As the model becomes complex, the number of data points increases. Sometimes, there are too many arrows crowded around a point to visualize the data properly. Such complexity in visualization can be alleviated by displaying the arrows partially.

The proportion of arrows for selective display is designated by the value entered in the editable text box labeled “Sampling out of” in the dialog. If the value is 1, one arrow is displayed at every data point. In other words, all the arrows will be displayed. If it is 5 for example, arrows are displayed only at one out of 5 data points.





One arrow displayed at every data point

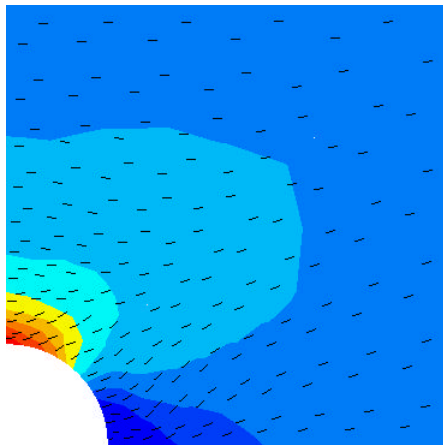


One arrow displayed at every 5 data point

< Displaying selectively sampled arrows >

### ■ Overlaying vector images

It is sometimes necessary to overlay two or more vector images. For example, the directions of two principal stresses,  $\sigma_1$  and  $\sigma_2$ , are desired to be displayed together in order to improve the visual understanding of the computed results. In this case, the directions of  $\sigma_1$  are first drawn, and those of  $\sigma_2$  are overlaid over the existing image. Such overlaying vector images can easily be achieved by using “None” option for surface rendering. This option adds the new vector image over the previously drawn image.



The contour image of  $\sigma_1$  is overlaid  
the arrow image of their directions

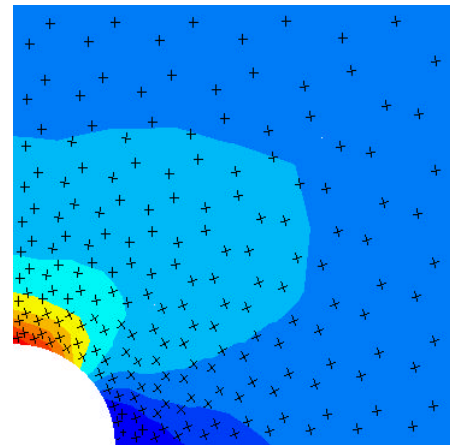
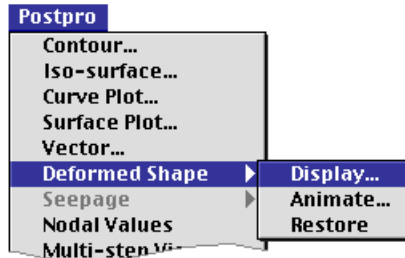


Image of  $\sigma_2$  direction is added.

< Overlaying vector images >

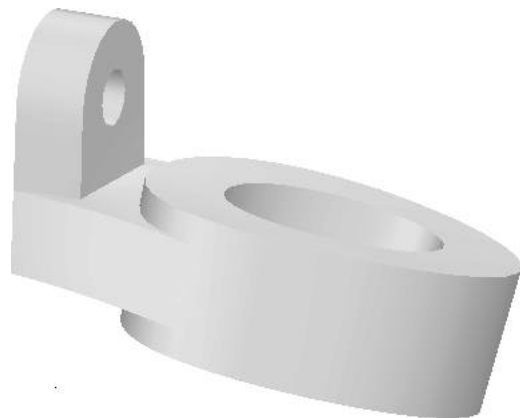
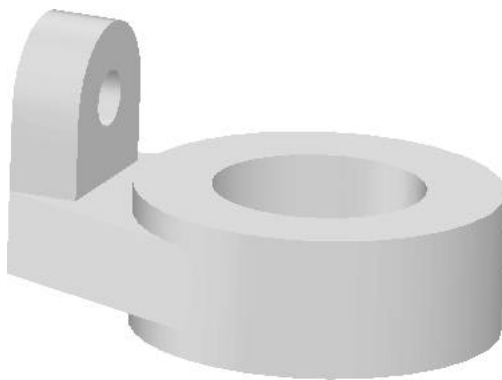
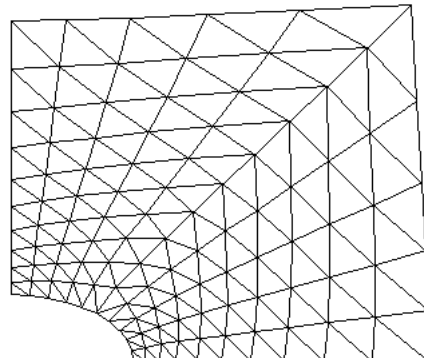
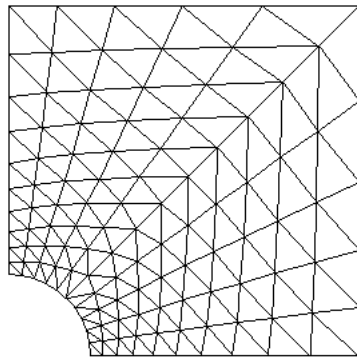
### Visualizing displacements by deformed shape and animation



Displacements obtained from structural analysis are another typical example of vector data. As other vector data are represented by arrows, so can displacement data. However, we may get better insight on the displacements from the image of deformed shape than from arrows. All the displacement components are incorporated in one image of deformed shape. The nodal displacements are added, with a specified magnification factor, to the nodal coordinates to form new nodal coordinates which represent the exaggeratedly deformed model.

Once the deformed shape is displayed, all the subsequent visualization of the model is done using the deformed nodal coordinates. However, a few operations including saving file, editing model data and solving, will automatically restore the undeformed original shape.

This function is applicable only to structural analysis.



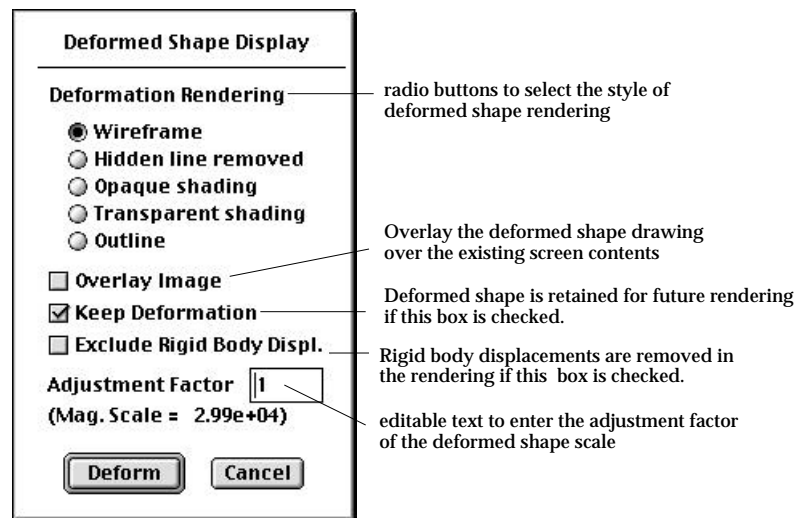
Undeformed shape

Deformed shape

< Deformed shape of a planar structural model >

### ■ Setting the display options

In order to get a deformed shape, first select "Display..." item from "Deformed Shape" submenu of **Postpro** menu. Then, "Deformed Shape Display" dialog appears on the screen as shown in the following figure. There are a few items in this dialog, including the type of rendering. Each item has a default setting or an initially suggested value like deformation scale. Change the setting if necessary. Click **Deform** button if every item is set as desired. Then, the deformed shape will be rendered as specified in the dialog. More details of setting the dialog is described in the following.



< "Deformed Shape Display" dialog >

### ■ Selecting the style of the deformed shape rendering

The deformed shape can be rendered in 4 different styles.

- "Wireframe": The deformed shape of the structural model is rendered in the form of wireframe without removal of hidden lines.
- "Hidden line removed": The deformed shape of the structural model is rendered in the form of wireframe with removal of hidden lines.
- "Opaque shading": The deformed shape of the structural model is rendered by opaque shading.
- "Transparent shading": The deformed shape of the structural model is rendered by transparent shading.
- "Outline": The deformed shape of the structural model is rendered by outline.

### ■ Overlaying the deformed shape image over the screen image

The deformed shape is drawn over the current image on the main window, if the "Overlay image" box is checked.

### ■ Retaining the deformed shape in future rendering

The deformed shape may or may not be retained in the future rendering. If "Keep Deformation" box is checked, the deformed shape is used in the future rendering such as contouring. If you transform the view or change the rendering method, the rendering is updated with deformed shape. Otherwise, undeformed shape is used in the future rendering.

### ■ Excluding the rigid body displacements from deformed shape

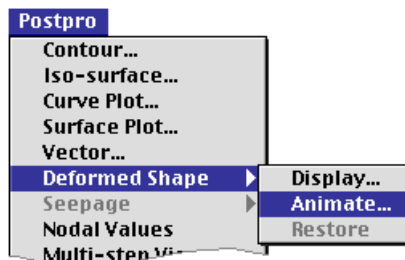
In case only a part of the model is to be visualized with its deformed shape, the displacements of other parts may sometimes dominate the overall picture. Thus the deformation of the interested part may be obscured by that of other parts. The portion of the displacements originated from other parts can be regarded as rigid body displacements for the displayed part, and therefore may be excluded from the deformation so that the image of the deformed shape is better scaled and positioned.

Check the "Exclude rigid body displ." box in the dialog in order to remove the unwanted portion of the displacements.

### ■ Setting the deformation scale

The displacements are usually very small as compared with the overall size of the structural model. Therefore, the displacements should be magnified to such a scale that their effects are discernible in the deformed shape. The deformation scale is the magnification factor that multiplies the displacements before adding them to the nodal coordinates. The deformation scale appropriate for display is initially determined by the software, and is shown in the dialog like "Mag.Scale=2.99e+04". The editable text item "Adjustment Factor" is provided for user's control of the displacement scale. The value is multiplied to the displacement scale and can be altered by editing the string in the editable test box. The altered value is applied in the subsequent display of the deformed shape.

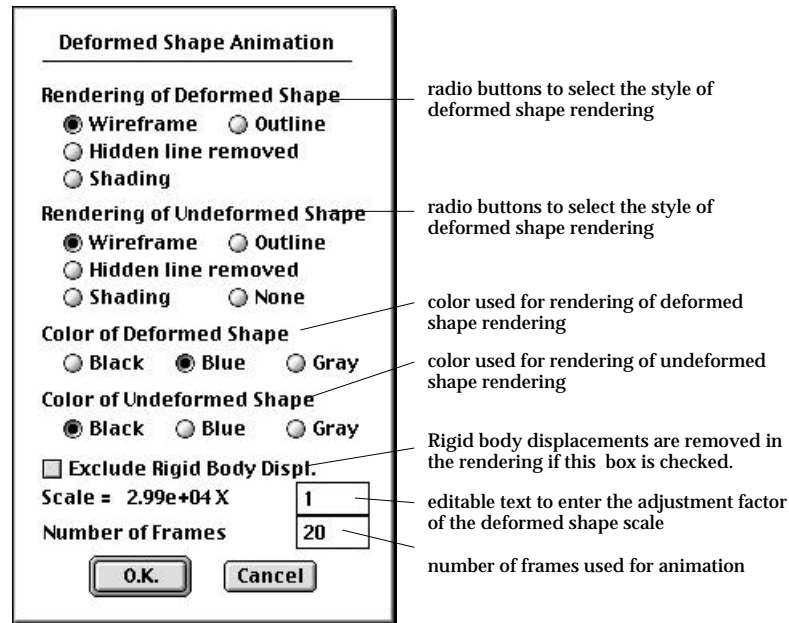
### ■ Visualizing the displacements by animation



The displacements obtained from the analysis can be visualized by animation. In order to get a deformed shape, first select "Display..." item from "Deformed Shape" submenu of **Postpro** menu. Then, "Deformed Shape Display" dialog appears on the screen as shown in the following figure.

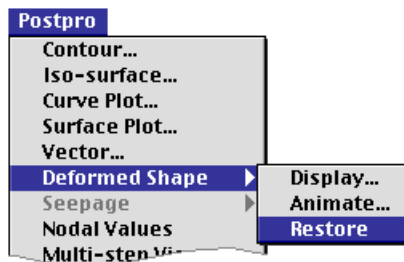
The model is animated by the images composed of frames from the undeformed shape and the deformed shape. The animation can be shown with or without overlaid image of the original shape.

The methods of rendering can be set independently for the deformed shape and the undeformed shape. The number of animation frame can be set using the editable text box in the dialog.

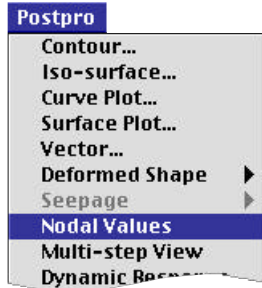


### ■ Restoring undeformed shape

Once the deformed shape of a structural model is displayed, that shape is maintained and used for subsequent data visualization like contouring, if "Keep Deformation" option is used. In this case, the undeformed shape is restored by selecting "Restore" item from "Deformed Shape" submenu of **Postpro** menu.



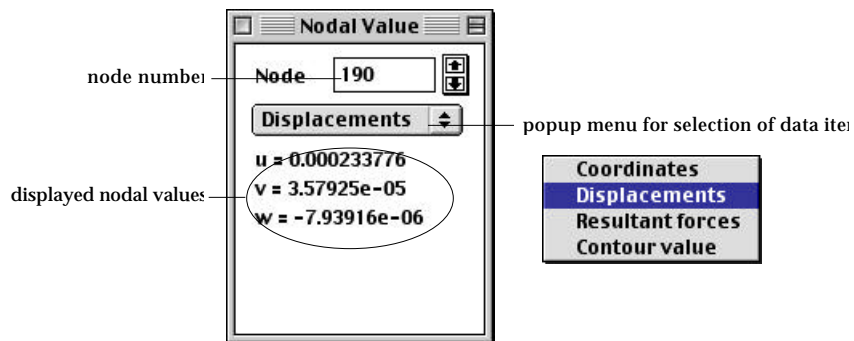
## Getting numerical values of nodal data



In order to obtain the numerical values at a node, select "Nodal Values" item from **Postpro** menu. Then, "Nodal Values" dialog pops up as shown below. Select a node and set the desired data item on the dialog. Then, the corresponding data at the selected node is displayed on the dialog.

A node is selected by clicking it. A node can also be selected by inserting the node number or by scroll button beside the node number text.

Nodal values of a contoured data can be obtained only when contour image of the data is displayed on the screen.



## Visualization of Multi-step Analysis Data

Nonlinear, dynamic, sequentially staged or adaptive solution is obtained by multi-step analysis, while linear static one is done by one-step analysis. Data obtained by multi-step analysis have information for each step of analysis. The data of intermediate steps may or may not have important meaning, depending on the type of analysis, or user's interest in the intermediate results.

For dynamic analysis, the intermediate steps correspond to time steps involved in the analysis, and thus dynamic behavior of the system is represented by the data sets obtained for time steps.

Nonlinear analysis may also involve more information than linear analysis in conjunction with its intermediate steps of load increment. The nonlinear behavior of the system can be observed by examining the data obtained at various levels of load increment.

As for adaptive analysis, data obtained at the intermediate steps may either show progressive behavior of the system or suggest numerical characteristics of the solution.

Thus, visualization of intermediate steps may be important not only for dynamic analysis but also for nonlinear, or adaptive analysis.

Additional capabilities of postprocessing are required for better treatment of multi-step analysis data as described below.

### Stepwise rendering of multi-step analysis results

The results obtained from multi-step analysis can be postprocessed in the same forms used for single step analysis, i.e., contour, vector, diagram, and so on. In case of multi-step analysis, however, data for more than 2 steps cannot be visualized at once. They should be expressed as sequential sets of data by employing step-by-step visualization.

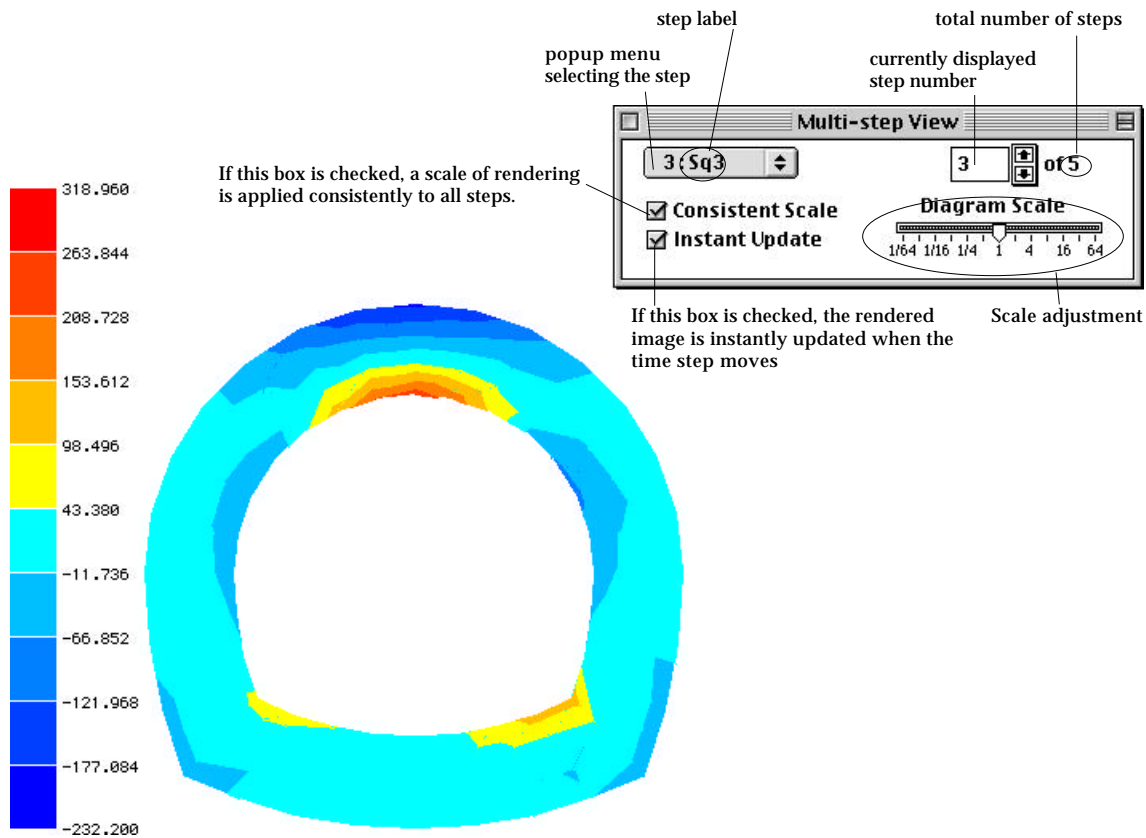
The transient behavior of a dynamic system, for example, can be expressed graphically by stepwise rendering of multi-step data. Nonlinear response of the system to load increment can also be represented by sequential visualization of data obtained at intermediate steps



#### ■ Stepwise rendering of multi-step analysis results

Stepwise rendering of multi-step analysis results can be initiated and controlled by using "Step Control Panel" dialog. The dialog pops up when you select "Multi-step View" item from **Postpro** menu.

The time step can be moved forward or backward using the control buttons, or specified directly by editing the step text box as described below. However, the image on the main window will not be updated when the time step is altered, unless "Instant Update" box is checked.



< Stepwise rendering of multi-step analysis >

- **Popup menu (Windows: dropdown list) :** The popup menu is used to set directly the desired step to the current step. The popup menu items indicate the step numbers and the step labels.
- **Step number text :** The step number text indicates the currently rendered solution step like **Step 18**. The number can be altered directly editing the text in the box, or by using scroll buttons. As soon as the number is changed, the image rendered on the main window is updated accordingly, if "Instant Update" text box is checked. Otherwise, the image will not be updated even if the number is altered.

The step number should be less than or equal to the total number of solution steps included in the multi-step analysis.

- **"Consistent Scale" check box :** If this check box is checked, a consistent scale is applied to all steps in rendering contours, deformed shape, and so on. Otherwise, independent scale is applied to each of the step.
- **"Instant Update" check box :** This check box is used to turn on or off the state of instant update by which the rendered image is renewed whenever the step number is altered.



### ■ Selecting the method of stepwise rendering

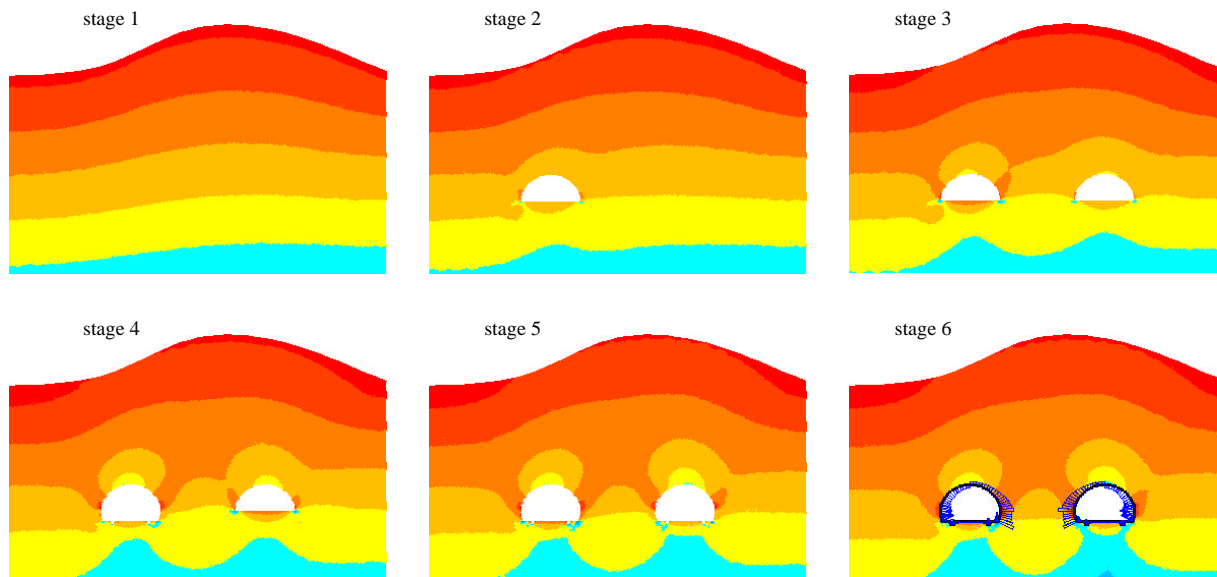
As already described above, various methods of rendering can be employed to represent multi-step analysis results. The method of stepwise rendering coincide with the method applied prior to initiating the stepwise rendering. As an example, if you start stepwise rendering after creating a contour image of a normal stress component, then all the subsequent images will be rendered by contours of the same stress component as long as "Step Control Panel" dialog is on the screen.

### ■ Ending stepwise rendering

Stepwise rendering mode is maintained while "Step Control Panel" dialog is on the screen, and can be terminated by closing the dialog or by starting any other menu command. The step number remains

### Stepwise rendering of sequentially staged analysis results

A sequentially staged analysis is also a kind of multi-step analysis, and accordingly its analysis results can be visualized and examined interactively by using multi-step view. The following figure shows an example of multi-step view of a sequentially staged analysis results with 6 stages. Consistent scale option is used in this example.



< Multi-step view of sequentially stage analysis results >

## Visualization of Dynamic Analysis Data

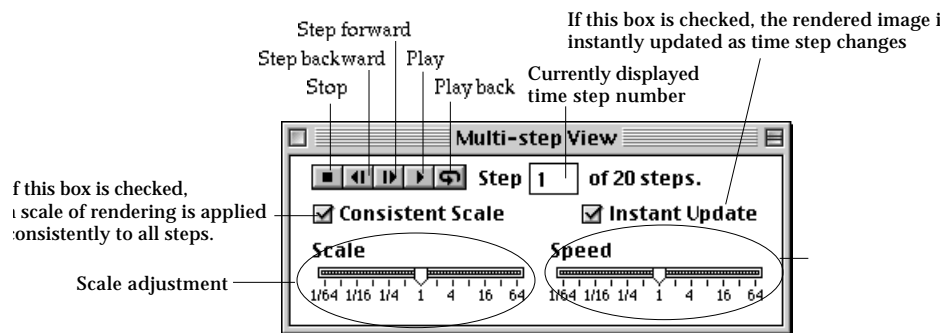
A dynamic analysis produces various data such as displacements, strains and stresses for each of the time steps involved in the solution process. The dynamic responses can be represented as functions of time on the basis of these data. The responses can be expressed using different coordinates, i.e., actual spatial coordinates and modal coordinates. Plotting in modal coordinates is available only when the solution is obtained by the method of mode superposition.

### Visualizing dynamic response

The dynamic analysis results can be visualized by multi-step view as described in the preceding section. On the other hand, these data can also be organized into transient variation, i.e. time history of a certain response in the dynamic system.

#### ■ Visualizing dynamic analysis results using multi-step view

Multi-step view is used in visualizing the dynamic analysis data along the time steps. Select "Multi-step View" item from **Postpro** menu. Then, "Multi-step View" dialog appears on the screen. Using the control buttons of this dialog, the analysis data can be rendered step by step,

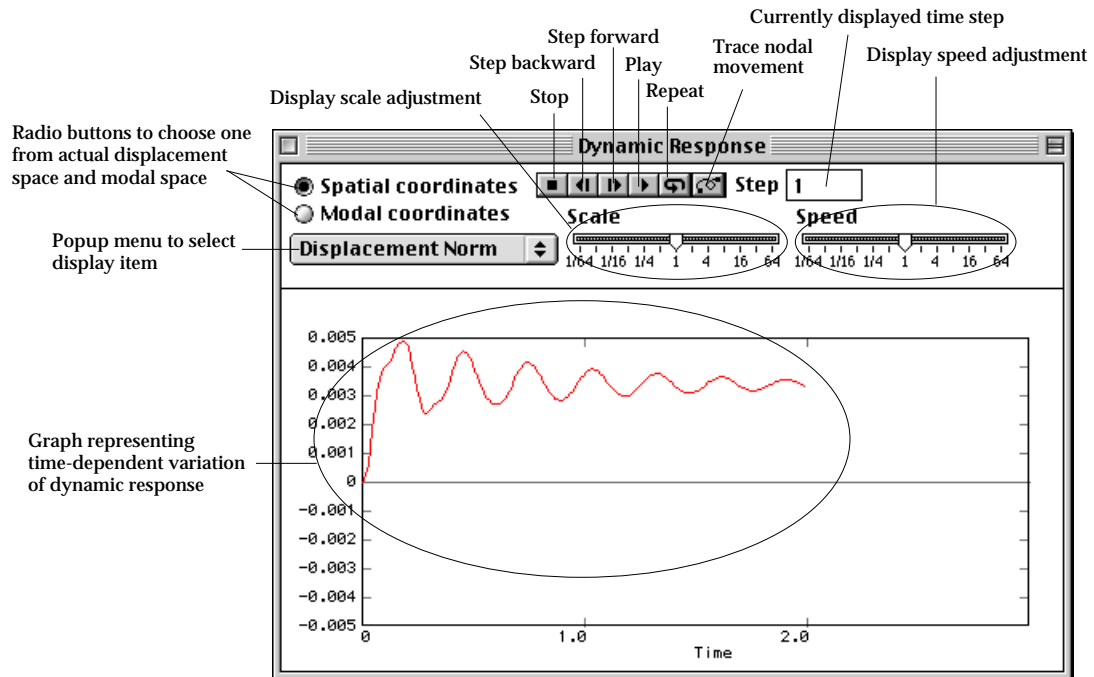


#### ■ Visualizing dynamic analysis results in time history form










Time history can be represented by plotting the transient variation with respect to elapse of time. In addition, stepwise rendering of dynamic motion paired with time history plot will give better visual effect.

"Dynamic Response" window is a kind of base for visualizing dynamic response as a function of time. In order to open the window, select "Dynamic Response" item from **Postpro** menu. "Dynamic Response" window has the upper part with various controls and the lower part with area of plotting time history data. Stepwise dynamic motion can be animated in association with the time history plot using the functions described below.



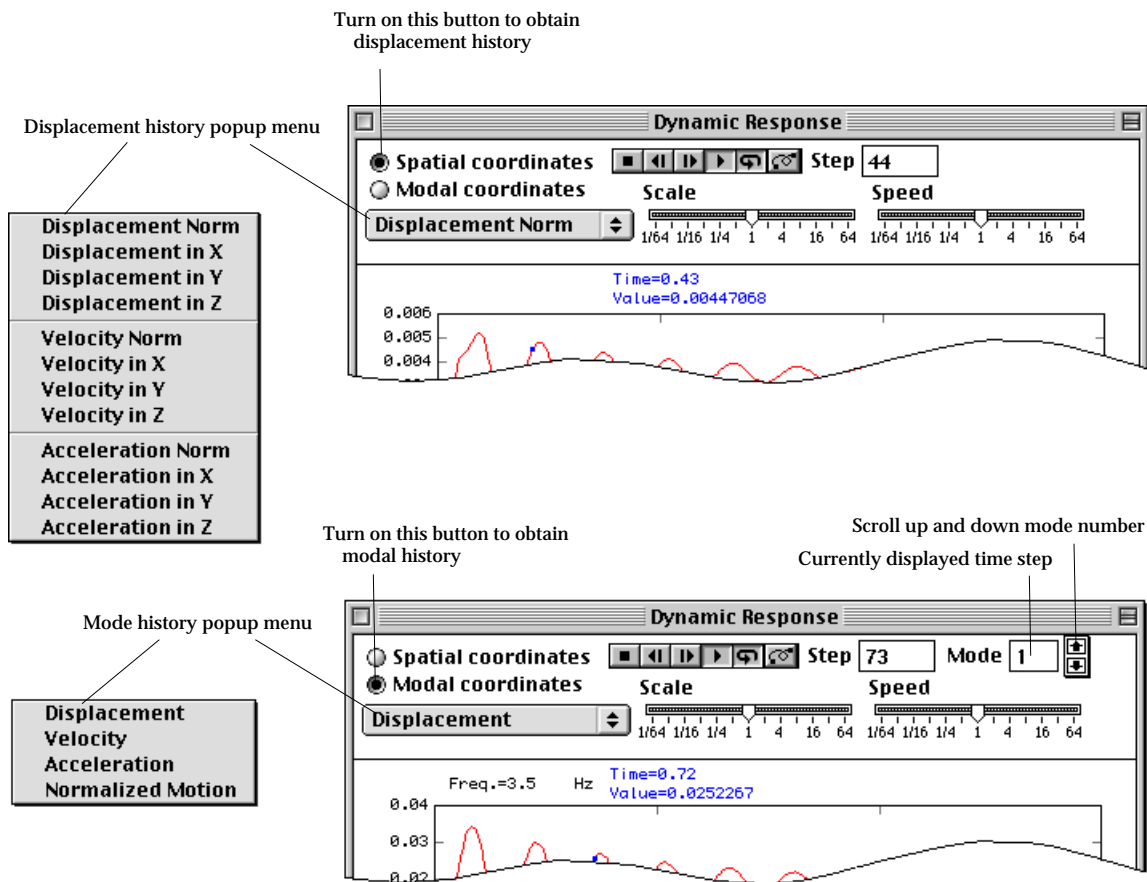


< An example of time history plot >

- Control buttons:
  - Stop button  : Click this button to stop sequential stepwise rendering, i.e., animation.
  - Step backward button  : Click this button to move one step backward in stepwise rendering. If animation is running, it will be terminated, and play button automatically pops up.
  - Step forward button  : Click this button to move one step forward in stepwise rendering. If animation is running, it will be terminated, and play button automatically pops up.
  - Play button  : Click this button to start animating stepwise rendering. Then, this button looks pressed like , stepwise animation goes on.
  - Repeat button  : If this button is not pressed, animation terminates at the last step. But, if this button is pressed like , animation continues backward, when the animation reaches the frame of the last step.
  - Trace button  : If this button is pressed like , the trace of nodal movement is recorded and displayed while animation continues.
- Step text box : This editable text item shows the currently displayed time step. The time step can be moved directly by editing this editable text.
- Mode text box : This editable text item shows the number of the currently

displayed dynamic mode. The dynamic mode can be altered directly by editing this editable text, or by using scroll button to the right of this text box. This text item is visible only when "Modal coordinates" radio button is turned on.

- Slide control of scale : The scale of dynamic motion or dynamic mode is adjusted by using this control.
- Slide control of speed : The speed of animation is adjusted by using this control.
- Radio buttons: There are 2 radio buttons to set the type of time history.
  - "Spatial coordinates" : The time history is expressed directly by nodal values of displacements, velocity and acceleration.
  - "Modal coordinates" : The time history is expressed by linear combination of modal coordinates for displacements, velocity and acceleration.
- Popup menu: The menu consists of display items. The contents of the time history plot are altered by changing the menu item.



<Selection of coordinates for plotting dynamic response>

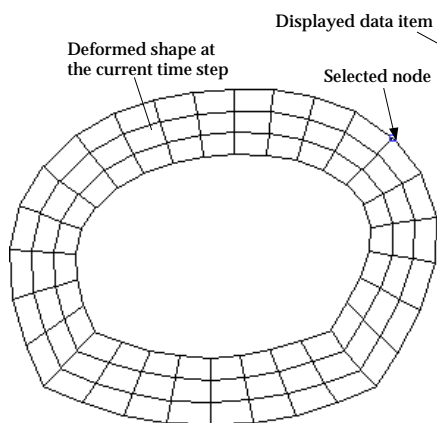
### ■ Plotting time history records of nodal value in spatial coordinates

In case "Spatial coordinates" radio button is on, a nodal value changing as a function of time is plotted on "Dynamic Response" window. The nodal value represents one of displacement, velocity, acceleration, and their components which have been recorded for each time step of dynamic analysis.

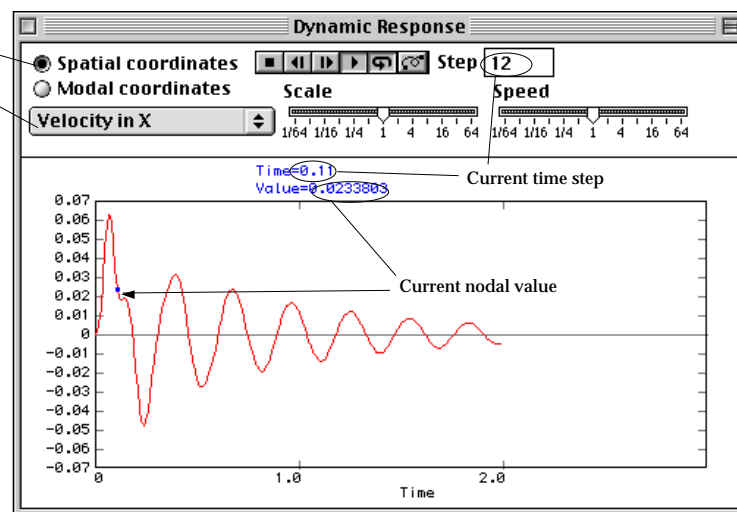
The following is the procedure of visualizing the time history of a nodal value.

- 1) Select "Dynamic Response" item from **Postpro** menu.  
Time history window pops up.
- 2) Turn on "Spatial coordinates" radio button, if it is not.  
Initially this radio button is on by default. While this button is on, the popup menu contains the items relevant to displacement time history.
- 3) Select the node whose record is to be displayed.  
The node for time history plot is not specified at the beginning, and thus "Dynamic Response" window show the message "Select the node for time history record". At the moment a node is selected from the model, the time history record of the node is plotted on "Dynamic Response" window.
- 4) Select the popup menu item to change the displayed data item.  
The popup menu contains following items which represent the types of the data record.

Turn on this button to display the nodal values directly.



Displayed data item



<Time history plot in spatial coordinates>

Displacement Norm
Displacement in X
Displacement in Y
Displacement in Z
Velocity Norm
Velocity in X
Velocity in Y
Velocity in Z
Acceleration Norm
Acceleration in X
Acceleration in Y
Acceleration in Z

- "Displacement Norm" : magnitude of displacement.
- "Displacement in X" : displacement component in X direction.
- "Displacement in Y" : displacement component in Y direction.
- "Displacement in Z" : displacement component in Z direction.
- "Velocity Norm" : magnitude of velocity.
- "Velocity in X" : velocity component in X direction.
- "Velocity in Y" : velocity component in Y direction.
- "Velocity in Z" : velocity component in Z direction.
- "Acceleration Norm" : magnitude of Acceleration.
- "Acceleration in X" : Acceleration component in X direction.
- "Acceleration in Y" : Acceleration component in Y direction.
- "Acceleration in Z" : Acceleration component in Z direction.

The time history record of the specified data item at the selected node is plotted on "Dynamic Response" window. The plot has horizontal axis for time and the vertical axis for magnitude of nodal value. The scales of horizontal and vertical axes are determined automatically so that the plotted graph makes the best of the space on "Dynamic Response" window.

### ■ Plotting time history records in modal coordinates

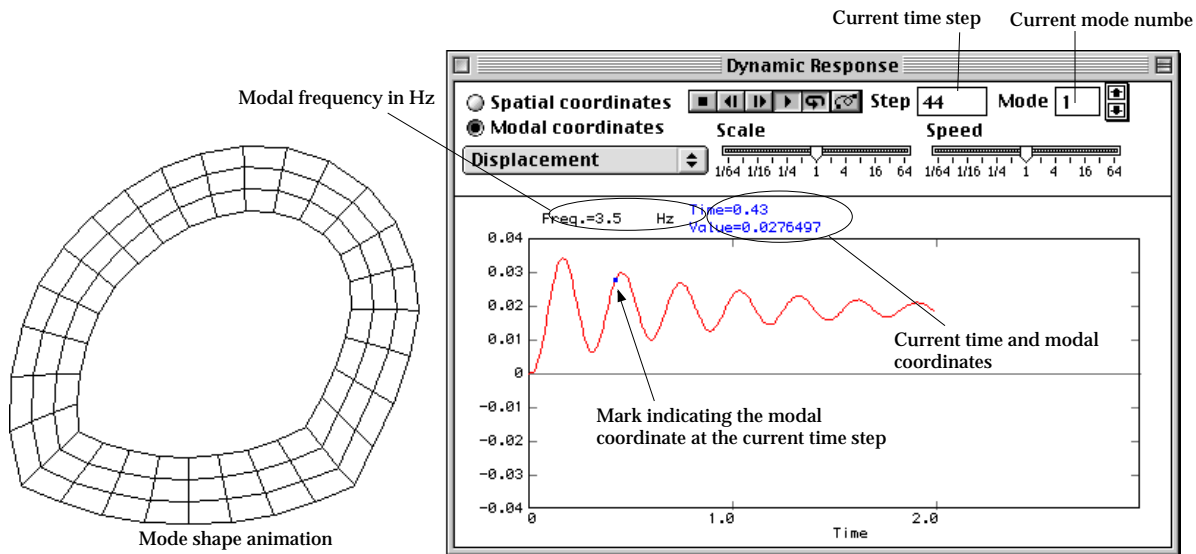
In case the dynamic solution is obtained by the method of mode superposition, the modal coordinates are recorded for each time step of the analysis. The record of the modal coordinates can be plotted against time by the following steps:

- 1) Select "Dynamic Response" item from **Postpro** menu.  
Time history window pops up.
- 2) Turn on "Modal coordinates" radio button, if it is not.  
If this button is on, the popup menu contains the items relevant to mode history, and the mode number text box **Mode 1** is shown at the top right corner of "Dynamic Response" window.
- 3) Set the number of the mode to display.  
The mode number can be set by directly editing the text in the mode number text box or using the scroll button.
- 4) Select the popup menu item to change the displayed data item.  
The popup menu contains following items which represent the types of modal coordinates.

Displacement
Velocity
Acceleration
Normalized Motion

- "Displacement" : modal coordinates for displacement.
- "Velocity" : modal coordinates for velocity.
- "Acceleration" : modal coordinates for acceleration.
- "Normalized Motion" : normalized periodic motion of a mode.

While the plotting of modal coordinates is shown on "Dynamic Response" window, the motion of the corresponding dynamic mode is displayed on the main window.



&lt;Dynamic response plot in modal coordinates&gt;

### ■ Animation of dynamic motion

The dynamic motion of the model can be animated in coupling with the nodal value plotted on "Dynamic Response" window. The animation is initiated by clicking the play button of the window, and proceeds one frame for every time step while the button is in pressed state . The animation is terminated when the frame moves to the last time step, and the play button returns to unpressed state. If the playback button is in pressed state , the animation continues with the time frame progressing forward and backward alternately.

While animation is running on the main window, the nodal value plot corresponding to the currently displayed animation frame is marked by a small square mark on "Dynamic Response" window.

In case of mode history plotting, the motion of a dynamic mode is animated in coupling with the modal coordinates plotted on "Dynamic Response" window.

### ■ Displaying the trace of nodal movement

If trace display button is in pressed state , the trace of a nodal movement is recorded and displayed while the animation is going on. In order to remove the trace, turn the button to unpressed state by clicking the button.

### ■ Changing the time step

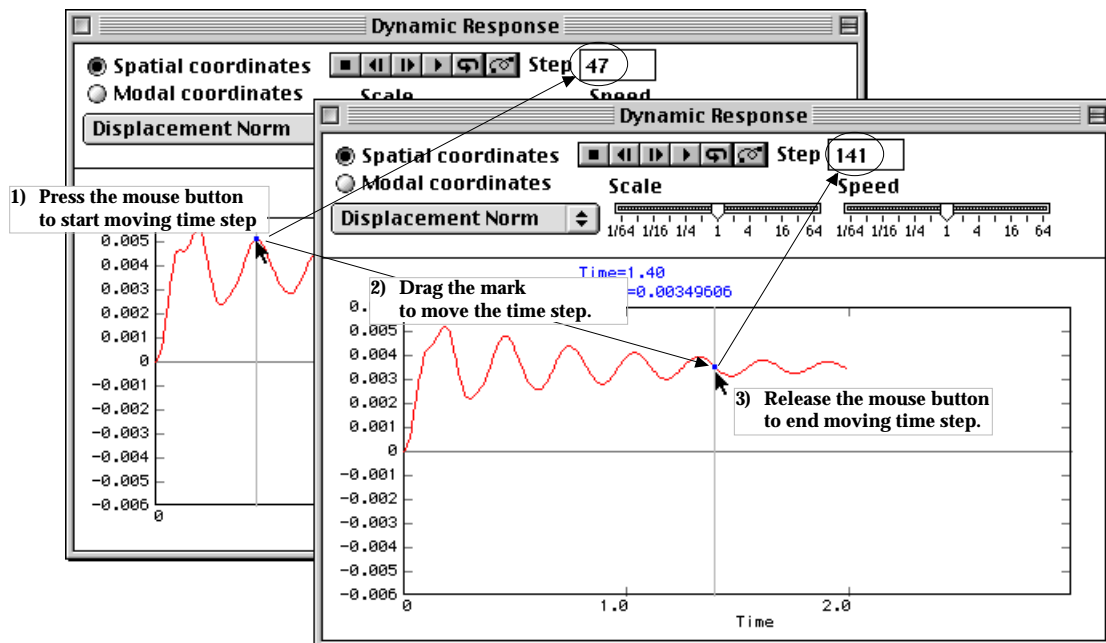
While animation of dynamic motion is going on, time also progresses and the time step number is reflected in the step text box. The time step can also be changed by using step forward button or step backward button .

The time step can also be changed by editing directly the number in the step text box. As soon as the step number is altered, the image of dynamic motion is updated to the frame of corresponding time step.

### ■ Dragging the time step

The time step representing the currently displayed animation frame can be moved forward or backward by dragging the square mark on the nodal value plot. The current time step changes along with the movement of the mark. This action can be achieved by the following procedure.

- 1) Place the cursor over the square mark and press the mouse button.  
A vertical line is drawn to indicate that dragging is started. Animation is paused, if it is running.
- 2) Drag the mark with mouse button pressed.  
The vertical line as well as the mark moves along with the mouse movement. The time step also changes as the mark moves. The image of the model on the main window is also updated in accordance with the motion at the time step.
- 3) Release the mouse button.  
Dragging is completed, and animation is resumed if it was running before the start of dragging. Depending on the direction of dragging, the progress of time is also altered either into forward or backward direction.



<Moving the time step by dragging>



### ■ Adjusting the animation speed and the deformation scale

Dynamic motion or dynamic mode is animated with the speed and the deformation scale optimally determined by the software. This speed and the scale can be adjusted using the slide controls provided on "Dynamic Response" window.

### ■ Resizing "Dynamic Response" window

"Dynamic Response" window can be resized by dragging size box(Mac OS) or window border lines(MS Windows). A subsequent time history image is drawn to fit the resized window.

### ■ Ending time history display




Time history display mode is going on while "Dynamic Response" window is open, and can be terminated by closing the window or by starting other menu command.

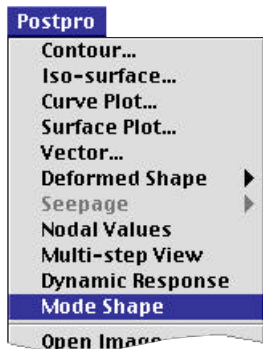
## Visualizing dynamic mode shape

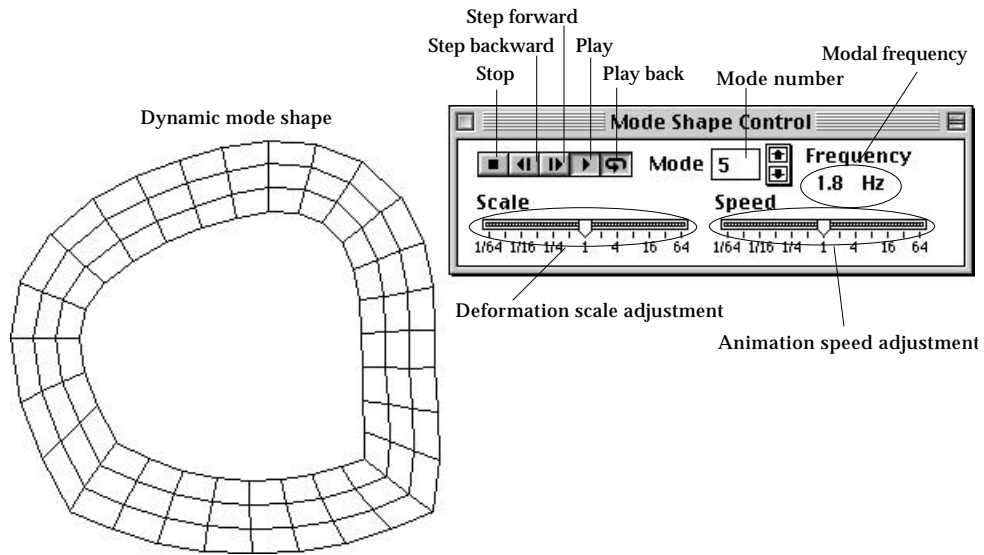
The dynamic motion can be decomposed into a number of dynamic modes, which are extracted by eigenvalue analysis of the dynamic system. Data related to dynamic modes are also important ingredients of dynamic analysis. The shape of dynamic modes can be represented by deformed shape or animation. In order to visualize dynamic modes, the system should be solved with "Mode superposition" or "Modal analysis" option in "Dynamic Analysis Options" dialog.

### ■ Starting and controlling mode shape display

"Mode Shape Control" dialog is used to control the display of mode shape. Select "Mode Shape" item from **Postpro** menu to open the dialog. The dialog has the same control buttons as "Step Control Panel" dialog described in the preceding section.

The mode shape is animated when play button  is pressed. The number of frames for animation is determined by the software regardless of the number of time steps involved in the analysis. The animation frame can be moved forward or backward one by one using forward button  or backward button  respectively.





&lt;Display of dynamic mode shapes&gt;

### ■ Selecting the mode to display

The dynamic modes are numbered in ascending order of their frequencies, i.e., smaller number for lower frequency and larger number for higher frequency. The mode for display is selected by its number. The number of the currently displayed mode is shown in the text box of the dialog. The number can be picked by directly editing the text or by using scroll buttons. As soon as the mode number is altered, the shape of the corresponding mode is displayed on the main window, and the frequency of the mode is indicated on "Mode Shape Control" dialog.

### ■ Scale of display

The mode shape is displayed with the deformation scale optimally determined by the software. The scale can be adjusted by using a slide control of the dialog.

### ■ Speed of animation

The speed of animation, i.e., the number of frames proceeding per second is initially determined by the software. The speed can be adjusted by using a slide control of the dialog.

### ■ Ending dynamic mode display

Dynamic mode display mode is going on while "Mode Shape Control" dialog is open, and can be terminated by closing the window or by starting another menu command.

## Visualization of Combined Load Case Results

Multiple load cases can be defined for a structural model as described in Chapter 5. Multiple sets of analysis results are obtained by solving such a model. You may easily and conveniently visualize not only the analysis results of the supplied load cases but also create new results by arbitrarily combining the load cases, using the multi-loading view function of this software.

### ■ Characteristics of the analysis results with multiple load cases

The multiple load solution is only possible for linear static analysis based on the assumption that the computed displacements are linearly proportional to the magnitude of the applied loads. If there are  $n$  load cases, the same number of displacement vectors  $(\mathbf{u}_1, \mathbf{u}_2, \dots, \mathbf{u}_n)$  are obtained respectively from the corresponding force vectors  $(\mathbf{F}_1, \mathbf{F}_2, \dots, \mathbf{F}_n)$ . Each of the force-displacement relations is expressed with an identical stiffness matrix  $\mathbf{K}$ .

$$\mathbf{K} \mathbf{u}_i = \mathbf{F}_i \quad (i = 1, \dots, n)$$

The stiffness matrix is decomposed only once, and back substituted for each of the force vectors to yield the corresponding displacement vector. Further combination of load cases is valid due to the linear force-displacement relationship. Let the force vector due to a combination of  $n$  load cases be,

$$\mathbf{F}_c = \sum_{i=1}^n c_i \mathbf{F}_i$$

Then the resulting displacement vector can be obtained simply by the same linear combination,

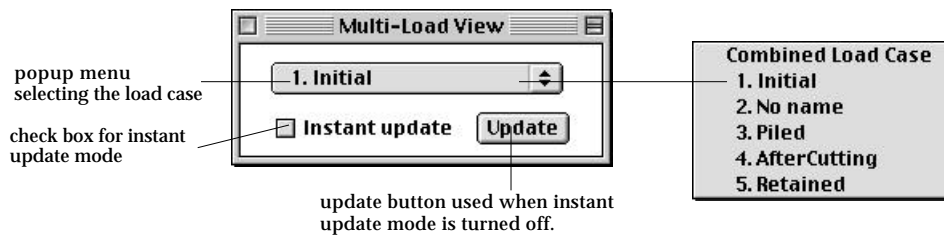
$$\mathbf{u}_c = \sum_{i=1}^n c_i \mathbf{u}_i$$

It is based on such linearity that arbitrarily combined analysis results can be derived from the results of the multiple load cases using the multi-loading view described below.

### ■ Starting multi-loading view



Select "Multi-loading View" from **Postpro** menu in order to start the function. Then, "Multi-load View" dialog appears. Using this dialog, you may view the analysis results of each load case, and go further for visualization of the synthesized results due to combined load cases. This multi-loading view applies to all analysis results including displacements, stresses, etc., and also to most of the postprocessing images such as contours, deformed shape, etc. In the case of interactive real time processing, the multi-loading view is also processed in real time.



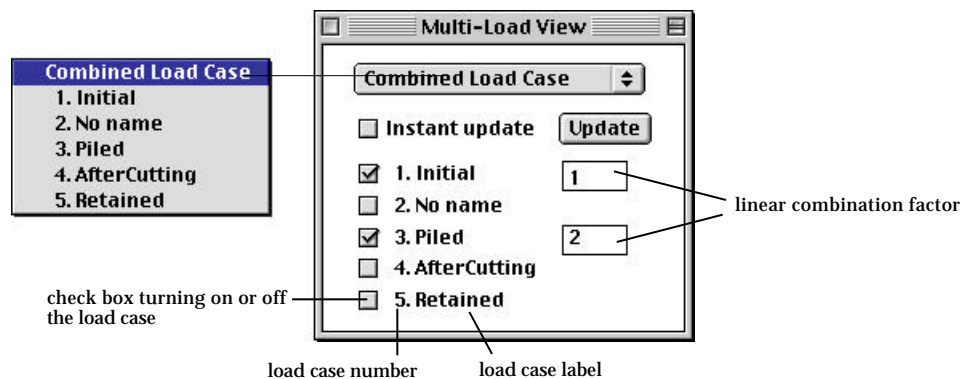
### ■ Visualizing the analysis results of each load case

The finite element processing produces as many data sets as the number of load cases supplied for the analysis. Each set appears as an item in the popup menu (Windows: dropdown list) of "Multi-load View" dialog. A computed data set is visualized by choosing the corresponding popup menu item. The visualization is instantly realized if "Instant update" box is checked. Otherwise, the view is updated only when **Update** button is clicked.

### ■ Visualizing the synthesized analysis results

In order to obtain the results synthesized by combining the load cases, select "Combined Load Case" item from the popup menu. Then, the dialog expands and the list of load cases appears at the lower part of the dialog. Each of the load cases is indicated by its number and label. Check the check box to the left in order to include a load case in the combination. There appears an editable text box to the right of the included load case item. Insert the load combination factor in the editable text box. The load combination is set by these factors of the included load cases.

If "Instant Update" item is checked, the view of the analysis results is instantly updated while the load combination is being set. Otherwise, the view is updated only when **Update** button is clicked.



## Visualization of Seepage Analysis Data

Analysis of a seepage problem gives various results including hydraulic heads, flow vectors, flow velocity, flow path, phreatic surface, and flow discharge. Some of them can be visualized by normal postprocessing functions such as contouring and vector representation. But there are other types of seepage analysis data which cannot be represented properly by such postprocessing functions. There are visualization functions prepared specially for seepage analysis. The menu items for these functions are provided in **Postpro** menu.

### Visualization of scalar and vector data

Contouring and vector representation are the most generalized method of visualizing finite element analysis results, and are appropriate for visualizing the scalar or vector data directly obtained from a seepage analysis.

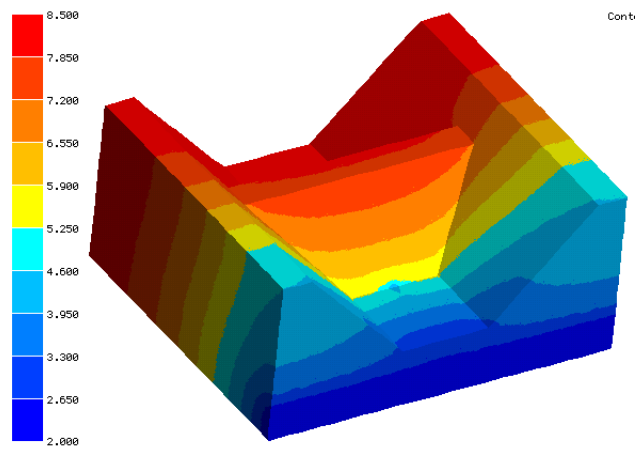
#### ■ Contouring of scalar data

There are a number of data items obtained from seepage analysis, which appear in the popup menu(Windows: dropdown list) of "Contour Display" dialog as shown below. The popup menu items are arranged in the same order as the data items are saved in the file. The scalar data from seepage analysis consist largely of hydraulic head, hydraulic gradient, flow velocity and pore pressure.

<b>Hydraulic Head</b>	Hydraulic head
<b>Gradient in X Dir</b>	Hydraulic gradient in X direction
<b>Gradient in Y Dir</b>	Hydraulic gradient in Y direction
<b>Gradient in Z Dir</b>	Hydraulic gradient in Z direction
<b>Gradient Norm</b>	Norm of hydraulic gradients
<b>Velocity in X Dir</b>	Component of flow velocity in X direction
<b>Velocity in Y Dir</b>	Component of flow velocity in X direction
<b>Velocity in Z Dir</b>	Component of flow velocity in X direction
<b>Velocity Norm</b>	Flow velocity
<b>Pore Pressure</b>	Pore water pressure

Various methods of applying contouring (for example, displaying contours on cut plane, parallel plane, etc.) are also valid for seepage analysis data. Refer to "Visualizing Scalar Data by Contours" section of this chapter.

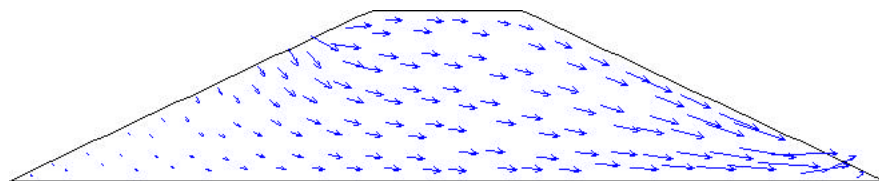
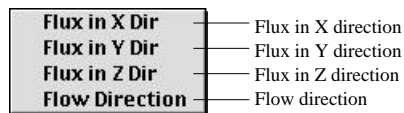
Other variant methods of displaying scalar data such as iso-surface representation and curve plotting can also be used in visualizing seepage analysis data. Refer to "Visualizing Scalar Data by Iso-surface and Others" section of this chapter.



&lt;Contour image of hydraulic head&gt;

### ■ Representing vector data

Flow velocities and its directions computed from seepage analysis can be visualized by vector representation. The X, Y and Z components of seepage flux can also be displayed by vector representation. Refer to "Visualizing Vector Data" section of this chapter for more detailed explanation on vector rendering.



&lt;Vector representation of flow velocity and directions&gt;

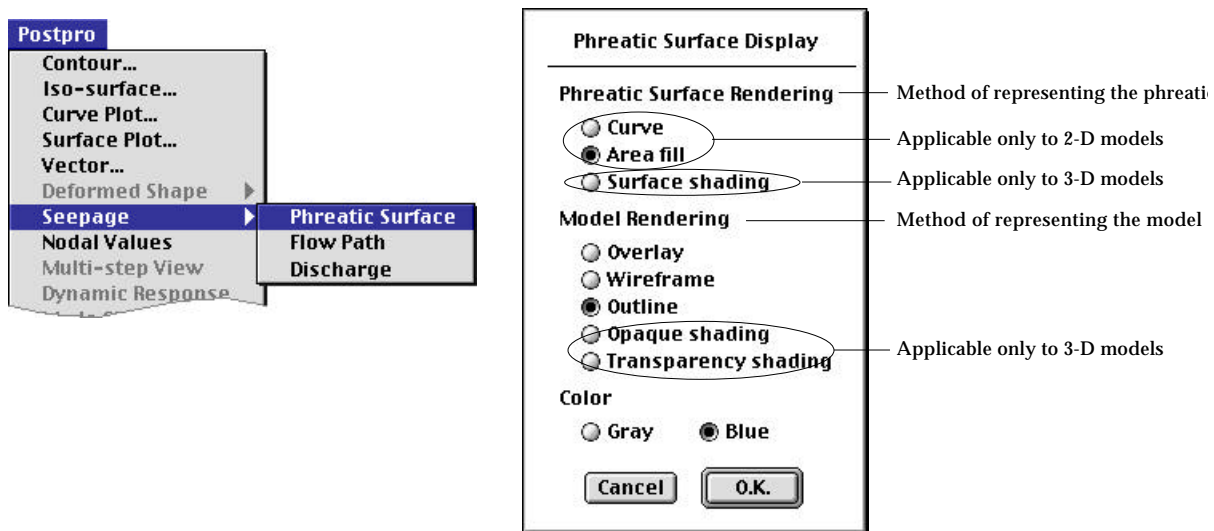
## Visualization of phreatic surface

The phreatic surface is one of the most informative and useful data obtained from unconfined seepage analysis. The phreatic surface is represented by a curve in plane and axisymmetric analysis, and by a surface in 3-D seepage analysis.

### ■ Rendering of phreatic surface in 2-D seepage analysis

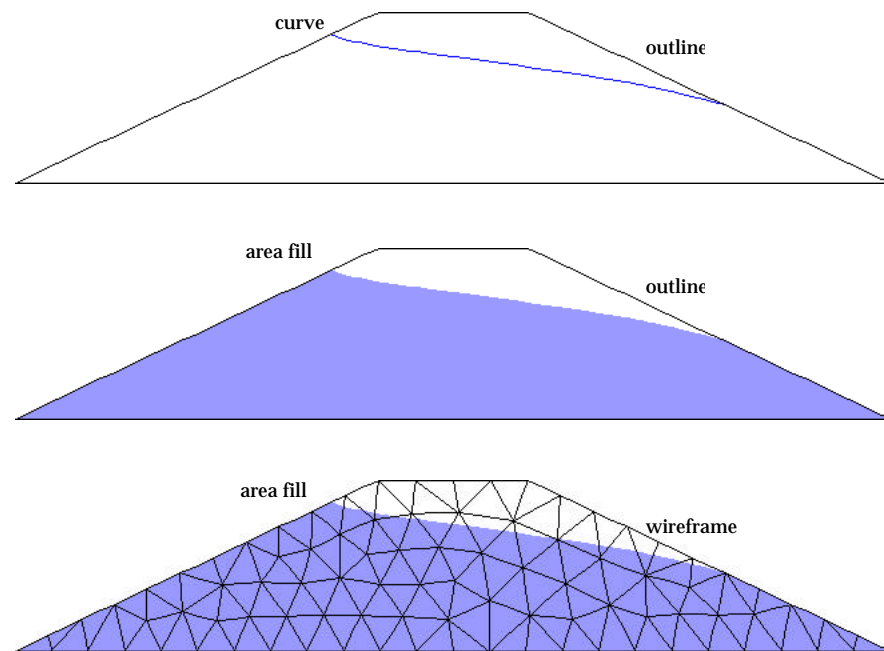
The phreatic surface is represented by a curve in plane or axisymmetric analysis. There are a number of options for rendering the phreatic surface which can be set by the following procedure.

- 1) Select "Phreatic surface" item from "Seepage" submenu of **Postpro** menu. "Phreatic Surface Display" dialog opens. Set the options related to rendering of the phreatic surface.



- 2) Set the "Phreatic Surface Rendering" option.  
The phreatic surface can be represented either by line(curve) drawing or by area fill. There are 3 items of this option, but only 2 items, "Curve" and "Area fill" are enabled for plane or axisymmetric seepage analysis. The other one is used only for 3-D volume seepage analysis.
- 3) Set the "Model Rendering" option.  
There are also options related to how the finite element model is rendered together with the phreatic surface. There are 5 items for this option, but only 3 items are enabled. The other one is used only for 3-D seepage analysis.
  - "Overlay" : The phreatic surface is drawn over the current screen image.
  - "Wireframe" : The wireframe mesh of the model is drawn.
  - "Outline" : The outline of the model is extracted and drawn.

- "Opaque shading" : A part of the model is rendered by shading. This option is applicable only to 3-D model.
  - "Transparent shading" : The boundary of the model is rendered as transparent objects. This option is applicable only to 3-D model.
- 4) Set the "Color" option.  
The rendering color of the phreatic surface can be selected from blue and gray.
- 5) Click  button to complete setting options of phreatic surface rendering. "Phreatic Surface Display" dialog closes, and the phreatic surface is rendered in accordance with the option setting.



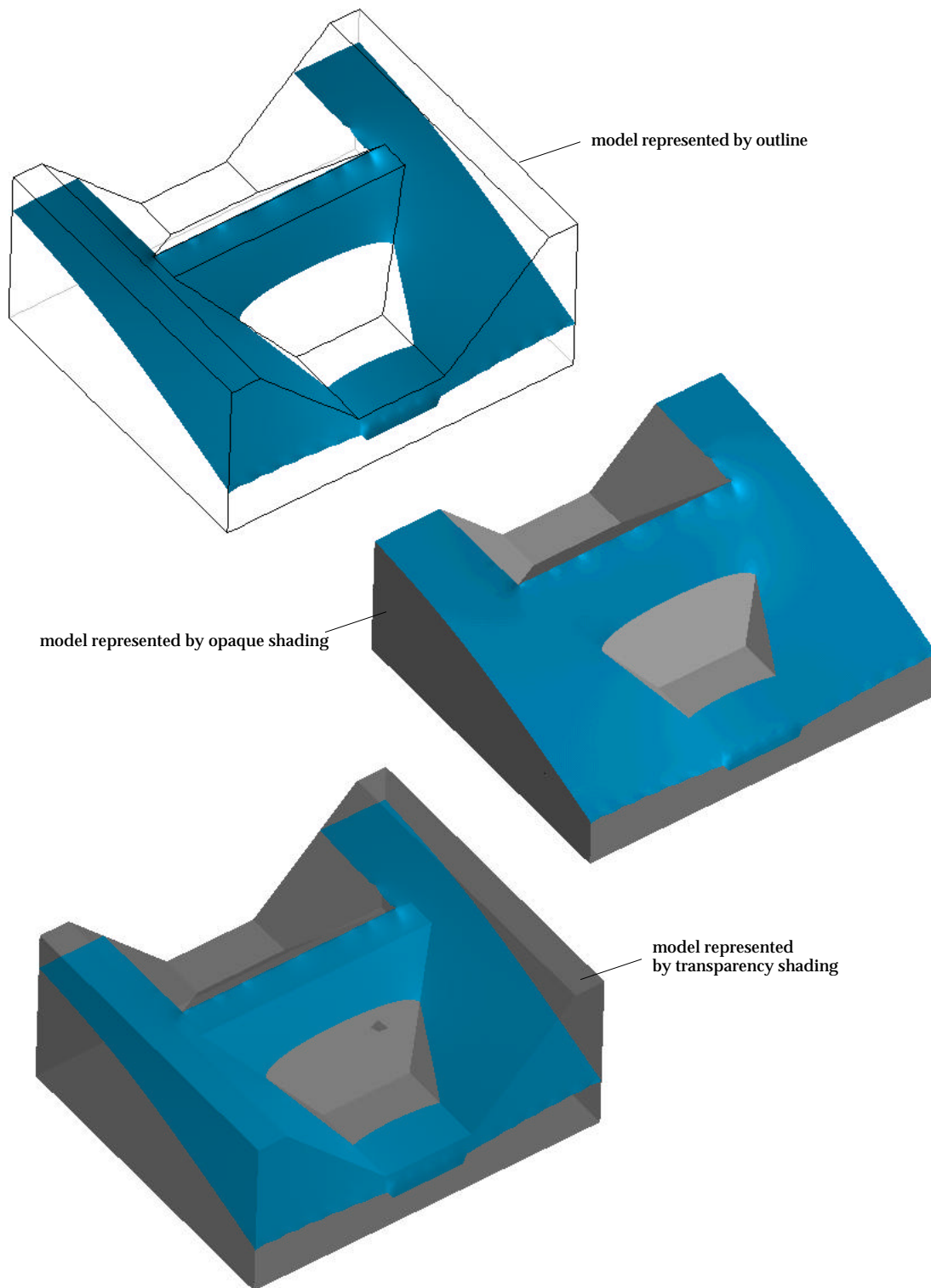
<Comparison of options for rendering of phreatic surface>

### ■ Rendering of the phreatic surface in 3-D seepage analysis

The phreatic surface is represented by a surface in 3-D seepage analysis. The phreatic surface is rendered in the form of iso-surface. Thus, phreatic surface rendering for 3-D analysis usually takes much longer time than 2-D analysis.

The rendering options are set by the same procedure as 2-D seepage analysis. There is no choice of phreatic rendering method other than "Surface shading." The finite element model can be rendered in various forms selectable from model rendering options in "Phreatic Surface Display" dialog.





< Rendering of phreatic surface in 3-D seepage analysis >

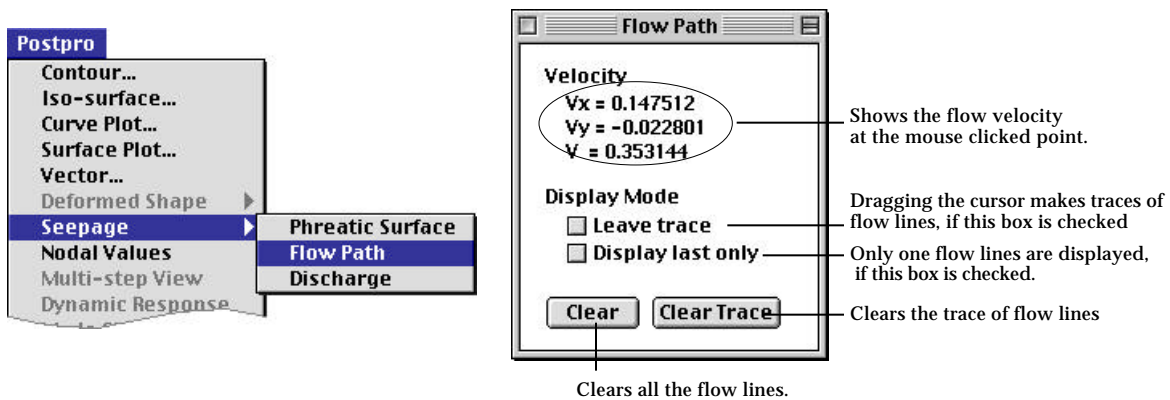
## Visualization of flow path

A flow path represents the route of water flow, and is obtained by tracing the velocity vector. A flow path is very close to but not identical with the flow line which can be obtained from a seepage analysis with reversed boundary conditions on the basis of dual relation between hydraulic head and flux.

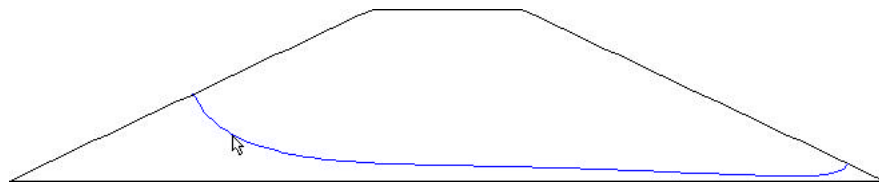
### ■ Interactively displaying a flow path

The phreatic surface is represented by a curve in plane or axisymmetric analysis. There are a number of options for rendering the phreatic surface which can be set by the following procedure.

- 1) Select "Flow Path" item from "Seepage" submenu of **Postpro** menu. "Flow Path" dialog opens. The flow lines and related information can be displayed interactively using dialog and mouse clicks.

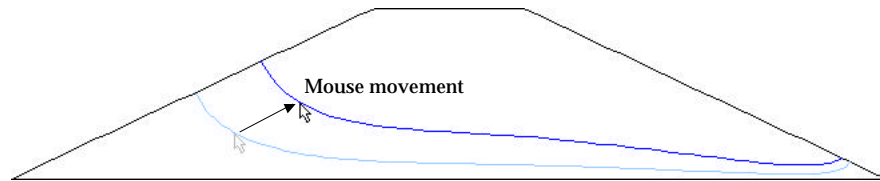


- 2) Select "Flow Path" item from "Seepage" submenu of **Postpro** menu. "Flow Path" dialog opens. The flow paths and related information can be displayed interactively using dialog and mouse clicks.
- 3) Click a point within the model.  
Then, the flow path passing through the point is drawn, and the flow velocity at that point is displayed on the dialog.



<Drawing flow path by clicking a point within the model>

- 4) Move the mouse with mouse button pressed to drag the flow path.  
The flow path is dragged following the cursor point, while moving the mouse with mouse button pressed.

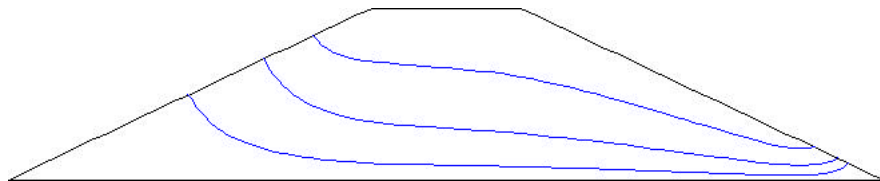


< Dragging the flow path >

### ■ Displaying multiple flow paths

A new flow path is drawn as another point is clicked. If "Display last only" item of "Flow Path" dialog is checked, only one flow line is drawn. Otherwise, all the flow lines at the clicked points are displayed together.

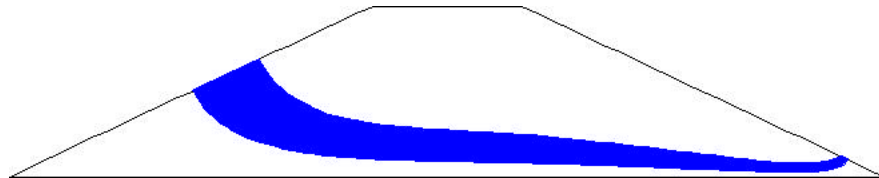
Uncheck "Display last only" box, and click points within the model. The flow paths respectively passing through the clicked points are displayed at the same time.



< Displaying multiple flow paths >

### ■ Overlaid display of flow paths

If "Leave Trace" item of "Flow Path" dialog is checked, all the flow paths drawn following the cursor movement are overlaid. Thus, the flow paths within a certain range can be displayed together as shown in the figure below.



< Overlaid display of flow paths>

### ■ Clearing flow paths

The displayed flow paths are deleted by clicking **Clear** button of "Flow Path" dialog. The overlaid trace of flow paths are cleared by clicking **Clear Trace** button which is enabled when "Leave trace" box is checked.

### ■ Getting flow velocity

The flow velocity and its components in X, Y and Z directions at the point of mouse click are displayed in "Flow Path" dialog. The display is continuously updated while the flow path is being dragged.

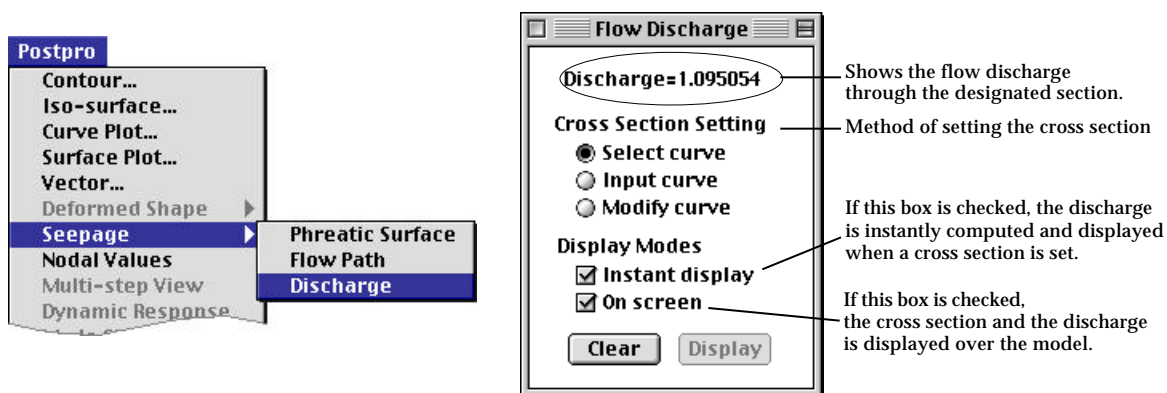
## Visualization of flow discharge

The flow discharge through a designated section can be viewed interactively. The section can be set at arbitrary position and in arbitrary direction as desired. There are some differences between 2-D and 3-D seepage models in setting the section.

### ■ Getting flow discharge in a 2-D seepage model

The flow section is represented by a straight line in 2-D seepage model. The cross section is interactively set and the discharge through this section is viewed by the following procedure:

- 1) Select "Discharge" item from "Seepage" submenu of **Postpro** menu. "Flow Discharge" dialog opens. The flow section is interactively designated with the aid this dialog, and the discharge is displayed in the section.



- 2) Choose the methods of setting the cross section in "Flow Discharge" dialog.
  - "Select curve" : The flow section is designate by selecting an existing straight line. This line may or may not be part of the analysis model.
  - "Input curve" : The flow section is designate by creating a new straight

line.

- “Modify curve” : The flow section is continuously updated by modifying an existing straight line.

3) Choose the display mode.

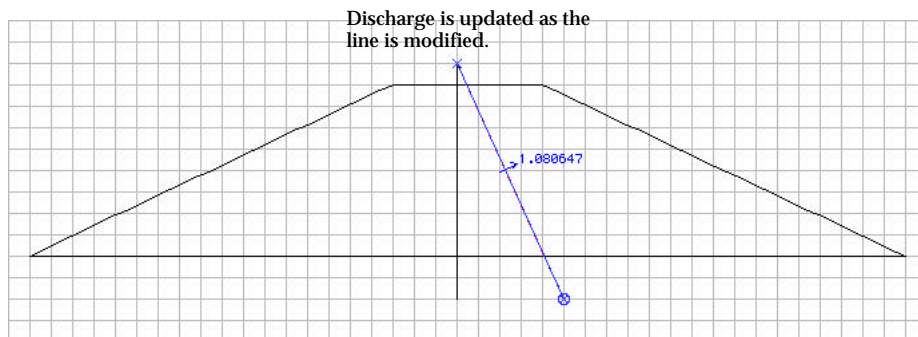
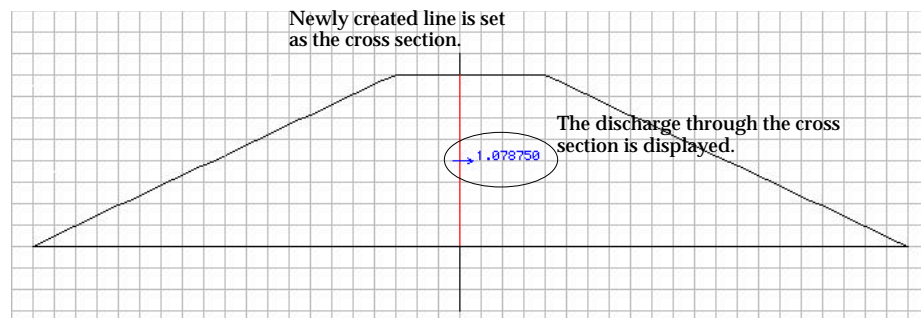
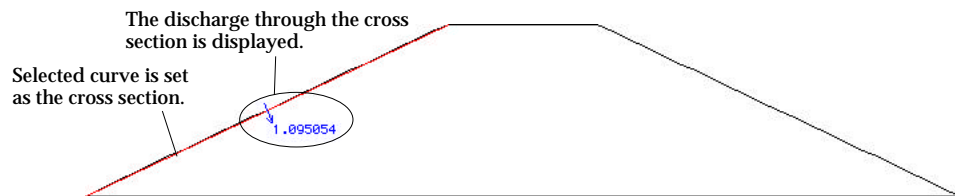
- “Instant display” : If this box is checked, the discharge is instantly computer, and displayed as soon as the cross section is designated.
- “On screen” : If this option is on, the discharge is displayed over the cross section image on the main window.

4) Set the cross section by the method chosen in step 2).

If "Instant display" mode is on, the discharge is displayed instantly as soon as the section is set. The discharge is displayed on "Flow Discharge" dialog. If "On screen" mode is on, the display and its direction is indicated over the cross section drawing.

5) Click **Display** button to display the discharge.

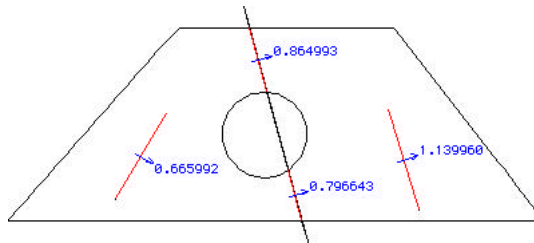
If "Instant display" mode is on, this step is not necessary. **Display** button is enabled only when "Instant display" mode is on.



< Displaying the discharge through specified cross section >

### ■ Getting flow discharges through multiple cross sections

More than one cross section can be specified for computation of discharge at the same time, simply by setting the cross sections one after another. Setting new cross section does not replace the old setting. Click **Clear** button to clear the setting.

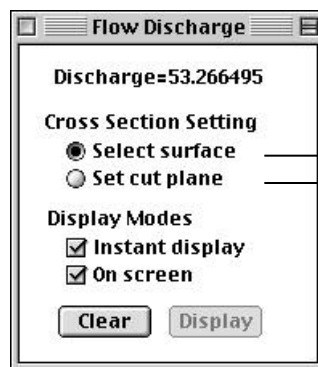


< Setting multiple cross sections for display of discharge >

### ■ Getting flow discharge in a 3-D seepage model

The flow section is represented by a plane or a surface mesh in 3-D seepage model. The cross section is interactively set and the discharge through this section is viewed by the following procedure:

- 1) Select "Discharge" item from "Seepage" submenu of **Postpro** menu. "Flow Discharge" dialog opens. The items of the dialog are not the same as that of 2-D model.



Choose this option to set the cross section using surface meshes.

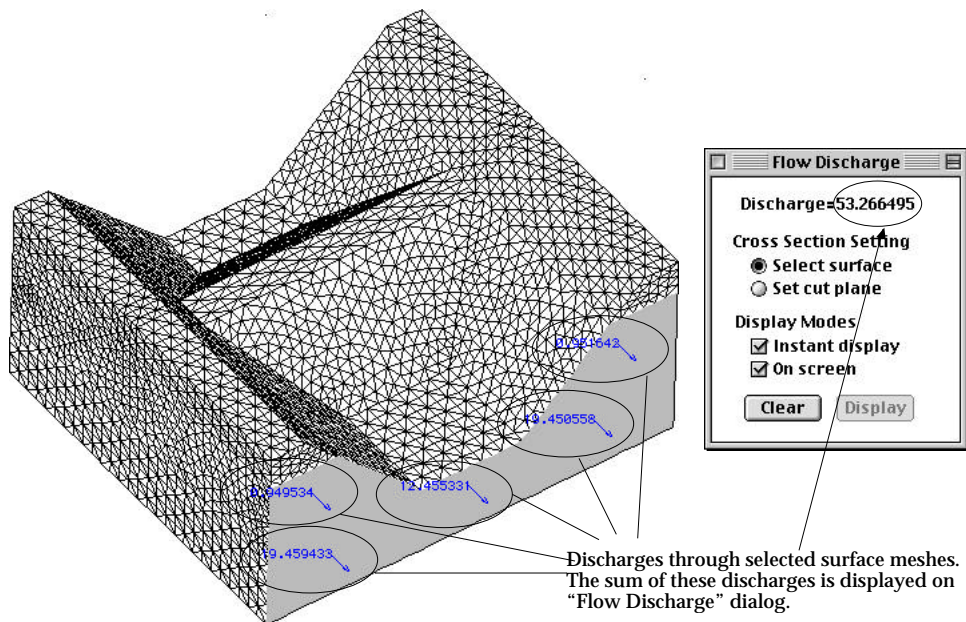
Choose this option to set the cross section using a cutting plane.

- 2) Choose the methods of setting the cross section in "Flow Discharge" dialog.
  - "Select surface" : The flow section is designate by selecting an existing surface mesh. The surface mesh should be part of the analysis model.
  - "Set cut plane" : The cross section is defined by the cut plane which is usually used for contouring.
- 3) Choose the display mode.  
It is the same as 2-D model. "Instant display" option is disabled if the method

of setting cross section is "Set Cut Plane".

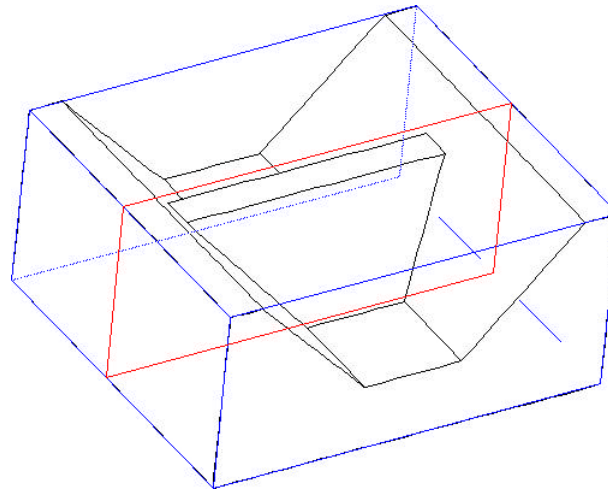
- 4) Set the cross section by selecting surface meshes if "Select surface" option is chosen in step 2).

It is possible to specify more than one surface mesh as the cross sections. The discharge is displayed over each of the surface meshes if "On screen" display mode is on. The sum of the discharge is also displayed on "Flow Discharge" dialog.

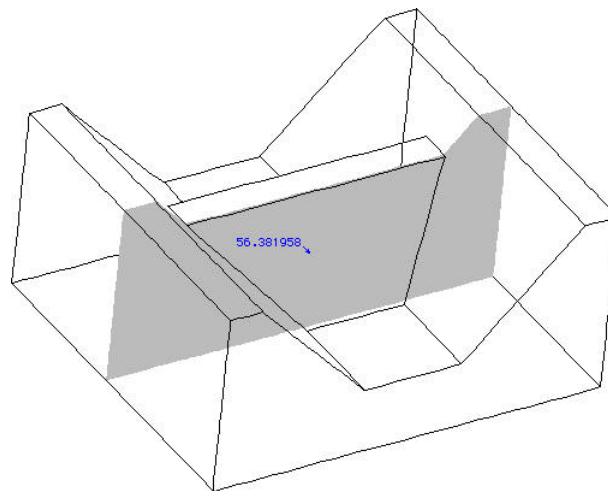


#### <Setting cross section by selecting surface meshes>

- 5) Set the cross section by cut plane if "Set cut plane" option is chosen in step 2). If this option is chosen, the cut plane setting mode is automatically activated, and "Instant display" mode is turned off. Instead, **Display** button is enabled. Set the cross section at the desired position by the cut plane. Refer to "Setting a cut plane for contouring" in this chapter.
- 6) Click **Display** button to display the discharge. If "Instant display" mode is on, this step is not necessary. **Display** button is enabled only when "Instant display" mode is on.
- 7) Click **Clear** button to clear the setting. The setting of the cross sections is cleared by clicking **Clear** button in the dialog.



<Setting the cross section by cut plane>



<Getting the discharge through the cross section>



## Image and Animation

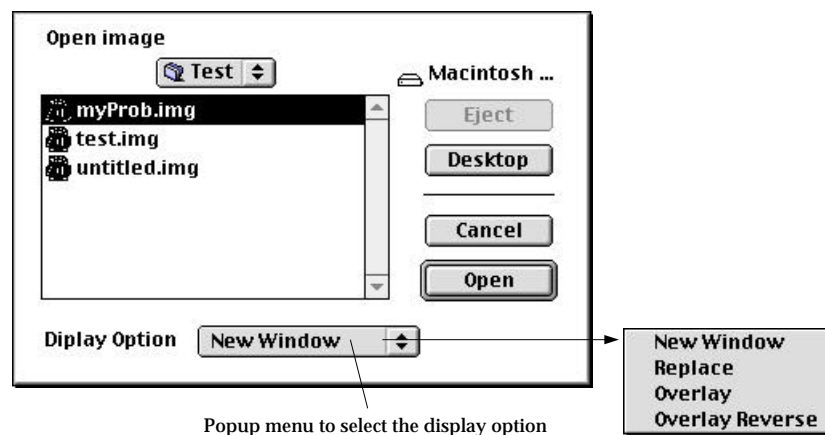
Postprocessing of continuum analysis creates various kinds of images visualizing the analysis results. The visualization can be recorded in the form of still images or animations. Images can be saved in files, and retrieved later. They can also be captured directly to the clipboard. Animations are created with a sequence of image frames directed by an animation script. They are saved in animation files for future replay on the screen.

### Image handling

There are 3 functions related with screen image on the main window: image saving, retrieving and capturing. The image can be saved in a file, or retrieved from the file. Current version of VisualFEA supports only images of its own format. The image of the main window can also be captured.

#### ■ Retrieving an image saved in a file

The image on the main window can be saved in a file, and can be read and displayed later. In order to retrieve an image, choose “Open Image...” item from **Postpro** menu. Then, the file opening dialog appears. The browser of the dialog shows only valid image files. Locate the desired image file using the browser, and click **Open** button after highlighting the file name or double click the file name. The file opens, and the image in the file is displayed on the screen. The file opening dialog has a popup menu to select the options for displaying the image. The image may be displayed in one of the following 4 different ways, which are provided as items of the popup menu.

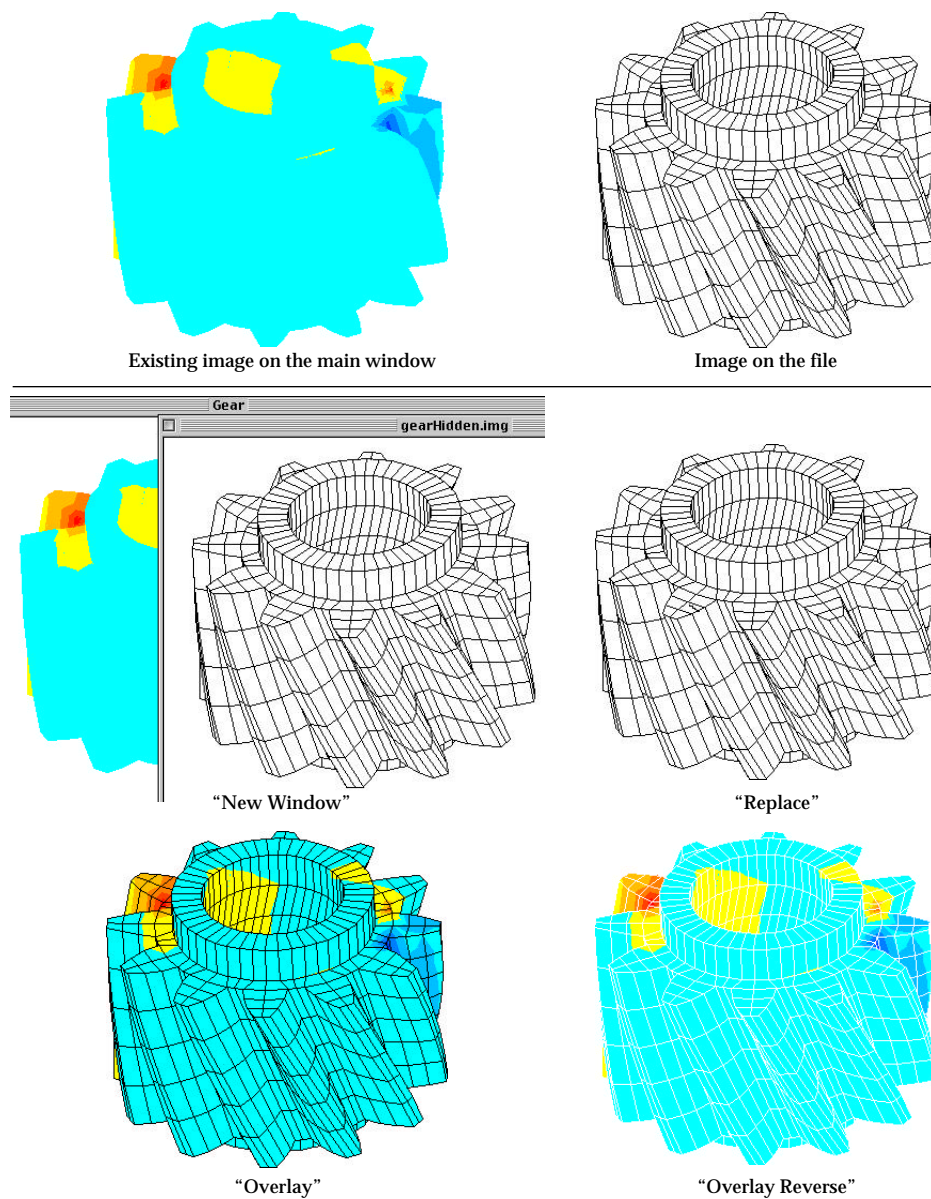


< File opening dialog for image retrieval >

- “New Window”: New window is opened, and the image read from the file is

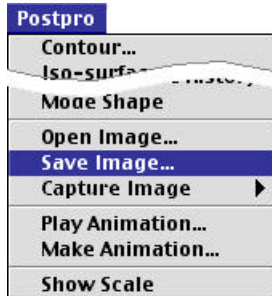
displayed on this window. The image on the main window remain intact.

- “Replace”: The existing image on the main window is replaced by the image read from the file.
- “Overlay”: The image read from the file is overlaid over the existing image on the main window. Only non-white pixels of the file image replace the corresponding pixels on the screen.
- “Overlay Reverse”: The image read from the file is overlaid over the existing image on the main window. Non-white pixels of the file image turn the corresponding pixels on the screen into white.



< 4 display options for retrieved image >

### ■ Saving the screen image in a file



The image displayed on the main window can be saved in a file for later retrieval. To save an image, choose “Save Image...” item from **Postpro** menu. Then, the standard file saving dialog appears. The file name is given initially as “untitled.img”. This name may be replaced by a desired one.

*Extension in image file name is not necessary, but recommended only for easy identification of file type. VisualFEA endows the type of the image file, and identifies the file by this type, and not by its extension.*

## Animation

VisualFEA has the capability of creating and playing animations by which the data resulting from the finite element analysis may be visualized dynamically. An animation is created with a sequence of image frames directed by an animation script. The animation is saved in a file for future replay on the screen.

### ■ Creating an animation

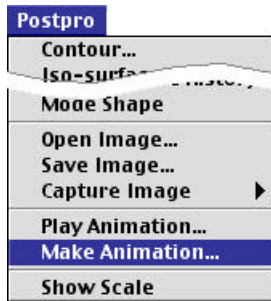
An animation file is created by the following steps:

- 1) Make an animation script file.

The animation script designates the data item for visualization and the sequence of animation. The script file should be written in ASCII text using any text editor such as “SimpleText”.

- 2) Launch VisualFEA and open the VisualFEA file to be used for animation.

The VisualFEA file should contain a completed finite element model with analysis results.



- 3) Choose “Make Animation...” from **Postpro** menu.

A standard file opening dialog appears on the screen, when the menu item is chosen.

- 4) Open the script file.

Browse the desired script file using the file opening dialog, and open the file. When the animation commands are completely read from the script file, a standard file saving dialog appears.

- 5) Give the title of the animation file to be created.

Insert the title of the animation file in the editable text box in the file saving dialog. The title is initially set as “untitled.ani”,. This title may be replaced by a desired one.

*Extension in the file name is not necessary, but recommended only for easy identification of file type. VisualFEA endows the type of the animation file, and identifies the file by this type, and not by its extension.*

- 6) Let VisualFEA create and save the animation.

Creation and saving of the animation file is started by clicking the button in the file saving dialog.

While the animation is created, it is displayed on the main window. At completion of creation, the animation is saved in the file with the designated title.

### ■ Writing the animation script

The script file is a text file with the commands directing the sequence of the animation. A string “11” representing animation script file should be placed at the beginning of the file and followed by animation script commands.

The commands are written in ASCII text. There are a number of script commands. A command is denoted by 3 characters, and are generally followed by the starting frame number and the ending frame number, and may also be followed by one or more additional argument(s). The generic form of their format can be written as:

```
com startingFrame endingFrame argument1 argument2...
```

The scrip commands are listed below.

- “con”: Contour command.

```
con startingFrame endingFrame scale1 scale2 scaleBarFlag
```

Contour image is displayed from *startingFrame* to *endingFrame* . The data designated by “dat” command is used for contouring. The contour is scaled by *scale1* at *startingFrame* and by *scale2* at *endingFrame* . If *scaleBarFlag* is 0, the scale bar is not shown. If *scaleBarFlag* is 1, the scale bar is displayed while animation is going on. The scale changes gradually *startingFrame* from to *endingFrame* . For example,

```
con 20 40 0.3 1.0 1
```

displays the contour image from frame 20 to 40. All the data values are scale down by the ratio of 0.3 at frame 20, and recovered to full scale (1.0) at frame 40. The scale of the data values changes gradually from frame 20 to frame 40. The scale bar is shown while the animation is going on.

- “dat”: Data command.

```
dat startingFrame endingFrame dataItem
```

The data of *dataItem* is applied from *startingFrame* to *endingFrame* . The argument *dataItem* indicates the data item in the popup menu of “Contour Display” dialog. For example,

```
dat 1 100 4
```

applies the specified data set to frame 1 to 100. The 4th menu item in the popup menu of “Contour Display” dialog reads “Shear Stress XY”, Thus, the shear stress  $\tau_{xy}$  is used for display from frame 1 to 100.

- “def”: Deformation command.

```
def startingFrame endingFrame scale1 scale2
```

Deformation of the model is animated from *startingFrame* to *endingFrame* . The deformation is scaled by *scale1* at *startingFrame* and by *scale2* at *endingFrame* . The scale changes gradually *startingFrame* from to *endingFrame* . For example,

```
def 5 60 0.0 1.0
```

displays the deformed shape of the model from frame 5 to 60. Deformations are depressed at frame 5, and get the full values at frame 60.

- “hid”: Hidden line removed mesh command.

```
hid startingFrame endingFrame
```

The model is rendered in the form of hidden line removed wireframe

- “out”: Outline command.

```
out    startingFrame    endingFrame
```

The model is rendered in the form of outline.

- “pan”: Panning command.

```
pan    startingFrame    endingFrame    hPan    vPan
```

The image is panned *hPan* pixels in horizontal direction and *vPan* pixels in vertical direction gradually from *startingFrame* to *endingFrame* . For example,

```
pan 30 40 200 -100
```

moves the screen image 200 pixels horizontally and -100 pixels vertically from frame 30 to 40.

- “rot”: Rotation command.

```
rot    startingFrame    endingFrame    xAngle    yAngle    zAngle
```

The view of the model is rotated *xAngle* degrees about x axis, *yAngle* degrees about y axis, and *zAngle* degrees about z axis gradually from *startingFrame* to *endingFrame* . For example,

```
rot 41 50 30.0 -120.0 0.0
```

rotates view 30° about x axis, -120° about y axis gradually from frame 41 to frame 50.

- “sha”: Shading command.

```
sha    startingFrame    endingFrame
```

The model is rendered in the form of opaque shading from *startingFrame* to *endingFrame* .

- “tra”: Transparent shading command.

```
tra    startingFrame    endingFrame    minTransparency    maxTransparency
```

The model is rendered in the form of transparent shading from *startingFrame* to *endingFrame* . The transparency ranges from *minTransparency* to *maxTransparency* in percentage. *minTransparency* and *maxTransparency* should have a value between 0 and 100. The value is 0 for opaque objects, and 100 for perfectly transparent objects. For example,

```
tra 51 70 20.0 80.0
```

displays the model by transparent shading from frame 51 to 70. The minimum transparency is set as 20%, and the maximum transparency as 80%.

- “zoo”: Zoom command.

```
zoo    startingFrame    endingFrame    zoomScale
```

The model is zoomed to the scale of *zoomScale*, gradually from *startingFrame* to *endingFrame* . For example,

```
zoom 51 70 2.0
```

zooms the view up to twice as large from frame 51 to 70.

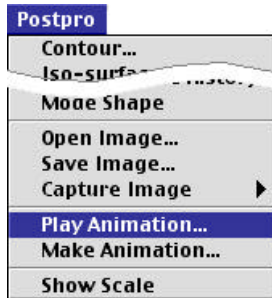
If 2 or more commands are in conflict with each other, the last one overrides the preceding ones. The use of the script commands is exemplified in the following, and their meanings are explained below.

11					
dat	1	80	3		
dat	81	200	6		
def	61	80	0.0	1.0	
rot	1	30	50.0	-120.0	90.0
con	1	40	0.0	1.0	1
con	41	80	1.0	0.0	1
def	81	100	1.0	0.0	
rot	121	150	-80.0	40.0	-120.0
con	101	160	0.0	1.0	1
con	161	200	1.0	0.0	1
sha	81	100			
tra	201	220	20.0	80.0	

The above script will result in an animation with the following contents.

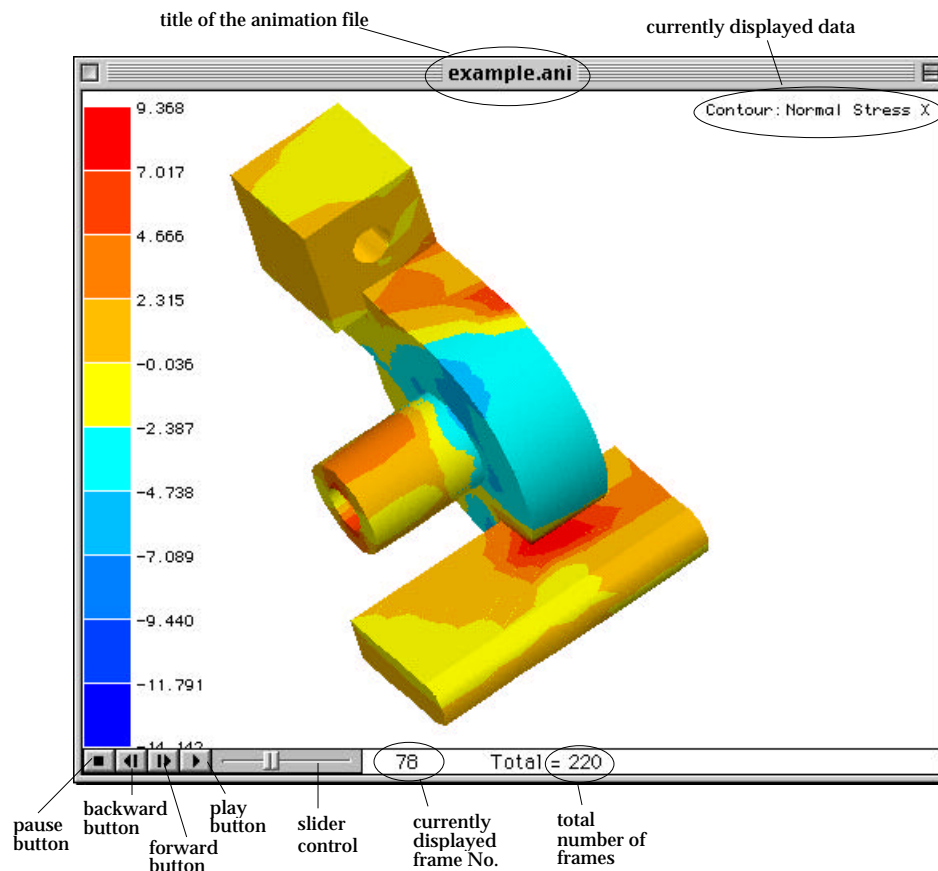
- The total number of frames in the animation is 220, because it is the maximum value of *lastFrame* appearing in the script.
- The data set designated by the popup menu item 3 is applied to frame 1 to 80.
- The data designated by the popup menu item 6 is applied to frame 81 to 200.
- Deformed shape of the model is displayed from frame 61 to frame 80. The deformation scale factor is 0 at frame 61 and 1.0 at frame 80.
- The view of the model is rotated 50° about x axis, -120° about y axis and 90° about z axis. The rotation starts at frame 1 and ends at frame 30.
- The contour image is displayed from frame 1 to 40. The data scale factor is 0.0 at frame 1 and 1.0 at frame 40. The scale change gradually from 0.0 to 1.0. If the scale is 0, all data values are set as 0. If it is 1.0, the actual data values are used. The data set of popup menu item 3 is applied for contouring, because the frames are between 1 to 80.
- The contour image is displayed from frame 41 to 80. The data scale factor is 1.0 at frame 41 and 0.0 at frame 80. The scale changes gradually from 1.0 to 0.0. The data set of popup menu item 3 is applied for contouring, because the frames are between 1 to 80.
- Deformed shape of the model is displayed from frame 81 to frame 100. The deformation scale factor is 1.0 at frame 81 and 0.0 at frame 100.
- The view of the model is rotated -80° about x axis, 40° about y axis and -120° about z axis. The rotation starts at frame 121 and ends at frame 150.
- The contour image is displayed from frame 101 to 160. The data scale factor is 0.0 at frame 101 and 1.0 at frame 160.
- The contour image is displayed from frame 161 to 200. The data scale factor is 1.0 at frame 161 and 0.0 at frame 200. The data set of popup menu item 6 is applied for contouring, because the frames are between 81 to 200.
- The shading image of the model is displayed from frame 81 to frame 100.
- The transparent shading image of the model is displayed from frame 201 to frame 220. The minimum transparency factor is 20%, and the maximum 80%.

### ■ Playing an animation



Once an animation is completed, it is saved in the designated file automatically. The animation can be replayed later by opening the file. In order to open an animation file, choose “Play Animation...” item from **Postpro** menu. Then, a standard file opening dialog appears on the screen. Using this dialog, browse and open the desired animation file. Only files with animation type appear on the browser. If the file is opened successfully, a new window will be created on the screen, and the saved animation will be played on the window.

The progress of the animation can be controlled by the buttons and the slider at the bottom left of the animation window. The total number of frames and the currently displayed frame number are shown at the bottom right of the window. The animation can be paused temporarily by clicking the pause button . The paused animation can be replayed by pressing the play button . The animation is moved one frame forward by clicking the forward button , or backward by clicking the backward button . The animation can be played forward or backward by dragging the nob on the slider back and forth.



< Animation window >



# **Chapter 8**

## **Diagrams for Frame Analysis**



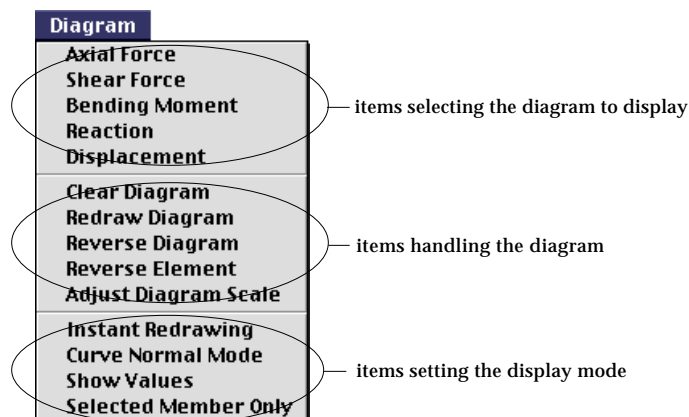
## Chapter 8 Diagrams for Frame Analysis

VisualFEA includes 2-D truss, 3-D truss, 2-D rigid frame and 3-D rigid frame in the category of frame analysis. Here, frame is the general name of both truss and rigid frame. The geometry of a frame is modeled by lines. Various kinds of data are generated from frame analysis: displacements, axial forces, shear forces, bending moments, etc. Owing to the characteristics of geometry and data in frame analysis, visualization of its results can be achieved more effectively by diagrams than surface images described in Chapter 7. Thus, **Diagram** instead of **Postpro** menu, appears on the menu bar if the analysis type is one of the following.

- 2-D truss
- 3-D truss
- 2-D frame
- 3-D frame

The procedure of frame analysis can be divided into the same three stages as in continuum analysis: preprocessing, analysis, and postprocessing. However, VisualFEA enables the users going back and forth through the whole procedure of frame analysis so interactively and freely that these stages may not be distinctive to the users. At the moment a menu item is selected, the corresponding diagram is instantly displayed. Thus, the user may feel as if the analysis stage were bypassed, although all the necessary steps were completed in a short period of time. Furthermore, whole cycle of frame analysis can be processed in real time. For example, the bending moment diagram of a frame is updated in real time, while its geometry is being modified. VisualFEA provides an option to take the above 3 stages step by step if desired, as in the case of continuum analysis.

The **Diagram** menu is composed of three parts as shown below. The first part of the menu consists of the items to select the diagram, the second part consists of items to handle the diagrams, and the last part consists of items to set the display modes.



## Diagrams for Truss and Rigid Frame

Frames in broad sense includes both truss and rigid frame, each of which is classified again into 2-D or 3-D case. The results of frame analysis are represented chiefly by member forces instead of stresses or strains. The frame member forces are usually represented best by diagrams, although axial force of truss members may be represented better by text strings than by a diagram. The commands displaying the diagrams are given as menu items in **Diagram** menu. As described at the beginning of this chapter, the first part of the menu consists of items to select the diagram and varies depending on the analysis type, and remaining items are common for all types.

Whenever the analysis type is altered by “Project Setup”, the menu items are changed to relevant ones. The first part items of the menu are enabled only when the model is complete for solution, and thus corresponding data are available.

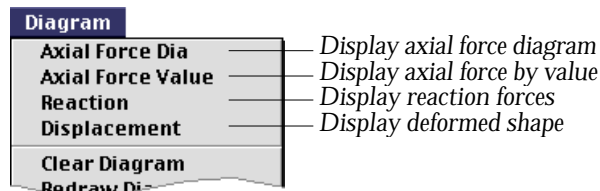
The diagram is displayed instantly after the corresponding menu item is chosen. The diagram is drawn over the shape of the structure. Two or more diagrams can be displayed at the same time.

### Visualizing analysis data of 2-D and 3-D trusses

If the analysis type is set to either 2-D or 3-D truss, **Diagram** menu has the items as shown below. The menu has only axial force items as for member forces. In addition, it has support reaction and displacement items. The following analysis results are available for 2-D and 3-D trusses.

- Axial force
- Support reaction
- Displacement

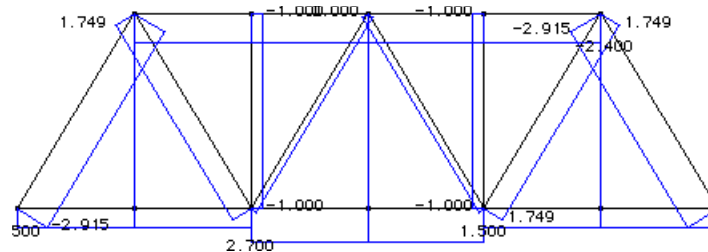
It is assumed that all truss members are pin-connected at the joints. Thus, truss members are subject to only axial force. The data from truss analysis are relatively simple, and can be visualized efficiently either by diagrams or by text strings.



#### ■ Displaying axial force diagram

It is assumed in VisualFEA that all truss members are straight, and forces are acting only at their joints, but not in the middle of the members. Thus, the axial force in a

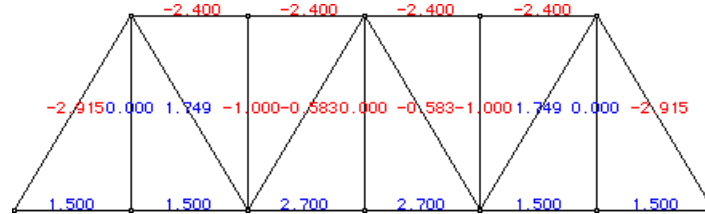
truss member is always uniform throughout its entire length. To display the axial force by diagram, choose “Axial Force Dia” item from **Diagram** menu. An example of axial force diagram is shown below.



<Axial force diagram >

### ■ Displaying axial force value

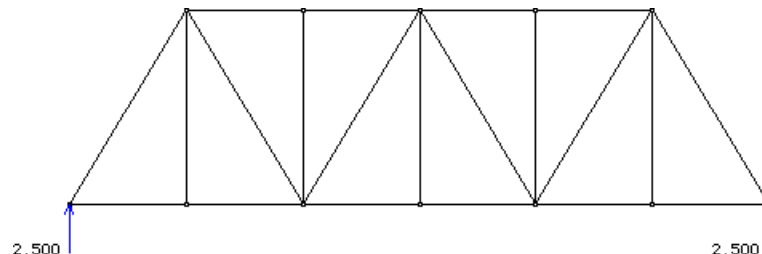
As the axial force in a truss member is uniform, it is simpler to express the value of the force magnitude by the text than to display its diagram. To display the axial force by text, choose “Axial Force Value” item from **Diagram** menu. If the axial force is compressive, the value bears negative sign and is represented in red letters. Otherwise, the value is positive and represented in blue letters.



< Axial force magnitude represented in text string >

### ■ Displaying reactions at supported nodes

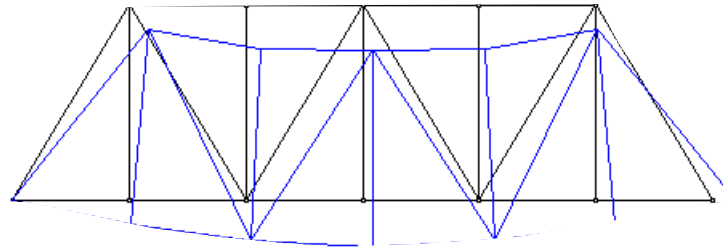
To display the support reactions, choose “Reaction” item from **Diagram** menu. The reactions are represented by a force symbol and the text of its magnitude. The reactions are shown only for constrained DOFs with non-zero value at supported nodes.



< Axial force magnitude represented in text string >

### ■ Displaying displacements

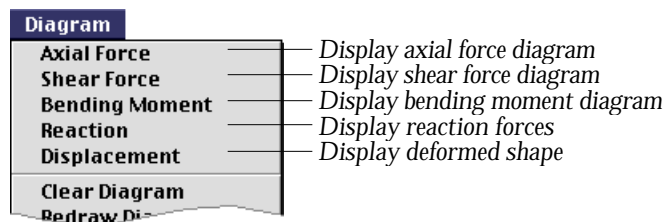
To display the displacements of the truss, choose “Displacement” item from **Diagram** menu. It is assumed that a truss member has only axial deformation. Accordingly, each member is represented by a straight line before and after deformation.



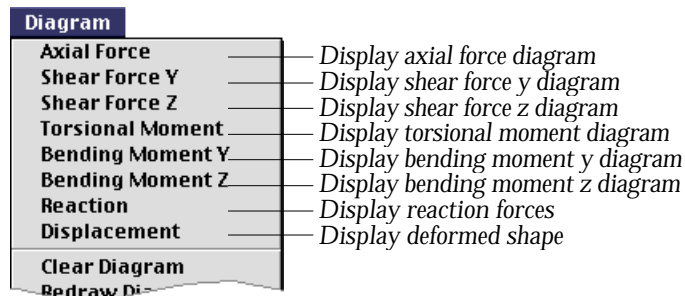
< Axial force magnitude represented in text string >

### Visualizing analysis data of 2-D and 3-D rigid frames

If the analysis type is set to 2-D rigid frame, **Diagram** menu has the items as shown below. As describe at the beginning of this chapter, the menu is composed of 3 parts. The first part of the menu has items of axial force, shear force and bending moment as for member forces. There are also support reaction and displacement items.



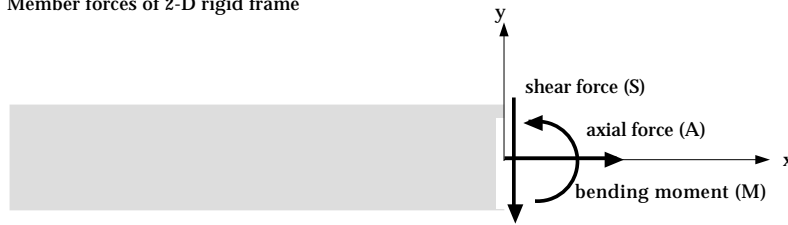
If the analysis type is set to 3-D rigid frame, **Diagram** menu has more items as shown below. The first part of the menu includes 6 components of an axial force, 2 shear forces, 2 bending moments, and a torsional moment. There are also support reaction and displacement items.



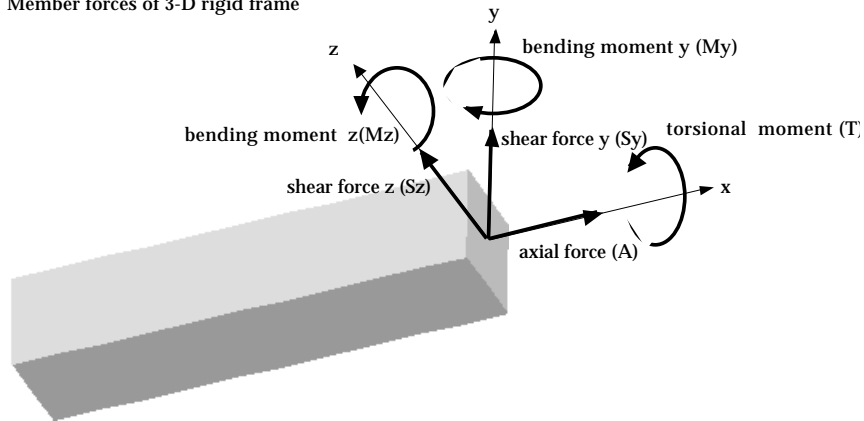
The menu items are enabled only when the model is complete for solution, and thus the corresponding data are available. The diagram is displayed instantly when the menu item is chosen. The displayed diagram is indicated by a check mark in front of the menu item.

The components of member force are given with respect to local coordinates as shown in the figure below. The sign of internal forces are determined in accordance with conventional usage.

Member forces of 2-D rigid frame



Member forces of 3-D rigid frame



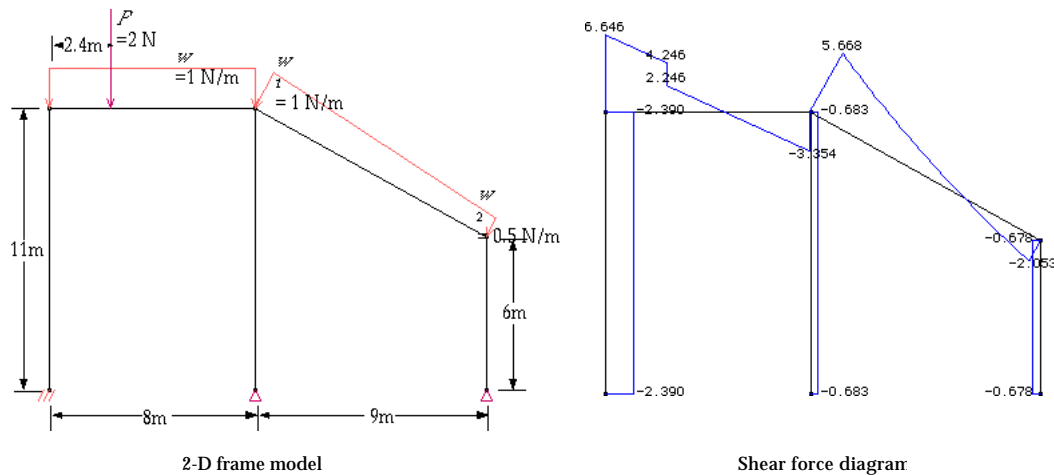
< Member forces in 2 and 3-D rigid frames >

### ■ Displaying axial force diagram

To display the axial force (A) diagram for 2-D and 3-D rigid frame, choose “Axial Force” item from **Diagram** menu. Axial force is equivalent to that of trusses. But its value is not necessarily uniform in a rigid frame member. Therefore, diagram is the only option to represent the axial force in rigid frames, differently from the case of trusses in which member force is uniform throughout the whole length and accordingly its magnitude can be represented by a single text string.

### ■ Displaying shear force diagram for 2-D rigid frame

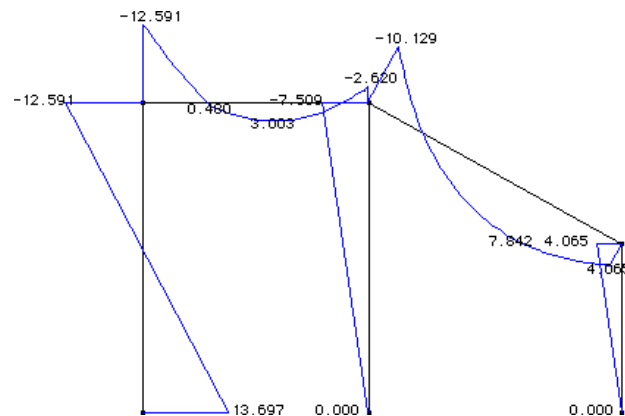
There is only one shear force component (S) for 2-D rigid frame. To display the shear force diagram, choose “Shear Force” item from **Diagram** menu. The diagram reflects the variation of shear forces within a member in its length direction. The values are shown in text string at member ends, at points of abrupt change, and at the points of local maximum or minimum.



< Shear force diagram for a 2-D rigid frame >

### ■ Displaying bending moment diagram for 2-D rigid frame

There is only one bending moment component (M) for 2-D rigid frame. To display the bending moment diagram choose “Bending Moment” command from the menu. The diagram reflects the variation of bending moments within a member in its length direction. The values are shown in text string at member ends, at points of abrupt change, and at the points of local maximum or minimum.

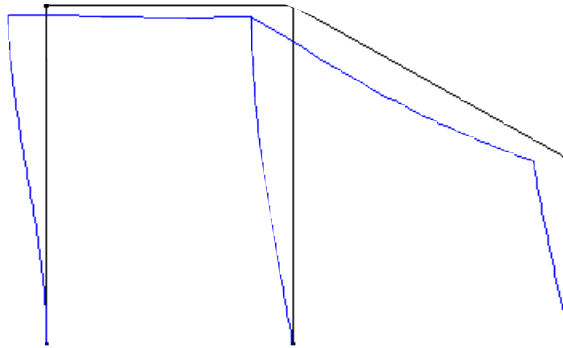


< Bending moment diagram >



### ■ Displaying deformed shape of rigid frame

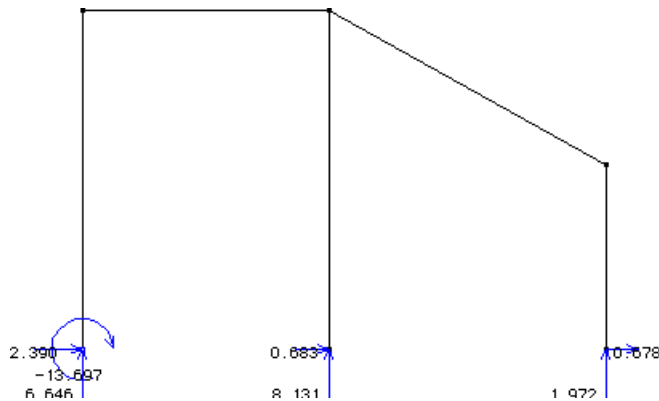
In order to display the deformed shape of 2-D or 3-D rigid frame, choose “Displacement” command from the menu. The deformed shape of a rigid frame member is no longer straight, because frame members are subject to not only axial but also bending deformation.



< Deformed shape >

### ■ Displaying support reactions of rigid frame

In order to display the support reactions of 2-D or 3-D rigid frame, choose “Reaction” command from the menu. The reactions are represented by a force symbol and the text of its magnitude. The reactions are shown only for constrained DOFs with non-zero value at supported nodes.

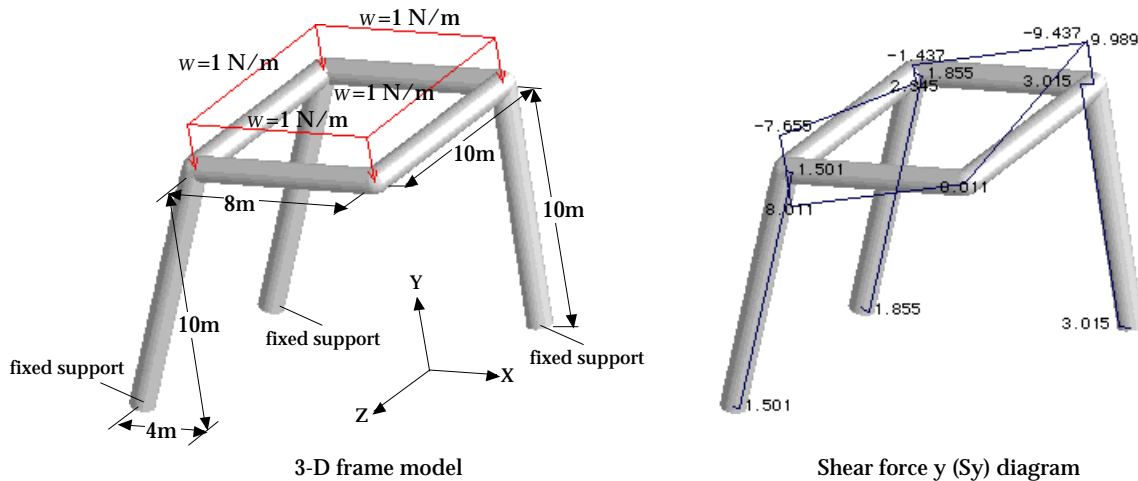


< Deformed shape >

### ■ Displaying shear force diagram for 3-D rigid frame

There are two shear force components ( $S_y$  and  $S_z$ ) for 3-D frames. To display the shear force diagram, choose “Shear Force y” or “Shear Force z” item from **Diagram** menu. Here, y and z directions are given in local coordinates defined on the basis of member axis.

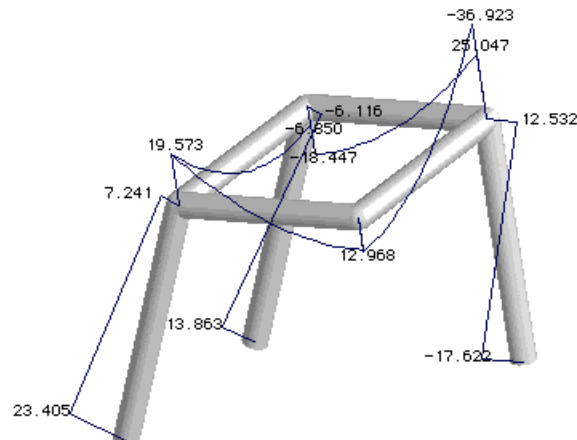
An example of 3-D frame modeling and shear force diagram is shown below. In this example, frame members are rendered in shading. Representation of frame members by shading is not essential, but desirable for 3-D frames to improve the understandability of the diagram. In order to get such an image, set the render mode to shading by choosing “Shading” from **Render** menu.



< Shear force diagram for a 3-D rigid frame >

### ■ Displaying bending or torsional moment diagram for 3-D frame

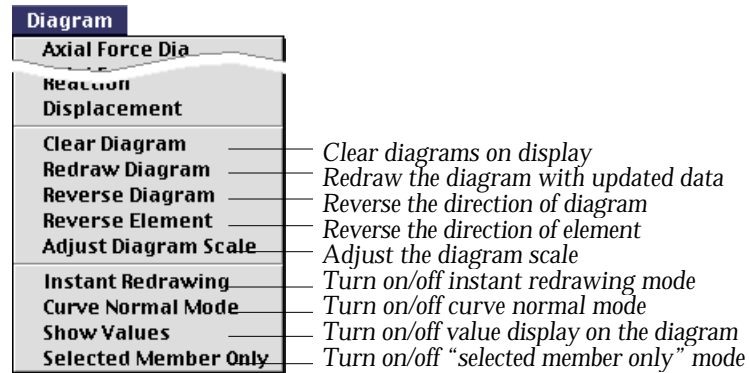
There are two bending moment components ( $M_y$  and  $M_z$ ) and a torsional moment ( $T$ ) for 3-D frames. To display the diagrams of bending moment about  $y$  and  $z$  axis, choose “Bending Moment  $y$ ”, and “Bending Moment  $z$ ” items respectively from **Diagram** menu. To display the torsional moment diagram, choose “Torsional Moment” from the menu.



< Bending moment  $z$  ( $M_z$ ) diagram for a 3-D rigid frame >

## Diagram Related Functions

As described at the beginning of this chapter, the **Diagram** menu is composed of three parts. Items in the second part are related to manipulating the display of the diagrams, and those in the third part are related to setting the diagram display modes. These menu items are common regardless of the analysis subject.



There are other functions, not in the menu items, to control the display of diagrams. They are also described in this section.

## Manipulating the diagrams

There are functions to control the display of diagrams, such as adjusting the scale of the diagram, changing the direction of the diagram, updating the diagram and so on. Some of them are provided as menu commands which are shown in the second part of the menu. Some of these functions are not provided as menu commands.

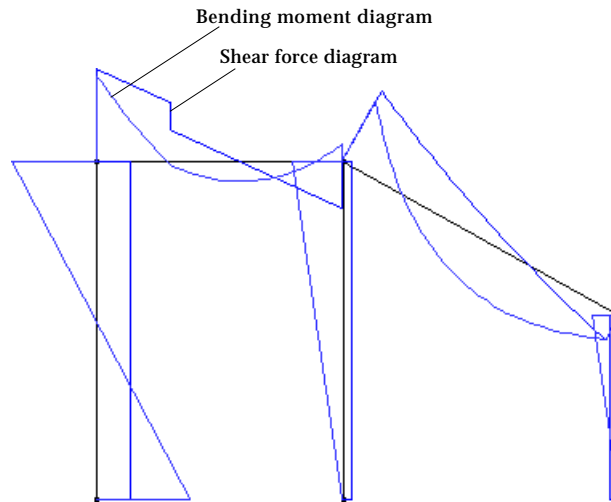
### ■ Displaying more than one diagram

More than one diagrams can be displayed at the same time by choosing the display items with *shift* key pressed. As an example, the shear force diagram and the bending moment diagram are displayed together in the figure below. To get such an image, first choose "Shear Force" command from menu. Now that the shear force diagram is displayed on the main window, press *Shift* key and choose "Bending Moment" command. Then, the bending moment diagram is overlaid on top of the shear force diagram. In this example, all the texts are turned off by pressing *Control* key in order to relieve complexity of the display.

It is also possible to remove a diagram from overlaid diagrams. Choose the menu command corresponding diagram to remove. Then, the display is updated with removal of the chosen item.

On the other hand, if a new diagram command is chosen from menu without pressing *shift* key, all the currently displayed diagram are replaced by the newly selected item.

All the menu items corresponding to the currently displayed diagrams are checked in front.



< Shear force diagram overlaid with bending moment diagram >

### ■ Clearing diagram

To remove the diagram(s), choose “Clear Diagram” command from **Diagram** menu. All the diagrams are cleared, and only the structural members are displayed. The check marks in front of the currently displayed items are also cleared.

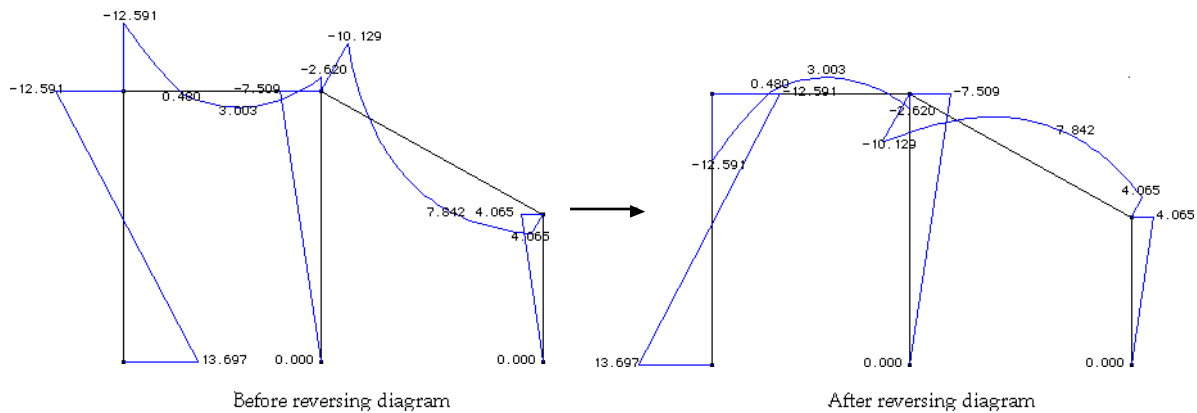
### ■ Redrawing diagram

To update the diagram after changing the model, choose “Redraw Diagram” command from **Diagram** menu. The diagrams are redrawn with the new analysis results.

If instant drawing mode is turned on, the diagrams are automatically updated upon any change in the model data, and therefore, this command has no additional effects. Instant drawing mode is explained in the later part of this section.

### ■ Reversing diagram directions as a whole





It is sometimes desired to reverse the positive direction of the diagrams. To reverse the diagram direction, choose “Reverse Diagram” command. The values remain unchanged, but the diagram directions of all elements are reversed.

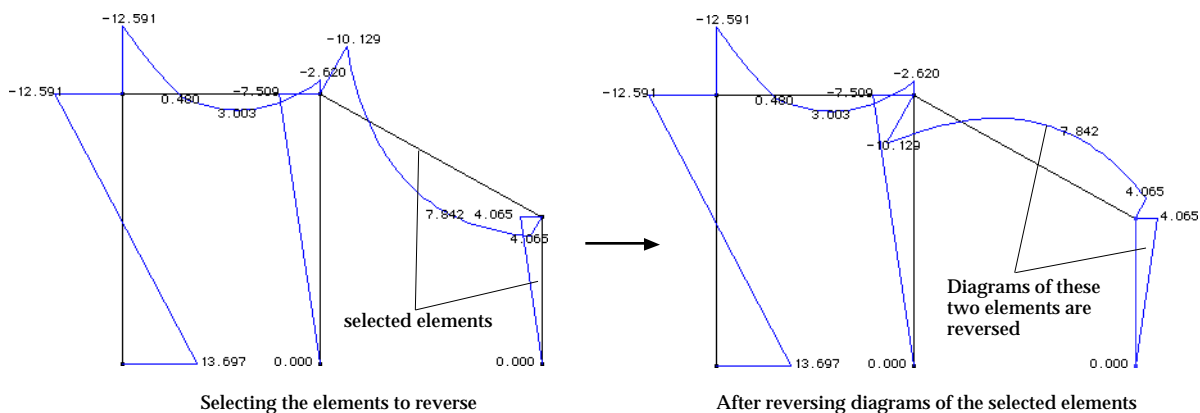


< Reversing the diagram directions as a whole >

### ■ Reversing diagram directions of selected element(s)

There are circumstances under which the diagrams of only certain elements need be reversed. Selected reversal of the diagram directions can be achieved by the following steps:

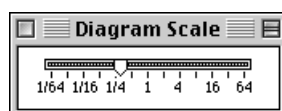
- 1) Start the element selection tool , or the curve selection tool , if any of these tools is not pressed.  
Either element selection tool or curve selection tool may be used to select frame elements for diagram reversal.
- 2) Select elements if  tool is pressed, or select curves if  tool is pressed.  
If a curve is selected, the frame element(s) on the curve is(are) automatically included for diagram reversal.
- 3) Choose "Reverse Element" command in **Diagram** menu.  
The diagram(s) of selected element(s) is(are) reversed, but the values remain unchanged.



< Reversing the diagram directions of selected elements >

### ■ Adjusting the scale of diagrams

The scale of a diagram is initially determined by VisualFEA, and the diagram of the entire structure is drawn in this scale. The scale can be altered interactively using the scale slider as shown below.



To bring up the slider, choose “Adjust Diagram Scale” command from **Diagram** menu. The scale of the diagram may be enlarged or reduced by moving the nob of the slider. Diagrams with different scales are compared in the following example.

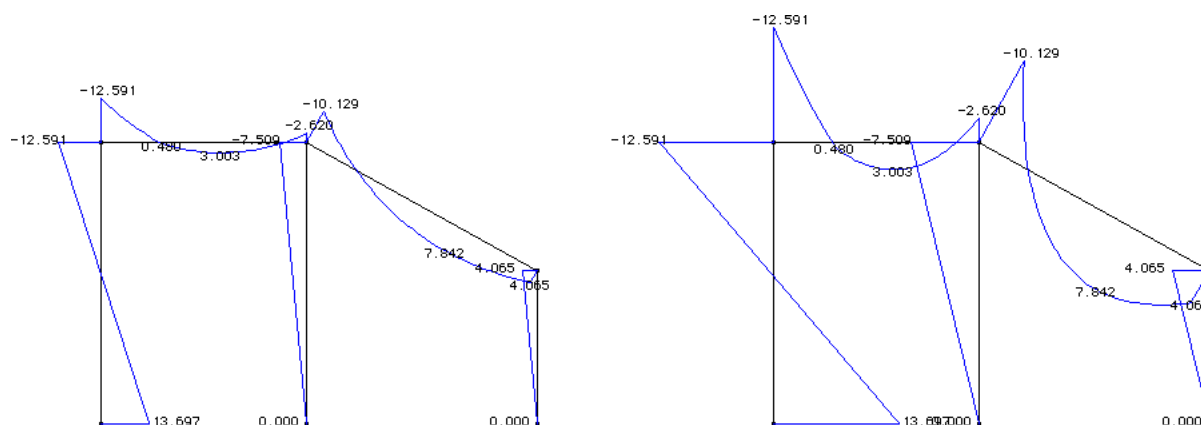


Diagram drawn with smaller scale

Diagram drawn with larger scale

< Adjusting the scale of the diagrams >

### Setting display options

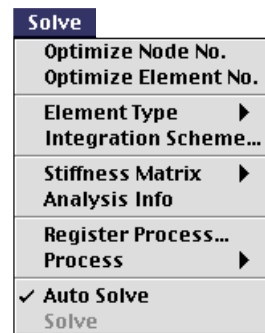
There are a few display options determining the manner by which the diagrams are drawn or updated. Each of these options can be turned on or off by selecting the corresponding item in **Diagram** menu. If the option is on, the item is check marked in front.

#### ■ Instant redrawing mode

If instant redrawing mode is turned on, the diagram is automatically updated immediately after modeling data are altered. Otherwise, the diagram is not updated until either another diagram item is selected, or “Redraw Diagram” command is issued.

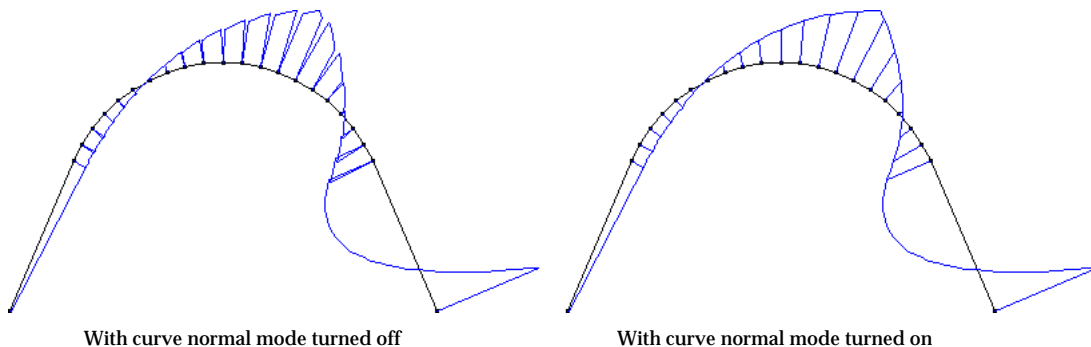
The instant redrawing mode is on when “Instant Redrawing” item of **Diagram** menu is checked. The menu item is enabled only when “Auto Solve” item of **Solve** menu is checked. Otherwise, the item is unchecked and disabled, and accordingly, the instant redrawing mode is turned off.

The instant redrawing mode is used for instant real time processing, in which one cycle of modeling, analysis and visualization is carried out in real time. Refer to “Instant real time processing” in Chapter 6 for more details.



### ■ Curve normal mode

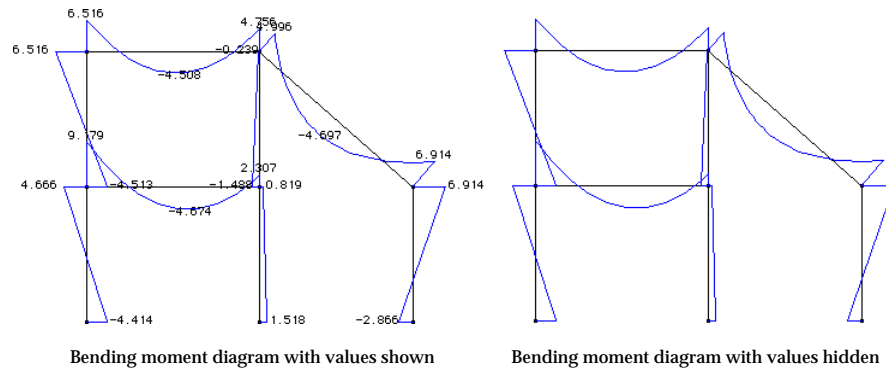
VisualFEA has only frame elements of straight line. Therefore, a curved member should be approximated by a number of line segments. Such an approximation brings error in computed results. The degree of error can be reduced by increasing the number of segments. However, artificiality in the diagram can not be avoided even if large number of segments are used, because diagram is drawn for each segment as an independent frame member. Curve normal mode is to make up for such a defect. The diagram is drawn normal to the original curve instead of line segments, and thus a continuous and smooth diagram is obtained along the curved member. Curve normal mode is turned on or off by choosing “Curve Normal Mode” item of **Diagram** menu.



< Effect of curve normal mode >

### ■ Turning on/off text of diagram values

It is necessary to show numerically some of the magnitudes represented by the diagram. The text strings indicating those values are initially displayed over the diagram. These text strings can be shown or hidden by choosing “Show Values” item of **Diagram** menu. If the item is checked, text strings are shown. Otherwise, they are hidden. If there are too many text strings to make out each of them, and if it is not essential to display the text, it is desirable to hide the text strings.



< Showing or hiding the text strings of the diagram values >

### ■ Temporarily turning off text of diagram values

The text strings of the diagram can be hidden temporarily by pressing *Control* key. As soon as the key is released, the text strings appear again.

### ■ Popping up the hidden texts

While the texts are hidden by pressing *Control* key, the screen cursor turns into question mark, **?**. As you move the cursor over the diagram, and scan through the diagram, the hidden text strings are popped up one after another. In case there are too many text strings to make out each of them, this is the one method of examining the value of the diagram.

### ■ Selectively turning on text of diagram values

It is also possible to display the text strings selectively by the following steps.

- 1) Check “Show Values” item of **Diagram** menu, if it is not.  
The text strings of the diagram values are displayed if the menu item is checked.
- 2) Press *Control* key.  
Text strings are suppressed temporarily by pressing the key, and at the same time, the screen cursor turns into **?**.
- 3) Move the cursor over the point of the diagram whose value is to be displayed.



The hidden text string pops up when the cursor is positioned around the corresponding point of the diagram. If the cursor moves away from the point, the string pops off.

- 4) Freeze the text string by clicking the point.

If the text string pops up, click the point by mouse button. Then, the string is frozen at the point, and will not disappear when the cursor moves away.

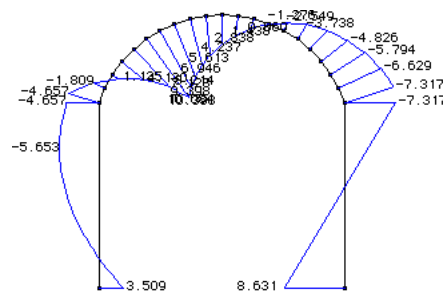
The frozen string can be hidden by clicking the point once again.

- 5) Repeat step 3) and 4) until all the desired text strings are popped up and frozen.

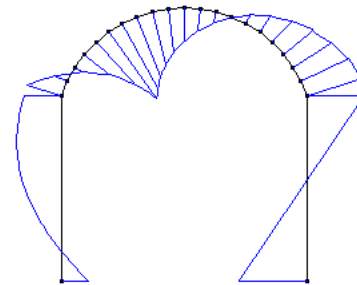
Scan through the diagram by moving the cursor. And freeze the desired text strings, whenever they pop up.

- 6) Release *Control* key.

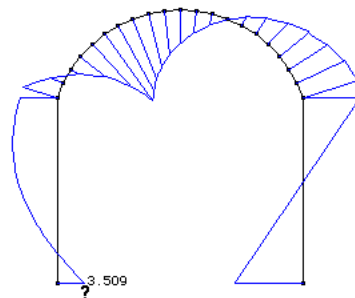
When *Control* key is released, the frozen text strings remain as they are, but the other text strings are suppressed. In this way, only selected text strings can be displayed.



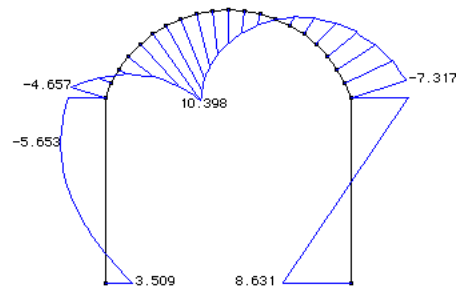
1. Display the diagram and the values.



2. Press *Control* key to turn off the text.



3. Move the cursor over the diagram.  
4. Freeze the text string.







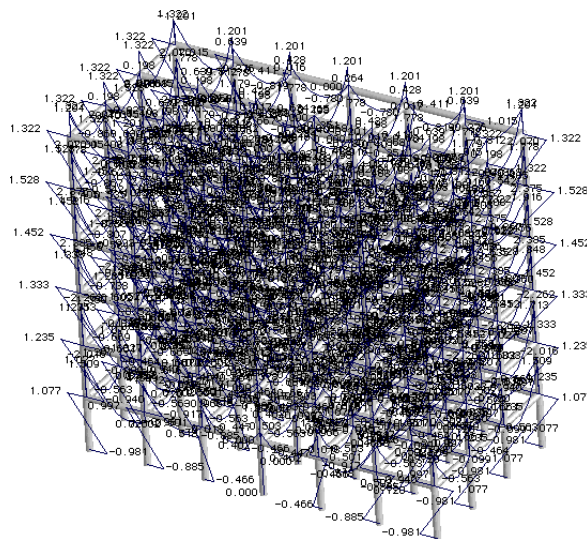
5. Repeat step 3 and 4 for all the text strings to be displayed.  
6. Release *Control* key.

<Selectively displaying the text strings of the diagram>

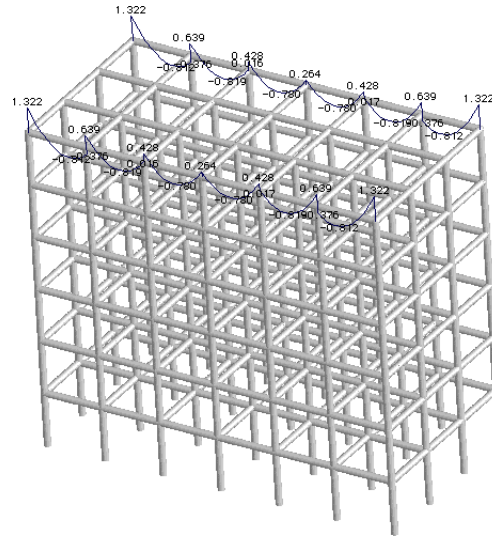
### ■ Displaying part of the diagram for selected members only

If the model is complex and the diagram is clustered together, it is difficult to make out the contents of the diagram. In such cases, it is useful to display only part of the diagram of the selected members. This can be achieved by the following steps:

- 1) Check "Show Values" item of **Diagram** menu, if it is not.  
The text strings of the diagram values are displayed if the menu item is checked.
- 2) Check "Selected Member Only" item of **Diagram** menu, if it is not.  
If the item is checked, diagram drawn only for the selected member(s).
- 3) Start the element selection tool , or the curve selection tool , if any of these tools is not pressed.  
Either element selection tool or curve selection tool may be used to select frame elements for selective display of the diagram.
- 4) Select elements if  tool is pressed, or select curves if  tool is pressed.  
If a curve is selected, the frame element(s) on the curve is(are) automatically included for selective display of the diagram. *Shift* key click or rubber-band rectangle may be used for multiple selection.  
Now, the diagram is partially displayed only for the selected members.



The entire diagram is displayed.



The diagram is partially displayed for selected members.

<Displaying part of the diagram for selected members only>

# **Chapter 9**

## **Data Interface with External Software**

## Chapter 9 *Data Interface with External Software*

## Chapter 9 Data Interface with External Software

In most cases, VisualFEA's main data files are handled through its own user interface. Thus, their internal contents are hidden to the end users. In some cases, however, the contents and structures of VisualFEA file are desired to be known for interface between VisualFEA and other software.

As already described in Chapter 6, you may use VisualFEA together with other solvers provided by a third party, or developed by yourself. The data within the VisualFEA file should be delivered directly, or by way of intermediate data interface programs, to external solvers. In other words, it is necessary for the programs to read the data within the VisualFEA file. If it is the case, the file contents and structures of VisualFEA as well as of the interfacing software should be known, as detailed in the following sections.

If the analysis data generated by other software are to be postprocessed by VisualFEA, they should be written in the form readable by VisualFEA. In this case, the software should supply a data file with full ingredient and structure of a VisualFEA data file. The software may customize VisualFEA's postprocessing items by recording appropriate information on the data file.

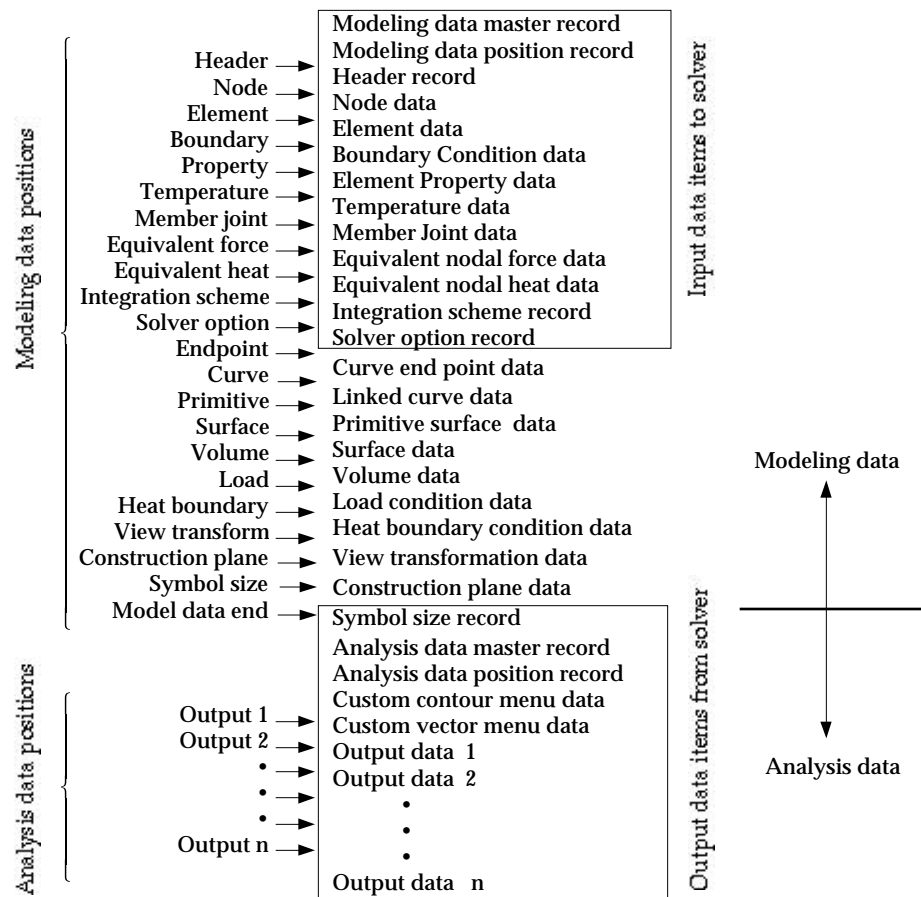
In postprocessing stage, the displayed data items are selected using popup menus within contouring or vector dialog box. There are default popup menus set by VisualFEA. However, the items of the popup menus can be customized by the external software, if necessary. The popup menu related data including menu strings are defined by the external solver and recorded in the VisualFEA file.

VisualFEA files are written using Macintosh or Windows API functions related with file opening, reading and writing. Their contents are not accessible in text form. Computer program codes reading or writing VisualFEA files should be written using Macintosh or Windows API's file utility functions and strictly following the file structure and format described in this chapter.

## Overview of File Contents

It is one of the characteristics of VisualFEA that all data pertaining to the whole procedure of finite element analysis are contained in a single file. A VisualFEA data file consists largely of two parts: modeling data and analysis data. If they are being accessed by an external solver and the analysis results are to be visualized by VisualFEA, the first part becomes the input to the solver and the second part is the output written from the solver.

Some parts of VisualFEA data are dedicated for user interface, graphical rendering and so on. So, not every part of the file is related to solvers as indicated in the following figure. Only the items enclosed within rectangular frames need be accessed from solvers. The other parts of the file may be ignored by external solvers. However, they should be kept in the original form created by VisualFEA, in order to be read by VisualFEA again for postprocessing.



<Overview of VisualFEA file structure>

## File position

The file consists of a number of data items. The beginning of each item is marked by a corresponding file position which indicate the relative location of the data within the file. Both Macintosh and Windows API have functions setting and getting these file positions. Thus, the data need not be read sequentially. Instead, data items can be located, in any order, using the file position record, and read the item starting from the position.

There are two file position records: modeling data position and analysis data position. The file position of the modeling data items are recorded in the modeling data position record, and those for the analysis result items are recorded in the analysis data position record.

The modeling data positions are obtained and recorded while the VisualFEA file is being saved at the end of modeling. The analysis data file positions are picked and written together with analysis results during processing stage, whichever one of VisualFEA's own solver or an external solver is used for processing. If the analysis results computed by external solvers are going to be visualized by VisualFEA, it is the responsibility of the solver to keep track of the file positions and record them together with the analysis data.

## Input data for external solver

If a finite element model created by VisualFEA is to be analyzed by an external solver, the modeling data should be passed to the solver as input data. The following items are the data necessary for finite element processing.

- Node data including nodal coordinates
- Element data including element node connectivities
- Structural boundary conditions for structural analysis
- Element properties such as material properties or section properties
- Temperature distribution, its gradient for structural analysis
- State of frame member joints for frame analysis
- Equivalent nodal forces evaluated from load conditions
- Equivalent nodal heats evaluated from heat conditions
- Number of integration points applied for each element shape and order
- Options to be applied for processing

Before reading the above items, the external solver should read the following data items:

- Modeling data master record
- Modeling data position record

The modeling data master record consists of information on the property of the

file, and the modeling data position record has the relative positions of the above items within the VisualFEA file.

### Output from external solver

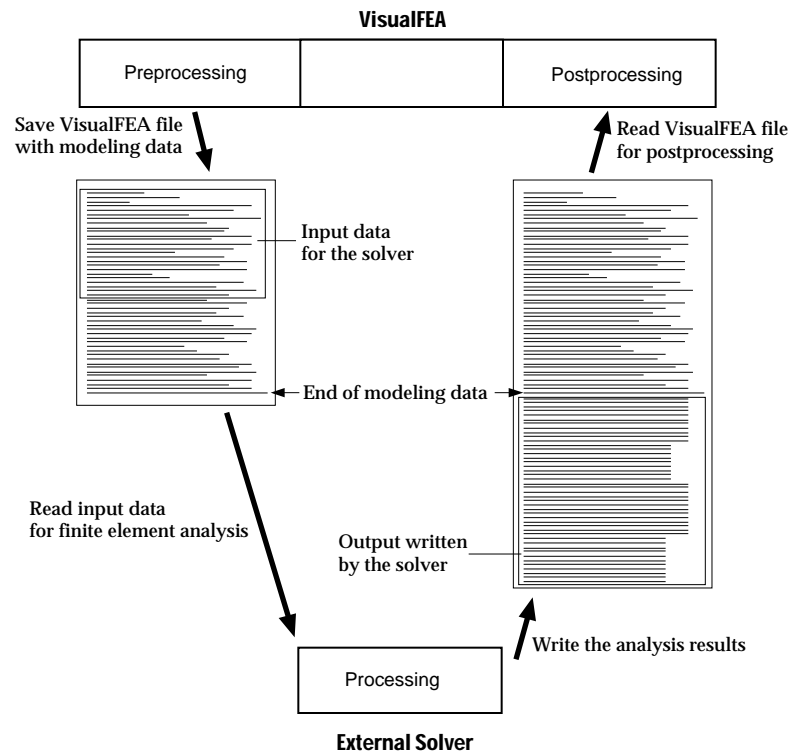
Once the modeling data of a VisualFEA file are read and processed by an external solver, the resulting data can be postprocessed by VisualFEA again. In order to make the output data readable from VisualFEA, they should be written over the original VisualFEA file. In other words, the output data should be appended to the end of the modeling data in the VisualFEA file. The output data items include:

- Data defining custom menu items for contouring
- Data defining custom menu items for vector display
- Data items for graphical visualization

The output part also has the master variables and the file positions, namely:

- Analysis data master variables
- Analysis data file positions

The parts of the VisualFEA file other than above output items should be kept intact by the external solver. Otherwise, Visualfea can neither read nor process the file properly.



<Data flow for combined use of VisualFEA and an external solver>



### **Other data for user interface, graphical modeling and rendering**

VisualFEA file contains not only the data necessary for finite element analysis but also other data related to user interface, graphical modeling, rendering and etc. These data are not directly used in finite element processing, but are essential to maintaining and managing the source of data in graphically interactive work environment. These data can be classified into 3 types: geometric data, source of assignment data, and input and view control data.

The geometric data are graphical data for hierarchical construction of finite element model, and consist of

- Coordinates of curve end points
- Curve division ratio
- Linked curve data
- Primitive surface data
- Surface mesh data
- Volume mesh data

The source of assignment data implies the native data used in interactive work environment. The actual data used in processing are extracted from this source.

- Load condition data
- Heat boundary condition data

The load condition data are used for interactive assignment of load conditions, and are the source of equivalent nodal force data which is actually used infinite element analysis. Likewise, heat boundary condition data are the source of equivalent nodal heat data. The input and view control data are related to viewing transformation, rendering and input environment of graphical models.

- View transformation
- Construction planes
- Symbol variables

There are also the kind of data which serve for the source of interactive graphical modeling as well as for direct use in finite element processing:

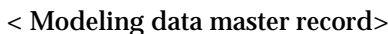
- Node data
- Boundary condition data
- Element property data
- Temperature data
- Member joint data

The data described in this section are also important for interface with other CAD or graphical modeling software. Data may be imported from such software for further processing by VisualFEA. Or, VisualFEA data may be exported for use in other software.

VisualFEA files consist of many data items as described in the previous sections. The structure and the contents of each data item are described in more detail in this section. The contents of VisualFEA files consist of two parts: modeling data and analysis data. Data items of the two parts are indicated in the figure at the beginning of this chapter. The modeling data items are generated at the stage of preprocessing. The analysis data items are obtained as a result of finite element processing, either by VisualFEA or by an external solver.

Modeling data are generated at the preprocessing stage through interactive user operations, and written on VisualFEA file when the file is being saved. The data are written in the file, item by item, in accordance with the sequence described here. But, the position of each item can be located by the modeling data position record, and thus can be read in arbitrary order.

Modeling data master record consists of file identifications and information necessary for reading the file. This record is chiefly used for compatibility between different platform, and between different VisualFEA versions. There are 30 entries of 4 byte long integers in the record as shown in the following figure. The blank spaces are reserved for future use.



- OS flag : The value of this flag is used to identify the platform under which the file was created. If the current and the file's platforms are identical, this

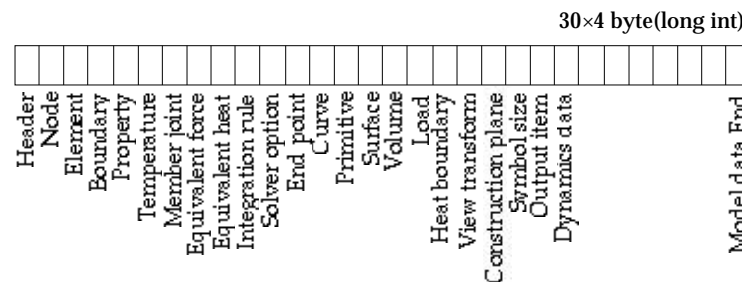
value should read '12345678'. Otherwise, the value gets a different number.

Binary data are represented differently under MacOS and Windows platforms. VisualFEA reads files created under different platform with automatic data conversion.

- VisualFEA version No. : This is to identify the version of VisualFEA which created the file.
- File I.D. : Each file is given an I.D. which is a random number generated at the time when the file is saved.
- Number of variable node data items : A part of node data has record length which may be varied from version to version. Thus, the information on the length is necessary for compatibility between VisualFEA versions.
- Number of variable node data items : A part of node data has record length which may be varied from version to version. Thus, the information on the length is necessary for compatibility between VisualFEA versions. The length is represented by the number of items.
- Number of variable element data items : The length of variable part of the element data. Similar to the above item.
- Number of variable curve data items : The length of variable part of the curve data. Similar to the above item.
- Number of variable surface data items : The length of variable part of the surface data. Similar to the above item.
- Number of variable volume data items : The length of variable part of the volume data. Similar to the above item.

### ■ Modeling data position record

Modeling data position record has the information on the offset distance, in byte, from the beginning of the file to the starting point of each data item. The record has 30 entities of 4 byte long integers. The blank spaces are reserved for future use.



< Modeling data position record >

- Header: The position of header record.
- Node: The position of node data.

- Element: The position of element data.
- Boundary: The position of structural boundary condition data.
- Property: The position of element property data.
- Temperature: The position of temperature data.
- Member joint: The position of frame member joint data.
- Equivalent force : The position of equivalent nodal force data.
- Equivalent heat: The position of equivalent nodal heat data.
- Integration rule : The position of integration rule record.
- Solver option : The position of solver option record.
- End point: The position of curve end point data.
- Curve: The position of curve data.
- Primitive: The position of surface primitive data.
- Surface: The position of surface mesh data.
- Volume: The position of volume mesh data.
- Load: The position of load condition data.
- Heat boundary: The position of heat boundary condition data.
- View transform: The position of view transformation data.
- Construction plane: The position of construction plane data.
- Symbol size: The position of Symbol size record.
- Output item: The position of output item record.
- Dynamics data: The position of Dynamics data record.
- Modeling data end: The position of the end point of the modeling data.

### ■ Header record

Header record has the basic information on the file including the analysis type, sizes of model ingredients, etc. The record has 30 entities of 4 byte long integer. The blank spaces are reserved for future use.

30×4 byte(long int)																													
Analysis subject	Flag of coupled analysis	Number of nodes	Number of active elements	Number of inactive elements	Number of boundary condition sets	Number of element property sets	Number of load condition sets	Number of equivalent nodal loads	Number of member joint sets	Number of heat boundary condition sets	Number of heat convection data	Number of nodal dynamics data sets	Number of load combination sets	Number of curve end points	Number of curves	Number of surface primitives	Number of surface meshes	Number of volume meshes	Number of mesh lines	Number of construction planes	Number of embedded bars	Flag of material nonlinear analysis	Flag of geometric nonlinear analysis	Flag of dynamic analysis	Flag of transient heat analysis	Flag of transient seepage analysis	Flag of sequential analysis		

< Header record >

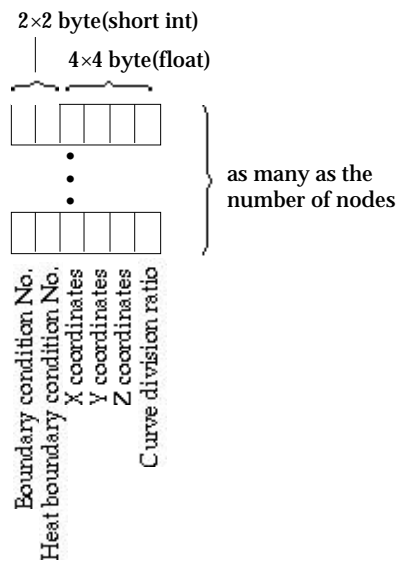
- Analysis subject: The analysis subject is coded by the following numbers. The analysis subject represents the major analysis type which can be mixed with different types of analysis class. (Refer to Chapter 5, "Element Properties".)
  - 0 : plane stress
  - 1 : plane strain
  - 2 : Axisymmetric
  - 3 : plate bending structure
  - 4 : shell structure
  - 5 : 3 dimensional solid structure
  - 6 : 2 dimensional truss
  - 7 : 3 dimensional truss
  - 8 : 2 dimensional rigid frame
  - 9 : 3 dimensional rigid frame
  - 10 : plane heat conduction
  - 11 : axisymmetric heat conduction
  - 12 : 3 dimensional heat conduction
  - 13 : plane seepage
  - 14 : axisymmetric seepage
  - 15 : 3 dimensional seepage
- Number of nodes: The total number of nodes in the model
- Number of active elements: The number of elements used in the finite element analysis. Only elements assigned with element properties are actually included in the analysis.
- Number of inactive elements: The number of elements without element property assignment. The unassigned elements are not included in the analysis, and thus termed as "inactive element."
- Number of boundary condition sets: The number of boundary condition sets which include both assigned and unassigned ones. Unassigned set implies the ones defined for user interface, but not yet assigned to any object.
- Number of element property sets: The number of element property sets which include both assigned and unassigned ones.
- Number of load condition sets: The number of load condition sets which include both assigned and unassigned ones.
- Number of equivalent nodal loads: The number of equivalent nodal loads is the same as the number of nodes with non-zero equivalent nodal load.
- Number of member joint sets: The number of member joint sets which include both assigned and unassigned ones.
- Number of heat boundary condition sets: The number of heat boundary condition sets which include both assigned and unassigned ones.
- Number of heat convection data: The number of heat convection data which

are extracted from heat boundary condition data.

- Number of temperature sets: The number of temperature sets which include both assigned and unassigned ones.
- Number of nodal dynamics data sets: The number of nodal dynamics data sets which include both assigned and unassigned ones.
- Number of curve end points: The total number of end points which define the starting and ending points of curves.
- Number of curves: The total number of curves, both created and generated.
- Number of surface primitives: The total number of surface primitives.
- Number of surface meshes: The total number of surface meshes, both created and generated.
- Number of volume meshes: The total number of volume meshes, both created and generated.
- Number of mesh lines: The total number of straight lines consisting the wireframe meshes.
- Number of construction planes: The number of construction planes defined for aiding the coordinate data.
- Flag of nonlinear analysis: The value indicates whether the analysis is linear or nonlinear: 0 for linear analysis and 1 for nonlinear analysis.
- Flag of dynamic analysis: The value indicates whether the analysis is static or dynamic: 0 for static analysis and 1 for dynamic analysis.
- Number of frontal buffers: The number of data buffers used in frontal solution process.

#### ■ Node data

Node data consist of records with nodal coordinates and assignment information, One record contains following information for a node. There are as many records as the number of nodes.



#### <Node data>

- **Boundary condition No.:** The No. of the structural boundary condition set assigned to the node. This value is zero based. (*Boundary condition sets are numbered starting from 0.*) If this value is -1, the node is not assigned with boundary condition. (2 byte short integer)
- **Heat boundary condition No.:** The No. of the heat boundary condition set assigned to the node. This value is zero based. If this value is -1, the node is not assigned with heat boundary condition. (2 byte short integer)
- **X, Y, and Z coordinates:** The coordinates of the node in X, Y, Z Cartesian coordinate system. (4 byte float for each one of X, Y and Z coordinates)
- **Curve division ratio:** The relative position of the node on the curve. The value is between 0 and 1.0; the node at the starting point of the curve has the value of 0, and the one at the end point of the curve has the value of 1.0. The nodes between the two have the value representing the normalized distance from the starting point.

This value has meaning only for the nodes created by dividing curves. This value is not used for solver. This information is used for future modification of the curve. The curve division ratios are retained after modification of curves. (4 byte float)

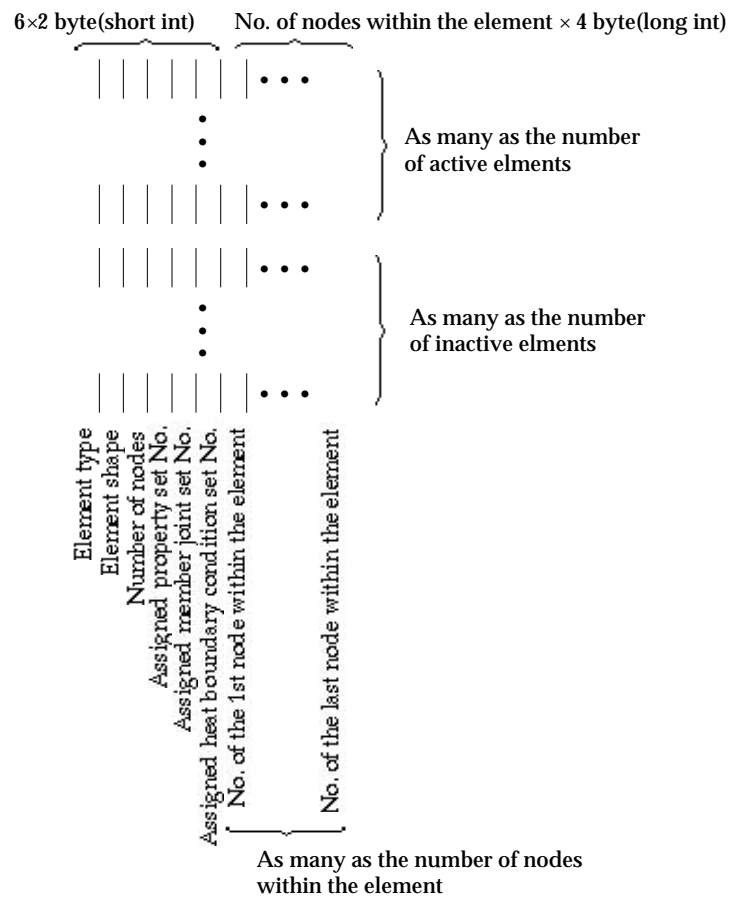
#### ■ Element data

Element data consist of records with nodal connectivities and assignment information. One record contains following information for an element. There are records for active elements followed by the records for inactive elements. Active elements are those actually involved in finite element processing. All the elements

assigned with element properties are active. Inactive elements are not used in the processing, but constitute curve divisions, surface meshes or volume meshes. VisualFEA represents the segment of curve divisions, surface meshes and volume meshes in the form of element data. These segments become active elements when they are assigned with element properties.

- Element type: Analysis type of the element. Only active elements have element type. (2 byte short integer)
  - 1 : not an element type (for inactive elements)
  - 0 : plane stress
  - 1 : plane strain
  - 2 : Axisymmetric
  - 3 : plate bending structure
  - 4 : shell structure
  - 5 : 3 dimensional solid structure
  - 6 : 2 dimensional truss
  - 7 : 3 dimensional truss
  - 8 : 2 dimensional rigid frame
  - 9 : 3 dimensional rigid frame
  - 10 : plane heat conduction
  - 11 : axisymmetric heat conduction
  - 12 : 3 dimensional heat conduction
  - 16 : interface element
  - 17 : embedded bar
- Element shape: The shape of the element (2 byte short integer)
  - 1 : 2 node line element
  - 2 : 3 node line element
  - 3 : 3 node triangular element
  - 4 : 6 node triangular element
  - 5 : 4 node quadrilateral element
  - 6 : 8 node quadrilateral element
  - 7 : 4 node tetrahedral element
  - 8 : 10 node tetrahedral element
  - 9 : 6 node prism element
  - 10 : 15 node prism element
  - 11 : 8 node hexahedral element
  - 12 : 20 node hexahedral element



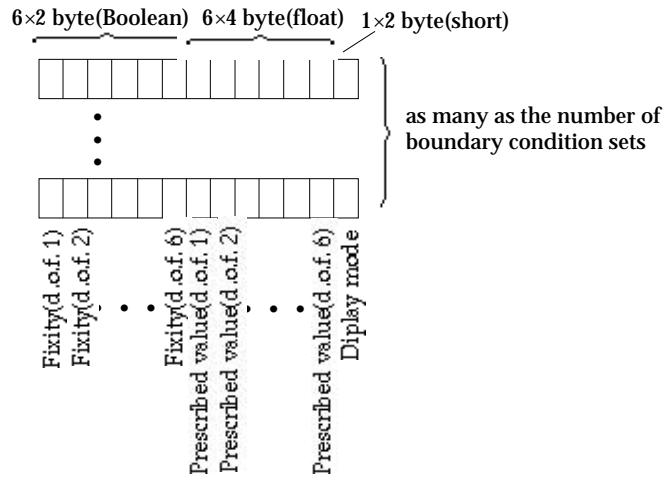


<Element data>

- **Number of nodes:** The number of nodes in the element. (2 byte short integer)
- **Assigned property set No.:** The zero based No. of the element property set assigned to the element. If this value is -1, the element is not assigned with element property set. (2 byte short integer)
- **Assigned member joint set No.:** The zero based No. of the member joint set assigned to the element. If this value is -1, the element is not assigned with member joint set. (2 byte short integer)
- **Assigned heat boundary condition set No.:** The zero based No. of the heat boundary condition set assigned to the element. If this value is -1, the element is not assigned with heat boundary condition set. (2 byte short integer)
- **Node No.:** The zero based No. of each node within the element. (number of nodes within the element x 4 byte long integer )

### ■ Boundary condition data

The boundary condition data consist of records with structural fixities and either initial displacements or spring constants of the node. One record contains information for one boundary condition set as follows.

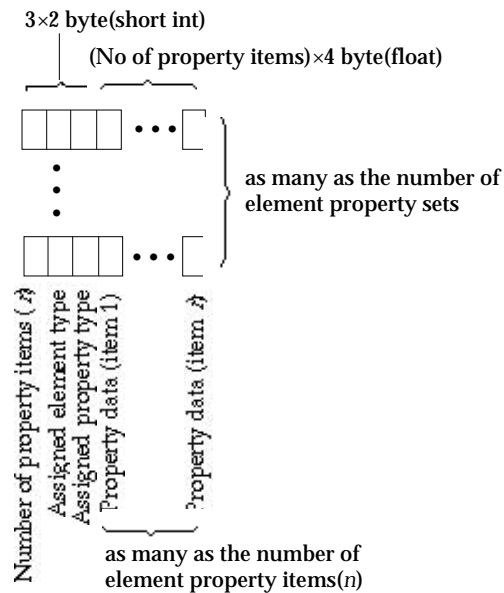


<Boundary condition data>

- **Fixity:** Structural fixity or restraint of each nodal d.o.f. at a node. There are 6 entries of fixity data, but only the entries corresponding to the actual nodal d.o.f. are in use. ( $6 \times 2$  byte short integer)
  - 0 : fixed
  - 1 : free
- **Prescribed value:** Value prescribed for each nodal d.o.f. at a node. There are 6 entries of prescribed values, but only the entries corresponding to the actual nodal d.o.f. are in use. The prescribed values are either initial displacements or spring constants. If the nodal d.o.f. is fixed, the prescribed value represent the initial displacement of the corresponding d.o.f. Otherwise, the value represents the spring constant assigned to the corresponding d.o.f. ( $6 \times 4$  byte float)
- **Display mode:** mode of displaying the prescribed values on “Struct Boundary” dialog. (Refer to Chapter 5) The edit text items of initial displacement, spring constant or both are displayed on the dialog depending on the mode.
  - 0 : Only edit text boxes of initial displacements are displayed.
  - 1 : Only edit text boxes of spring constants are displayed.
  - 2 : All edit text boxes are displayed.

## ■ Element property data

The element property data have a variable number of data items. The number of data items and the entity of each item are determined by the element type defined for the set, and appear on “Property” dialog.



### <Element property data>

- **Number of property items:** The number of element property items are variable, and retrieved by this entry. This value is originally determined by the element type at the stage of preprocessing, but saved and retrieved as an entry of element property data for quick retrieval of the data items. (2 byte short integer)
- **Assigned element type:** The element type is primarily determined by the analysis type. However, VisualFEA allows mixed use of different element types in a model. (2 byte short integer)
  - 1 : not an element type (for inactive elements)
  - 0 : plane stress
  - 1 : plane strain
  - 2 : Axisymmetric
  - 3 : plate bending structure
  - 4 : shell structure
  - 5 : 3 dimensional solid structure
  - 6 : 2 dimensional truss
  - 7 : 3 dimensional truss
  - 8 : 2 dimensional rigid frame
  - 9 : 3 dimensional rigid frame

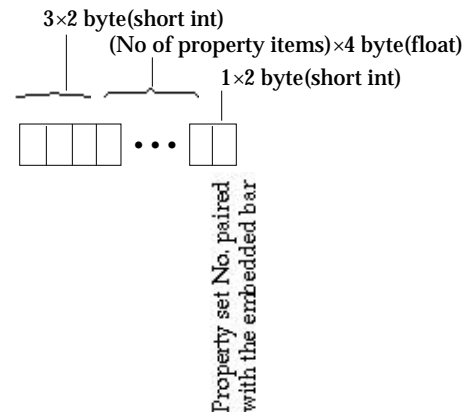
- 10 : plane heat conduction
- 11 : axisymmetric heat conduction
- 12 : 3 dimensional heat conduction
- 13 : plane seepage
- 14: axisymmetric seepage
- 15 : 3 dimensional seepage
- 16 : interface element
- 17 : slip bar
- 18 : embedded bar
- Assigned property type: The type of material property (2 byte short integer)
  - 0 : linear elastic and isotropic
  - 1 : linear elastic and orthotropic
  - 2 : elasto-plastic( appears only for nonlinear analysis)
  - 3 : friction( appears only for interface elements of nonlinear analysis)
- Property data items: The number of data items is retrieved as an entry of a data record, and the entity of each item is determined by the element type as shown in the following table. (number of data items  $\times$  4 byte float)

&lt;Entities of element property items&gt;

Element type	Element property items								
	1	2	3	4	5	6	7	8	9
Plane stress	$E_x$	$E_y$	$\nu_{xy}$	$t$	$w_o$	$\alpha$			
Plane strain	$E_x$	$E_y$	$\nu_{xy}$	$w_o$	$\alpha$				
Axisymmetric	$E_x$	$E_y$	$\nu_{xy}$	$w_o$	$\alpha$				
Plate	$E_x$	$E_y$	$\nu_{xy}$	$t$	$w_o$	$\alpha$			
Shell	$E_x$	$E_y$	$\nu_{xy}$	$t$	$w_o$	$\alpha$			
Solid	$E_x$	$E_y$	$E_z$	$\nu_{xy}$	$\nu_{yz}$	$\nu_{zx}$	$w_o$	$\alpha$	
Interface element	$E_L$	$E_g$	gap	$t$	*				
Embedded bar	$E_{bar}$	$E_{bond}$	$A$						
Truss 2D & 3D	$E$	$A$	$w_o$	$\alpha$					
Frame 2D	$E$	$A$	$I$	$w_o$	$\alpha$				
Frame 3D	$E$	$\nu$	$A$	$I_y$	$I_z$	$J$	$w_o$	$\alpha$	$\beta$
Plane heat	$k_x$	$k_y$							
Axisymmetric heat	$k_x$	$k_y$							
Volume heat	$k_x$	$k_y$	$k_z$						

\* for plane stress only

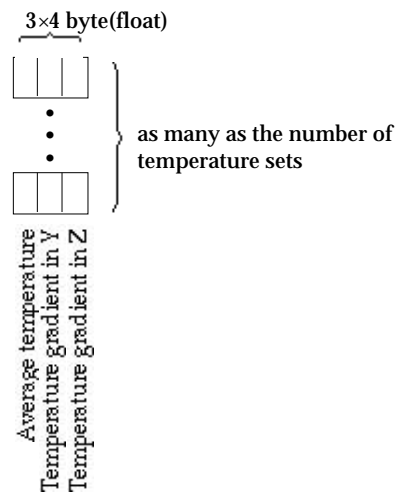
In case the element type is embedded bar, one more data item is added as shown below. The additional item points to the set No. of the property surrounding the embedded bar.



<Element property data for embedded bar>

## ■ Temperature data

Temperature data are currently applicable only to 2-D and 3-D frame analysis. The data consist of the average temperature of a frame member and the temperature gradient across the member section.



<Temperature data>

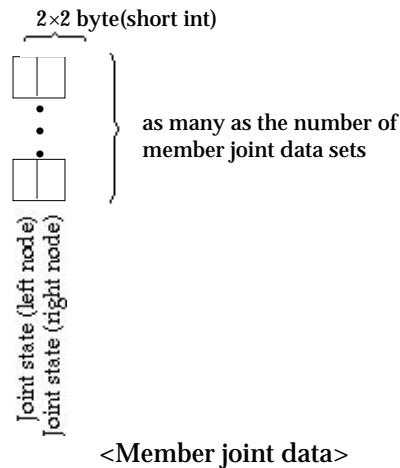
- Average temperature: Average temperature of the frame member(4 byte float)
- Temperature gradient in Y: Temperature gradient across the member section in Y direction. The gradient is equivalent to the temperature difference between top and bottom of the section divided by the height of the section. (4 byte float)
- Temperature gradient in Z: Temperature gradient across the member section in Z direction. This item is used only for 3-D frames. (4 byte float)

Temperature data are not supported by user interface in the current version of VisualFEA. Instead, they are treated as thermal load conditions. Refer to Chapter 5. However, their space is left in VisualFEA file for use by external software.

### ■ Member joint data

Member joint data are currently applicable to 2-D and 3-D frame analysis. One record of member joint data has only 2 data items: joint states at both end of the member.

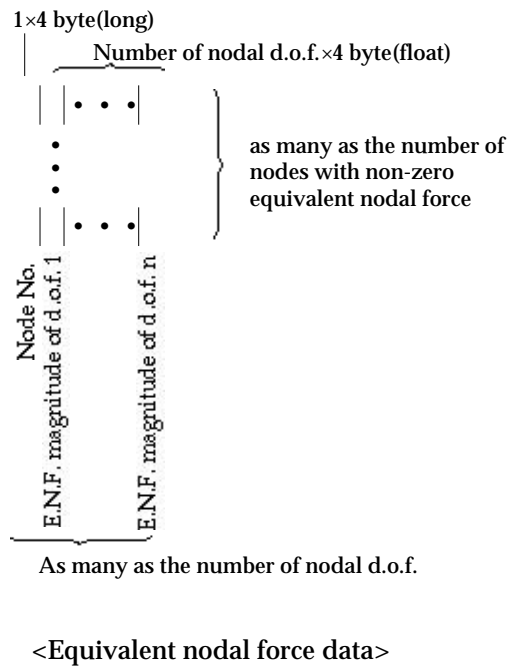
- Joint state (left node): The state of joint at the starting point of the member. (2 byte short int)  
     0 : Rigid joint  
     1 : Pin joint
- Joint state (right node): The state of joint at the ending point of the member. (2 byte short int)



### ■ Equivalent nodal force data

The equivalent nodal forces are computed at each node from the load condition data, which is described in one of the later sections. The equivalent nodal forces are directly involved in finite element analysis, while the load condition data are maintained chiefly for graphical user interface. The data has records only for nodes with non-zero equivalent nodal forces. The components of the equivalent nodal forces are as many as the number of nodal d.o.f.

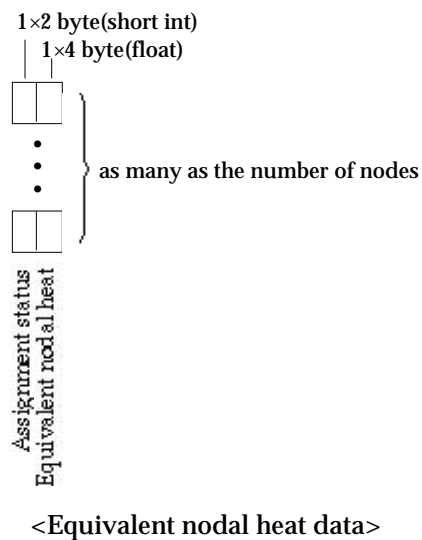
- Node No.: The node No. identifying the node for which the equivalent nodal forces are evaluated. (4 byte long int)
- Equivalent nodal force magnitude: Magnitude of each force component which matches with each one of the nodal d.o.f. (number of nodal d.o.f.  $\times$  4 byte float)



### ■ Equivalent nodal heat data

This data portion consists of two parts: equivalent nodal heat and convection boundary condition data. Both parts are evaluated from heat boundary condition data.

The equivalent nodal heat data has records for all nodes in the model. Each record has only 2 data entries:



- Assignment status: Flag indicating whether the node is assigned with nodal

heat. (2 byte short int)

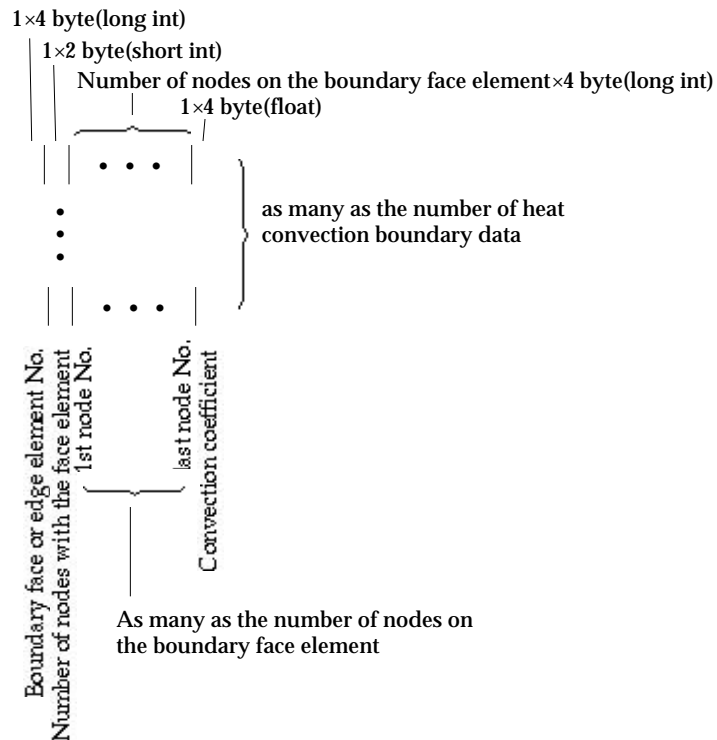
0 : Assigned

1 : Not assigned

- Equivalent nodal heat: Magnitude of nodal heat (4 byte float)

In the second part are the convection boundary condition data, which are evaluated from convection type of heat boundary condition. The data consist of convection coefficients and information on boundary face or edge on which the convection boundary condition is applied.

- Boundary face or edge element No.: The convection boundary conditions are applied to the inactive elements on boundary edges (2-D case) or on boundary faces (3-D case). This data entry has the element No. on the boundary edge or face. (4 byte long int)
- Number of nodes within the face or edge element: The face or edge consists of a number of nodes. This entry has the number of nodes within the element. (4 byte long int)
- Node No.: All the node No. consisting the boundary face or edge element. (number of nodes within the boundary element  $\times$  4 byte long int)
- Convection coefficient: The convection coefficient of the heat boundary condition set from which the equivalent nodal heat is evaluated. (4 byte float)





### ■ Nodal dynamics data

The number of nodal dynamics data sets is included in the data header record, and is always zero for static analysis. The following represents only one set of nodal dynamics data records. These data records occur as many times as the number of nodal dynamics data sets. Each set of data records starts with an attribute classifying the type of nodal dynamics data.

- Type of nodal dynamics data: This data entry represent the type of nodal dynamics data as follows:
  - 1 : Nodal dynamic motion - displacement
  - 2 : Nodal dynamic motion - velocity
  - 3 : Nodal dynamic motion - acceleration
  - 4 : Nodal dashpot constant
  - 5 : Nodal mass

The contents for the rest of the nodal dynamics data records are different depending on the type of nodal dynamics data. In case of nodal dashpot and nodal mass, the data consist of 6 entries which represent the values for each nodal d.o.f.

1×2 byte(short int)



Type of nodal dynamics data

<Attribute classifying the type of nodal dynamics data>

### ■ Integration scheme record

The integration scheme is set independently for each shape and each order of element. The number of integration points set for various element shapes are saved in the integration scheme data record as shown in the following figure. There are 30 entries of 4 byte long integer in the record. The blank spaces are reserve for future use.

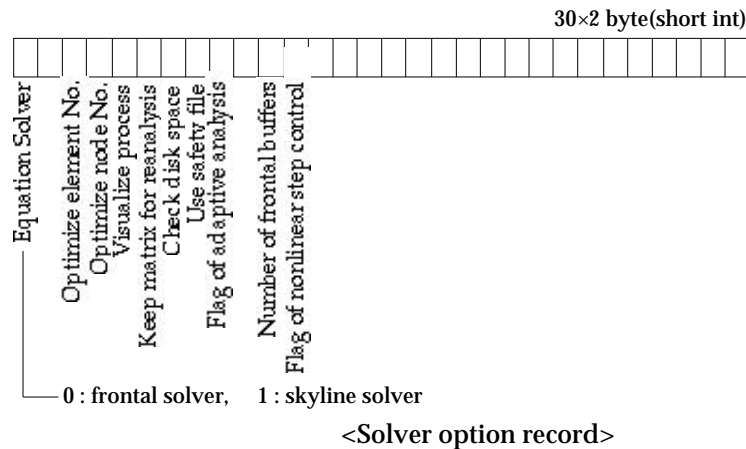
[illegible]

For 3 node triangle element  
For 6 node triangle element  
For 4 node quadrangle element  
For 8 node quadrangle element  
For 4 node tetrahedron element  
For 10 node tetrahedron element  
For 6 node prism element  
For 15 node prism element  
For 8 node hexahedron element  
For 20 node hexahedron element  
For 6 node triangle shell element  
For 8 node quadrangle shell element  
For 4 node quadrangle shell element  
For 3 node triangle shell element

- For 3 node triangle element: Number of integration points set for a 3 node triangle element. It is either of 1, 3 or 7.
- For 6 node triangle element: Number of integration points set for a 6 node triangle element. It is either of 1, 3 or 7.
- For 4 node quadrangle element: Number of integration points set for a 4 node quadrangle element. It is either of 1, 4 or 9.
- For 8 node quadrangle element: Number of integration points set for a 8 node quadrangle element. It is either of 1, 4 or 9.
- For 4 node tetrahedron element: Number of integration points set for a 4 node tetrahedron element. It is either of 1, 4 or 5.
- For 10 node tetrahedron element: Number of integration points set for a 10 node tetrahedron element. It is either of 1, 4 or 5.
- For 6 node prism element: Number of integration points set for a 6 node prism element. It is either of 1, 2 or 6.
- For 15 node prism element: Number of integration points set for a 15 node prism element. It is either of 1, 2 or 6.
- For 8 node hexahedron element: Number of integration points set for a 8 node hexahedron element. It is either of 1, 8 or 27.
- For 20 node hexahedron element: Number of integration points set for a 20 node hexahedron element. It is either of 1, 8 or 27.

There are a number of options which are applied for finite element processing. They are saved in the solver option record with 30 entries of 2 byte short integer. Each entry has value of 0 or 1. The option is on if the value is 1 and off otherwise,

except the equation solver option.



- **Equation solver:** One of skyline solver and frontal solver is selected as the equation solver.
  - 0 : Frontal solver
  - 1 : Skyline solver
- **Optimize element No.:** Element number optimization is executed before finite element processing begins, if this option is on.
- **Optimize node No.:** Node number optimization is executed before finite element processing begins, if this option is on.
- **Visualize process:** The progress of finite element processing is graphically visualized, if this option is on.
- **Keep matrix for reanalysis:** The relevant matrix files necessary for reanalysis are not removed at the end of the processing, if this option is on.
- **Check disk space:** The available disk space is continuously checked while the processing is going on, if this option is on.
- **Use safety file:** A temporary file is created and used while processing is going on, in order to protect the original file from being damaged under abnormal situation, if this option is on.
- **Flag adaptive analysis:** An iterative analysis process with adaptive mesh generation is applied if this option is on.
- **Number of frontal buffer:** This entry specifies the number of matrix data used for frontal solver.
- **Flag of nonlinear step control:** If this flag is on, the rate load increment is specified individually for each of the load step.

## ■ Analysis output item record

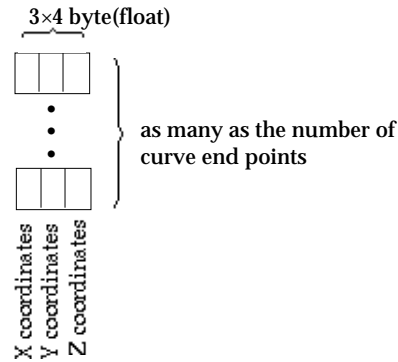
Each entry has value of 0 or 1. If the value is 1, corresponding item is turned on as an analysis output item. There are 30 entries of 2 byte short integer in the record. The blank spaces are reserved for future use.





### ■ Curve end point data

Every curve has two end points, a starting point and a ending point. The coordinates of these end points are saved as curve end point data.

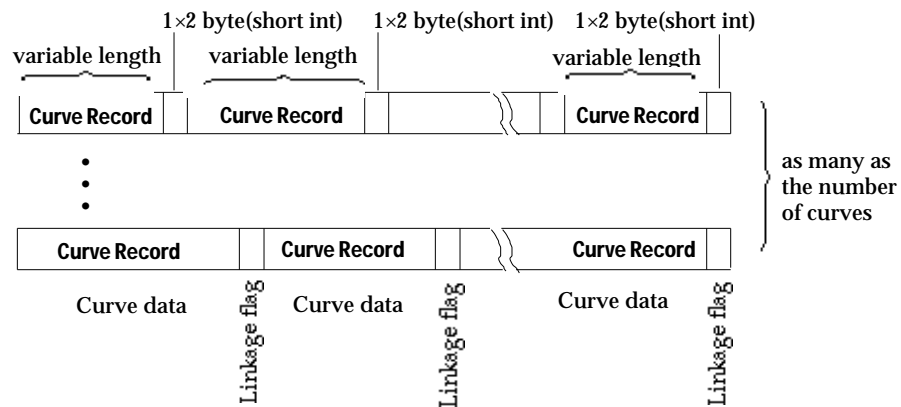


<Curve end point data>

- X, Y, and Z coordinates: The coordinates of the end point in X, Y, Z Cartesian coordinate system. (4 byte float for each one of X, Y and Z coordinates)

### ■ Curve data

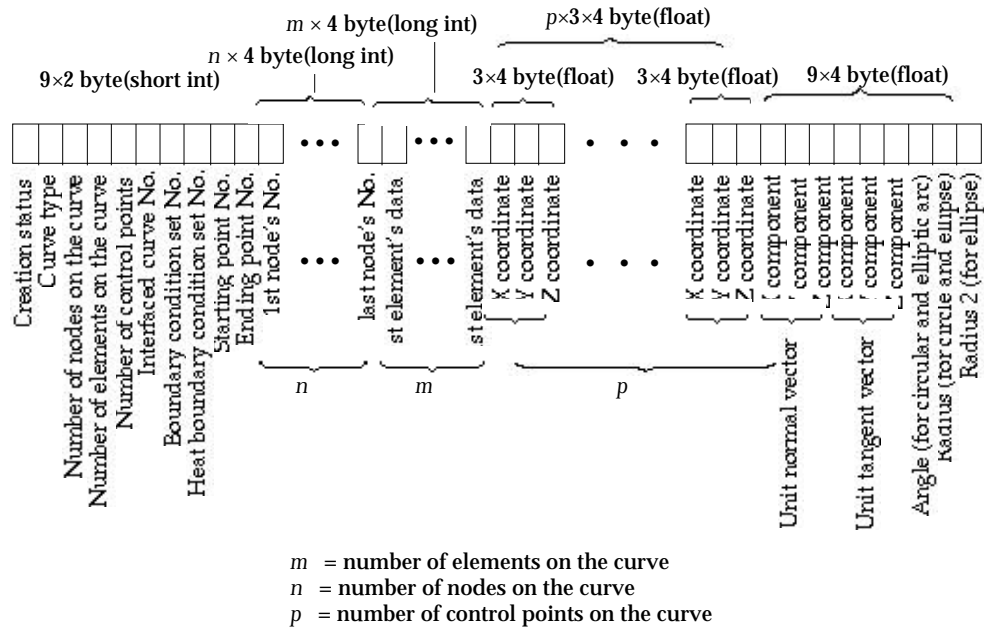
A curve exists either as a single entity or as multiple entities linked together. The curve data are generalized in the form of linked curve as shown in the following figure.



<Linked curve data>

- Curve record: A curve record has information on a single curve as described below. (variable length)
- linkage flag: flag indicating if the curve is linked at the end. (2 byte short int)
  - 0 : No further linkage at the end. The flag is always 0 at the end of linked curves.
  - 1 : curve linked at the end. The flag is always 1 in the middle of linked curves.

A curve record contains information on the geometric composition and the attribute assignment, and other characteristics of a single curve. A curve is composed of end points, control points, nodes and elements.



#### <Curve record>

- **Creation status:** A curve may be either created by user interaction inputting control points, or generated in the process of finite element mesh generation. The boundary of surface meshes are always defined by curves. The creation status indicates whether the curve was created or generated. (2 byte integer)
  - 0 : Generated.
  - 1 : Created
- **Curve type:** This value represent the type of the curve as follows.
  - 1 : Not a valid curve.
  - 0 : Point
  - 1 : Straight line
  - 10 : Acute arc
  - 11 : Clockwise arc
  - 12 : Counter clockwise arc
  - 13 : Three point arc
  - 14 : Center angle arc
  - 16 : Clockwise circle
  - 17 : Counter clockwise circle

- 18 : Three point circle
- 19 : Center radius circle
- 21 : Quarter ellipse
- 22 : Half ellipse
- 23 : Full ellipse
- 31 : Cubic spline curve
- 32 : B-spline curve
- 33 : Bezier curve
- 34 : Polynomial curve
- 35 : Polyline
- 36 : Segmented curve
- 41 : Rectangle

- Number of nodes on the curve: The number of nodes which were created by dividing the curve, or generated together with the curve.
- Number of elements on the curve: The number of elements which were created by dividing the curve, or generated together with the curve.
- Number of control points on the curve: The number of control points which define the geometry of the curve.
- Interface curve No.: When interface elements are assigned along the curve, a duplicate of the curve is generated to form another boundary of the interface gap. In this case, this entry has the pairing interface curve No.
- Boundary condition set No.: If the curve is assigned with a boundary condition, this entry has its No. which is zero based. If boundary condition is not assigned, the value is -1.
- Heat boundary condition set No.: If the curve is assigned with a heat boundary condition, this entry has its No. which is zero based. If heat boundary condition is not assigned, the value is -1.
- Starting point No.: The No. of the end point serving as the starting point of the curve.
- Ending point No.: The No. of the end point serving as the ending point of the curve.
- Node No.: The No. of each node on the curve. This value is zero based. In other words, the node No. starts from 0.
- Element No.: The No. of each element on the curve. This value is zero based. In other words, the element No. starts from 0.
- Coordinates of control points: X, Y and Z coordinates of control points on the curve. Z value is zero for 2 dimensional case.
- Unit normal vector: This entry applies only to circular arc, circle, elliptical arc and ellipse. The directional cosines of the unit normal vector to the curve are saved respectively in X, Y and Z components.
- Unit tangent vector: This entry applies only to closed cubic spline, B-spline or

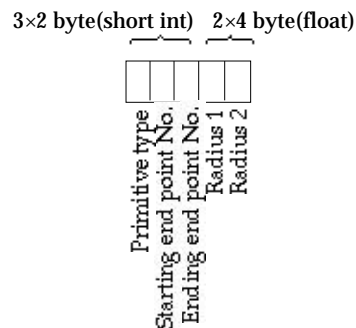


Bezier curves. The unit tangent vector is tangent to the curve at the starting or the ending point. The directional cosines of this unit tangent vector are saved respectively in X, Y and Z component.

- Angle : This entry applies only to circular arc and elliptical arc. Angle implies the central angle of the curve.
- Radius: This entry applies only to circle and ellipse. This is the radius of a circle, and first radius of an ellipse.
- Radius 2: This entry applies only to ellipse. This is the second radius of an ellipse.

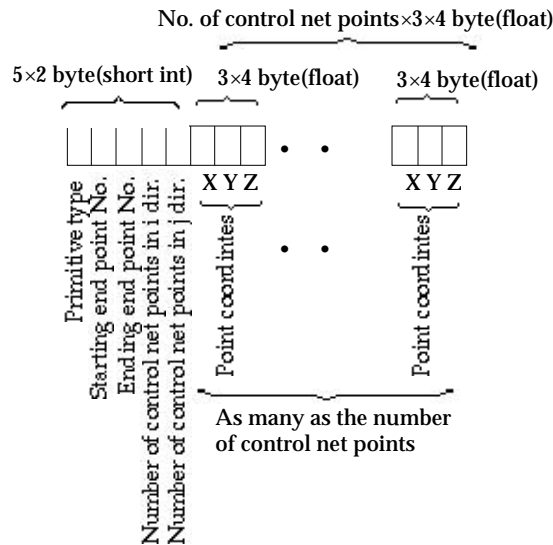
### ■ Primitive surface data

Primitive surface data consists of primitive records. Of course, there are as many records as the number of primitive surfaces. The records have two different compositions depending on the type of the primitives. The primitive record of sphere, cylinder, and torus is one kind, and that of plane, Lagrangian surface, B-spline surface and Bezier surface is another kind. The primitive record for the first kind is shown in the following figure.



<Primitive record for sphere, cylinder and torus>

- Primitive type: This entry has the code indicating the type of the primitive.
  - 82 : Sphere
  - 83 : Cylinder
  - 84 : Torus
- Starting end point No.: The No. of the end point serving as the starting point of the primitive.
- Ending end point No.: The No. of the end point serving as the ending point of the primitive.
- Radius 1: The first radius of the primitive surface
- Radius 2: The second radius of the primitive surface. Sphere has only one radius, thus this value is ignored in sphere data.



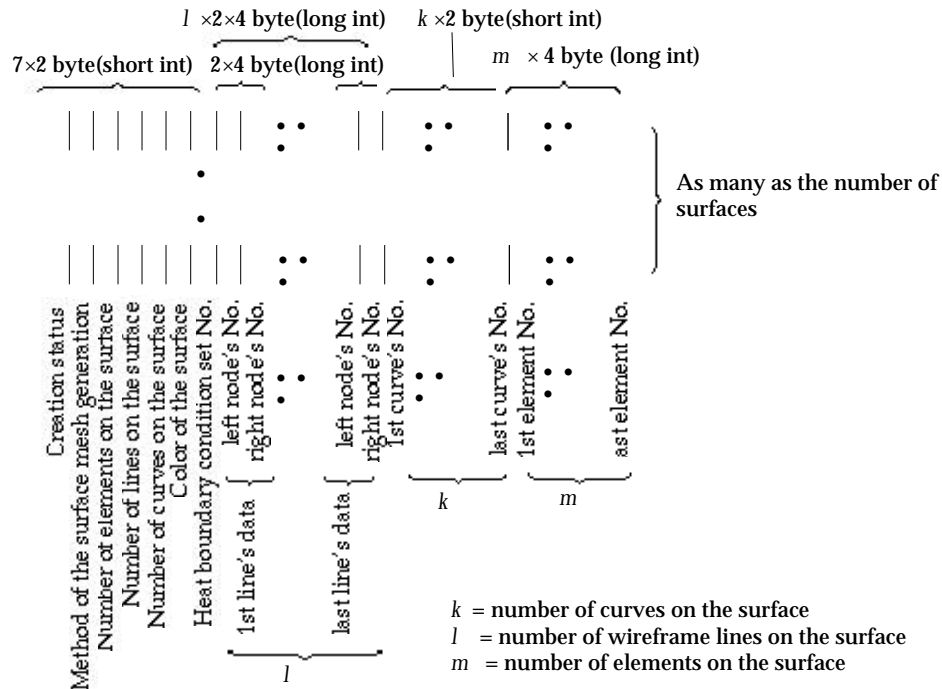
<Primitive record for plane, Lagrangian, B-spline and Bezier surfaces>

- **Primitive type:** The type of the primitive
  - 85 : B-spline surface
  - 86: Bezier surface
  - 87 : Lagrangian surface
  - 90 : Flat plane
- **Starting end point No.:** The No. of the end point serving as the starting point of the primitive.
- **Ending end point No.:** The No. of the end point serving as the ending point of the primitive.
- **Number of control net points in i direction:** The primitive surface is defined by  $m \times n$  control net points where  $m$  and  $n$  represent the number of control points in i and j direction. This entry has the value of  $m$ .
- **Number of control net points in j direction:** This entry has the value of  $n$ .
- **Control net point coordinates:** The X, Y, and Z coordinates of each of  $m \times n$  control net points.

## ■ Surface mesh data

Surface data have the information on the surface meshes including their compositions, attribute assignments, and other characteristics.

A surface mesh is composed of nodes, elements, wireframe lines and boundary curves. Information on the nodes on the surface can be derived from that of the elements on the surface. And therefore, information on the nodes are not included in the surface data.



#### <Surface data>

- **Creation status:** A surface mesh may be either created by user interaction using surface mesh generation command, or generated in the process of volume mesh generation. The boundary of volume meshes are always defined by surface meshes.

The creation status indicates whether the surface mesh was created or generated.

0 : Generated.

1 : Created

- **Method of surface mesh generation:** The entry has the code representing the method of surface mesh generation which was applied to create the surface.

0 : Surface mesh generation by 2 edge mapping

1: Surface mesh generation by extrusion

2: Surface mesh generation by 4 edge mapping

3: Surface mesh generation by 3 edge mapping

4: Surface mesh generation by revolution

5: Surface mesh generation by translation

12: Surface mesh generation by automatic triangulation

13: Surface mesh generation by automatic tetrahedronization

14: Volume mesh generation by duplication with "Move"

15: Surface mesh generation by duplication with "Move"

18: Surface mesh generation by duplication with "Revolve"

21: Surface mesh generation by duplication with “Mirror”

24: Surface mesh generation by twisting

28: Surface mesh generation by duplication with “Extrude”

29: Surface mesh generation by projection

- Number of elements on the surface: Number of all elements, either active or inactive, composing the surface.
- Number of lines on the surface: Number of all lines representing the wireframe of the surface mesh.
- Number of curves on the surface: Number of all curves attached to the surface.
- Color of the surface: The color assigned to the surface. (currently not in use, but reserved for future use)
- Heat boundary condition set No.: No. of the heat boundary condition set assigned to the surface.
- Line data: Information on the lines forming the wireframe. A line is represented by two end nodes. Thus, the data for a line consist of 2 entries; the 2 end node's No.
- Curve No.: The No. of each curve in the surface mesh.
- Element No.: The No. of each element in the surface mesh.

#### ■ Volume mesh data

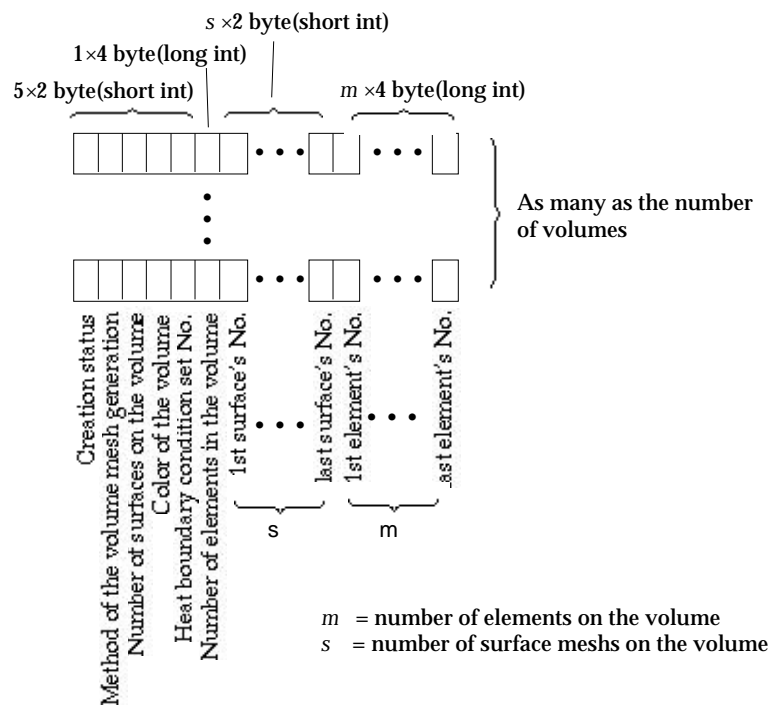
Volume data have the information on the volume meshes including their compositions, attribute assignments, and other characteristics.

*A surface mesh is composed of nodes, elements, surface meshes. Information on the nodes on the volume can be derived from that of the elements on the volume. And therefore, information on the nodes are not included in the volume data.*

- Creation status: Not used in the current version. Reserved for future use.
- Method of volume mesh generation: The entry has the code representing the method of volume mesh generation which was applied to create the volume.
  - 6: Volume mesh generation by 2 surface mapping
  - 7: Volume mesh generation by extrusion
  - 8: Volume mesh generation by box edge mapping
  - 9: Volume mesh generation by tetrahedron edge mapping
  - 9: Volume mesh generation by revolution
  - 10: Volume mesh generation by translation
  - 17: Volume mesh generation by duplication with “Revolve”
  - 20: Volume mesh generation by duplication with “Mirror”
  - 27: Volume mesh generation by prism edge mapping
  - 23: Volume mesh generation by twisting
- Number of surfaces on the volume : The number of surfaces attached to the

volume. The boundary surface of a volume is composed of surface meshes.

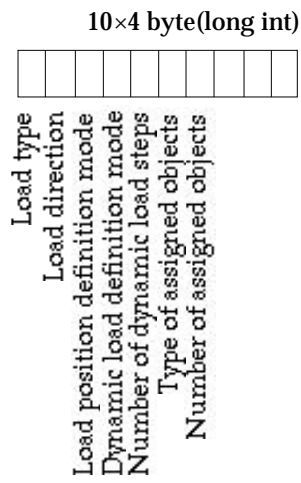
- Color of the volume : Code of the color assigned to the volume. This entry is not used in the current version, but reserved for future use.
- Heat boundary condition set No.: No. of the heat boundary condition set assigned to the volume mesh.
- Number of elements in the volume: Number of all elements, either active or inactive, composing the volume mesh.
- Surface No.: The No. of each surface in the volume mesh.
- Element No.: The No. of each element in the volume mesh.



<Volume data>

### ■ Load condition data - header record

These data are maintained chiefly for graphical user interface, and involved indirectly, through equivalent nodal forces, in finite element analysis. They are also the source from which the equivalent nodal forces are computed. The contents of load conditions vary depending on their type as described below.



< Load condition data >

- Load type: The code representing the type of the load.
  - 0: Nodal force
  - 1: Point force
  - 2: Uniform force
  - 3: Non-uniform force with linear variation
  - 4: Non-uniform force with parabolic variation
  - 5: Bilinear force
  - 6: Nodal moment
  - 7: Point moment
  - 8: Uniform moment
  - 9: Body force
  - 10: Hydrostatic in X
  - 11: Hydrostatic in Y
  - 12: Hydrostatic in Z
  - 13: Self straining force
  - 14: Thermal force
- Load direction: The code representing the direction of the load.
  - 0: X direction
  - 1: Y direction
  - 2: Z direction
  - 3: Direction normal to curve
  - 4: Direction tangent to curve
  - 5: Direction normal to surface
  - 6: Direction tangent to surface and normal to X direction
  - 7: Direction tangent to surface and normal to Y direction
  - 8: Direction tangent to surface and normal to Z direction

- Load position definition mode: This entry designates how the position of the load is defined. This applies only to mid-point force or mid-point moment, and is ignored for other load types. The position of mid-point force or moment can be defined either by the actual distance from one end of the structural member, or by the relative distance represented by the ratio to the length of the member. Distance ratio is equivalent to the distance divided by the total length of the member. The load position definition mode can be applied also for a curve instead of a member.
  - 0: Actual distance
  - 1: Relative distance
- Dynamic load definition mode: Time dependent dynamic loads can be defined either as a sinusoidal harmonic load or as a transient load.
  - 0: Harmonic load
  - 1: Transient load
- Number of dynamic load steps: The number of steps representing the dynamic load history. This data entry is effective only for transient load, and ignored for static loads or harmonic loads.
- Type of assigned objects: The type of the object to which the load condition is assigned.
  - 1: Not assigned
  - 1: Curve
  - 2: Surface
  - 3: Volume
  - 4: Element
  - 5: Node
  - 7: Surface primitive
  - 11: Line element
  - 12: Surface element
  - 13: Volume element
- Number of assigned objects: The type of the object to which the load condition is assigned.

#### ■ Load condition data - attribute record

This data record consists of load attributes such as its magnitude, position and etc. Only one of the following 3 records is applied depending on whether the load is static, dynamic transient, or dynamic harmonic,

4×4 byte(float)



Value 1  
Value 2  
Value 3  
Value 4

&lt; Load attribute record for a static load&gt;

4×4 byte(float)



•  
•  
•

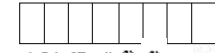


Value 1  
Value 2  
Value 3  
Value 4

As many as the number  
of load time steps

&lt; Load attribute record for a transient load&gt;

4×4 byte(float)



Value 1  
Value 2  
Value 3  
Value 4  
Starting time  
Ending time  
Phase lag time  
Angular velocity(radian/sec)

&lt; Load attribute record for a harmonic load&gt;

- Value 1, Value 2, Value 3, Value 4: These entries are data entered using the editable text items of “Load Condition” dialog. Their contents vary depending on the load type as shown below. Also refer to Chapter 5.
- Starting time: The time from which the harmonic load becomes effective.
- Ending time: The time when the harmonic load ends.
- Phase lag time( $t_0$ ): Phase lag to the starting point of the sinusoidal loading curve.
- Angular velocity( $\omega$ ): Radian per second.

A harmonic load is defined by the following sine wave form

$$F \sin \omega (T - T_0)$$

The amplitude of the load is given by value 1, 2, 3 or 4 in the load attribute record.

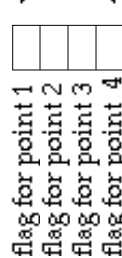


## &lt;Contents of load condition values&gt;

Load Type	Value 1	Value 2	Value 3	Value 4
Nodal force	$P$	-	-	-
Mid-point force	$P$	$d/L$ or $L$	-	-
Uniform force	$w$	-	-	-
Trapeziform force	$w_1$	$w_2$	-	-
Parabolic force	$w_1$	$w_2$	$w_3$	-
Bilinear force	$w_1$	$w_2$	$w_3$	$w_4$
Nodal Moment	$M$	-	-	-
Mid-point moment	$M$	$d/L$ or $L$	-	-
Uniform moment	$M$	-	-	-
Body force	$a$	-	-	-
Thermal load	$T_{av}$	$T_{gr}$	-	-
Self-straining force	$P$	-	-	-
Hydrostatic in X,Y,Z	$p_1$	$p_2$	$h_1$	$h_2$

For bilinear force, the coordinates of the 4 corner points defining the distribution of the load should be supplied in addition as follows.

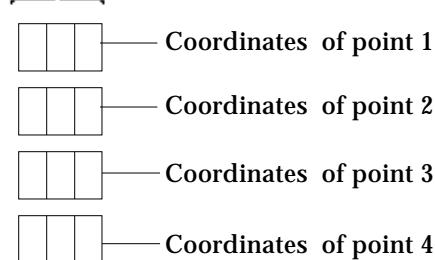
4×2 byte(short int)



Flags indicating whether the coordinates of each of the 4 corner points are defined by data input.

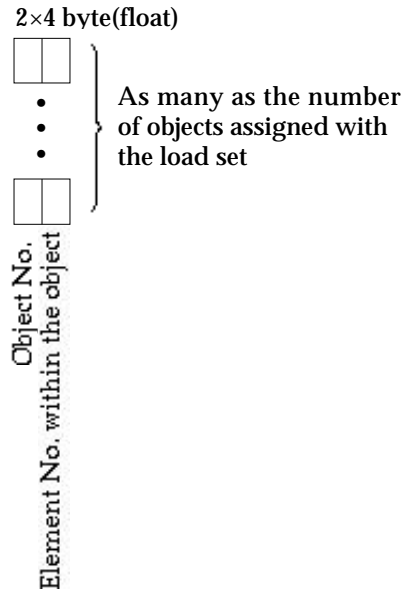
=0 : the coordinates of the point are not yet defined  
=1 : the coordinates are defined.

3×4 byte(float)



<Coordinates of 4 corner points of bilinear force>

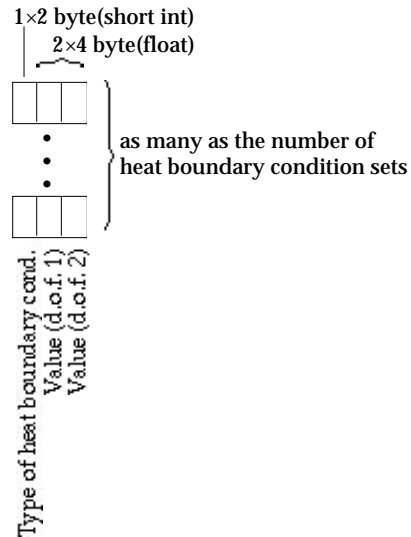
### ■ Load condition data - assigned objects



- Object No.: The No. of the object to which the load condition is assigned. Or, if the object type is “Element”, this entry has the No. of the object to which the element belongs.
- Element No. within the object: The No. of the element within the object. This entry is used, only in case the Object type is element.

### ■ Heat boundary condition data

The heat boundary condition data are maintained chiefly for graphical user interface, and involved indirectly, through equivalent nodal heat and convection boundary conditions, in finite element analysis. They are also the source from which the equivalent nodal heats and convection boundary conditions are computed. The items of heat boundary conditions vary depending on their type as described below.



## &lt;Heat boundary condition data&gt;

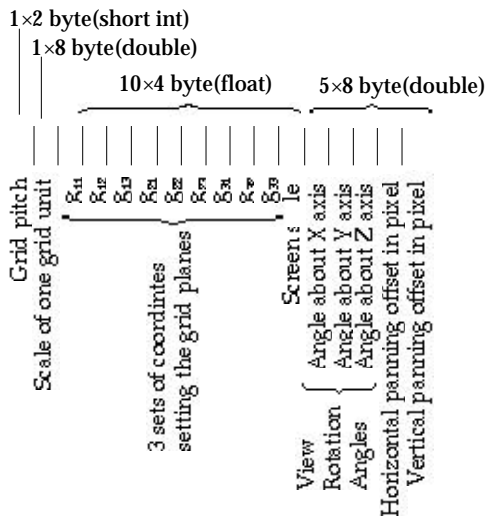
- Heat boundary condition type: The code representing the type of the heat boundary condition.
  - 0: Temperature
  - 1: Convection
  - 2: Heat flux
  - 3: Point source
  - 4: Element source
  - 5: Region source
- Value 1 - 4: These entries are data entered using the editable text items of “Heat Boundary” dialog. Their contents vary depending on the type of the heat boundary condition as shown below. Also refer to Chapter 5.

## &lt;Contents of load condition values&gt;

Type	Value 1	Value 2
Temperature	temperature	-
Convection	convection coeff.	ambient temperature
Heat flux	flux	ambient temperature
Point source	heat quantity	-
Element source	heat quantity	-
Region source	heat quantity	-

### ■ View transformation data

View transform data consists of information on grid setting and view transformation in interactive work environment, the state of which at the time of file saving is retrieved and applied as the initial setting when the file opened. The view transformation data may be saved in an independent view files, if necessary. The record contains the same contents both for a VisualFEA file and for an independent view file.

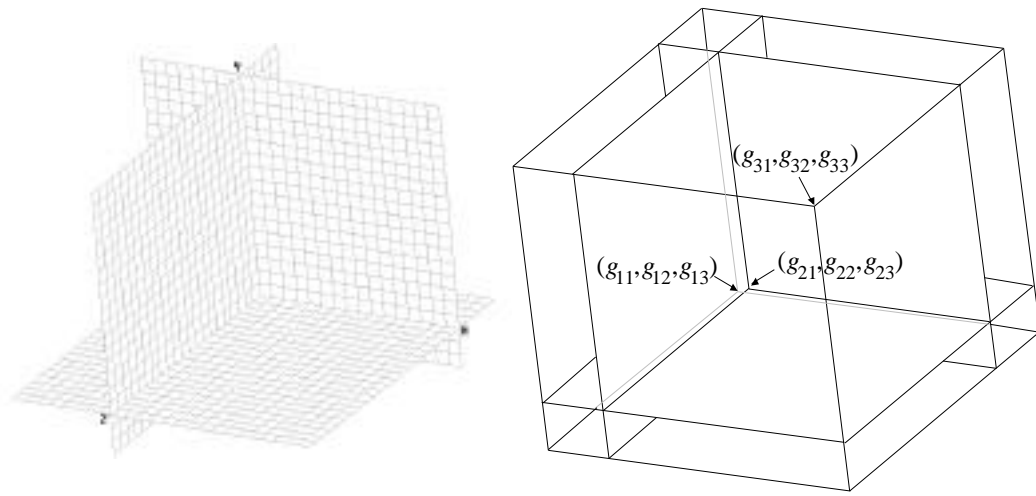


#### < View transformation data>

- **Grid pitch** : Number of pixels between grid lines on the screen with initial view transformation. This is the value entered “Grid Setting” dialog, or “Preference” dialog as described in Chapter 2.
- **Grid scale** : The actual distance represented by the interval between two adjacent grid lines. This value is also entered using “Grid Setting” dialog, or “Preference” dialog.
- **Grid box coordinates** : 3 sets of XYZ coordinates setting the planes as shown in the figure below. These values are inputted by interactively moving or resizing the grid planes as explained in Chapter 2.
- **Screen scale** : Zoom factor inputted by interactive view scaling action as explained in Chapter 2. The view scale, which is the ratio between the screen coordinates in pixels and the actual coordinates, are determined by

$$\text{view scale} = \text{grid pitch} \times \text{grid scale} \times \text{screen scale}$$

- **View rotation angles** : Angles of view rotation about X, Y and Z axes respectively. These angles are used in computing the view rotation matrix **R** by the following equation.



&lt; Grid box coordinates&gt;

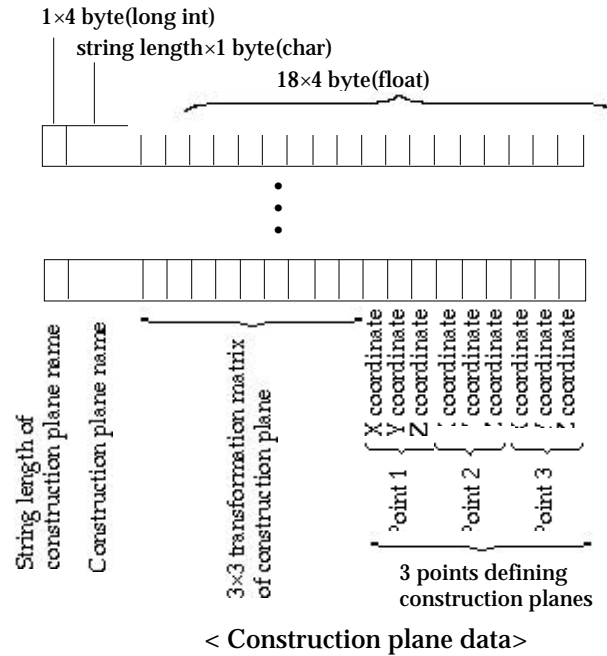
$$\mathbf{R} = \begin{matrix} \cos\theta_z & \sin\theta_z & 0 & \cos\theta_y & 0 & -\sin\theta_y & 1 & 0 & 0 \\ -\sin\theta_z & \cos\theta_z & 0 & 0 & 1 & 0 & 0 & \cos\theta_x & \sin\theta_x \\ 0 & 0 & 1 & \sin\theta_y & 0 & \cos\theta_y & 0 & -\sin\theta_x & \cos\theta_x \end{matrix}$$

- Horizontal panning offset : Horizontal offset distance of screen view in pixels. This value is changed by panning the screen view in horizontal direction as described in Chapter 2.
- Vertical panning offset : Vertical offset distance of screen view in pixels. This value is changed by panning the screen view in vertical direction as described in Chapter 2.

### ■ Construction plane data

Construction plane is a user defined grid plane with grid points for coordinate input. They are created interactively as described in Chapter 2. A plane in 3 dimensional space is uniquely defined by 3 points on the plane. The planes are not usually parallel to X, Y or Z axis. Their orientation is saved in a 3×3 transformation matrix. Each construction plane has its own name which is saved as a character string. The length of the name is variable.

The construction plane data have as many records as the number of construction planes which is contained in the header record. Refer to Chapter 2 for creation and use of user defined grid plane.



- String length of construction plane name : The number of characters used for the construction plane name.
- Construction plane name: Character string of the construction plane name. The name is endowed at the time of creating the construction plane. And this name is used to retrieve the construction plane.
- Transformation matrix of construction plane: 3×3 matrix setting the orientation of the construction plane.
- Coordinates of 3 points defining the construction plane: X, Y and Z coordinates of 3 points which were inputted in order to define the plane.

### ■ Symbol size record

A force of a load condition is symbolized by an arrow. The length of the arrow is drawn approximately in proportion to the length of the force magnitude. The proportion can be set differently for various load types. It is also possible to set the reference magnitude of the force for unit length of the arrow differently for various load types. The force types are classified differently from the actual types used in load condition data. They are point load, curve force, surface, volume force, point moment, curve moment, self straining force and thermal force.

The line length of the arrow in pixels is computed as follows:

$$\text{arrow length} = \frac{\text{force magnitude}}{\text{reference magnitude}} \times \text{unit length} \times \text{symbol scale}$$

- Symbol scale : One symbol scale for each one of the load types,
- Reference magnitude: One reference magnitude for each one of the load

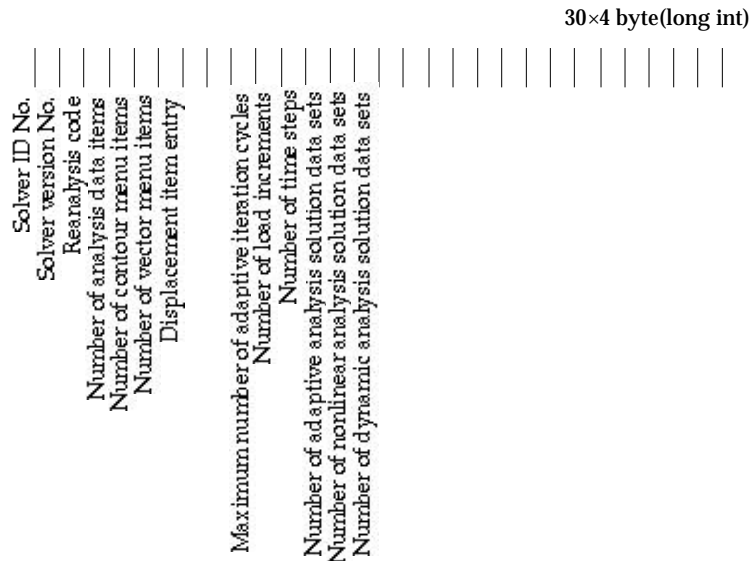


## Analysis data

The analysis data are generated as results of finite element processing either by VisualFEA or by an external solver. In order to visualize the analysis data by VisualFEA, they should be appended to the end of the modeling data in the same VisualFEA file.

### ■ Analysis data master record

Analysis data master record consists of the solver's identification and basic information on the analysis items. The record has 30 entities of 4 byte long integer. The blank spaces are reserved for future use.



<Analysis data master record>

- **Solver ID No.:** A number given to the solver in order to identify the external solver. In order to avoid conflict due to assigning the identical number for different solvers, it is advisable to use a large number within the range of 4 byte long integer.
- **Solver version No.:** Solver's version No if any. This value should be an integer number. This entry may be used to overcome incompatibility between different versions of the solver.
- **Reanalysis code:** This entry indicates whether the matrix file necessary for reanalysis is retained or not. The matrix file may be either saved for reanalysis or removed at the end of the finite element processing, depending on the setting of the solver options.
  - 0: The matrix file was removed. Reanalysis mode is not allowed.
  - 1: The matrix file was save. Reanalysis mode is allowed.
- **Number of analysis data items:** The number of data items obtained from the



finite element processing, and written in the VisualFEA file.

- **Number of contour menu items:** The number of popup menu items in the “Contour” dialog for contouring of the analysis data.

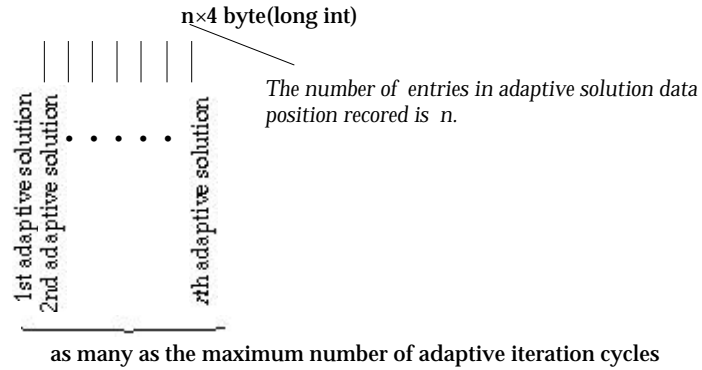
*The contour menu items may be customized by the external solver. It is the responsibility of the solver to count the custom popup menu items and write the number into this entry.*

- **Number of vector menu items:** The number of popup menu items in the “Vector” dialog for contouring of the analysis data.
- **Displacement item entry:** This entry directs the displacement data position entry in the analysis data position record.
- **Maximum number of adaptive iteration cycles:** The maximum number of iteration cycles is set as a criterion of terminating the iteration, prior to adaptive processing. This value is used to secure the space of the adaptive analysis data position record, and is ignored for non-adaptive analysis.
- **Number of load increments:** The value of this entry is used to secure the space of the nonlinear analysis data position record, and is ignored for linear analysis.
- **Number of time steps:** The value of this entry is used to secure the space of the dynamic analysis data position record, and is ignored for static analysis.
- **Number of adaptive analysis solution data sets:** The number of data sets including the intermediate and the final data sets obtained from adaptive analysis process.
  - 0: for non-adaptive analysis.
  - 2: for adaptive analysis in which only the data sets of the initial and the final stages are stored.
  - 3 or greater: for the data sets of the intermediate as well as the initial and the final stages.
- **Number of nonlinear analysis solution data sets:** The number of data sets stored in the intermediate and the final steps of load increments for nonlinear analysis.
  - 0: for linear analysis.
  - 1: for nonlinear analysis in which only the final solution is stored.
  - 2 or greater: for nonlinear analysis in which the intermediate data as well as the final solution data are stored.
- **Number of dynamic analysis solution data sets:** The number of data sets stored for the intermediate and the final time steps of dynamic analysis. This entry is ignored for static analysis.

#### ■ Adaptive analysis data position record

Adaptive analysis data position record has the information on the offset distance, in bytes, from the beginning of the file to the starting point of each data sets stored

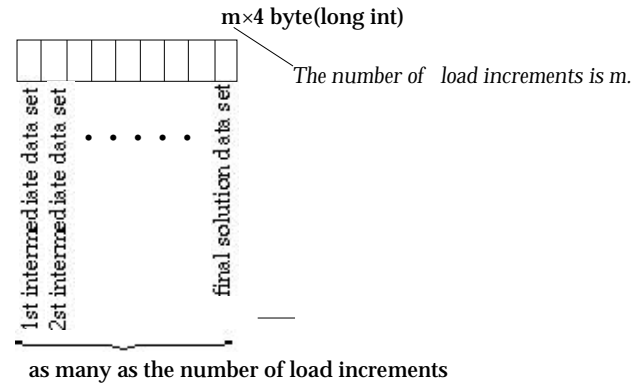
for the intermediate process of an adaptive analysis. The record has entities of 4 byte long integers, as many as the maximum number of adaptive iterations. This record is included only in case of adaptive analysis.



<Adaptive analysis data position record>

#### ■ Nonlinear analysis data position record

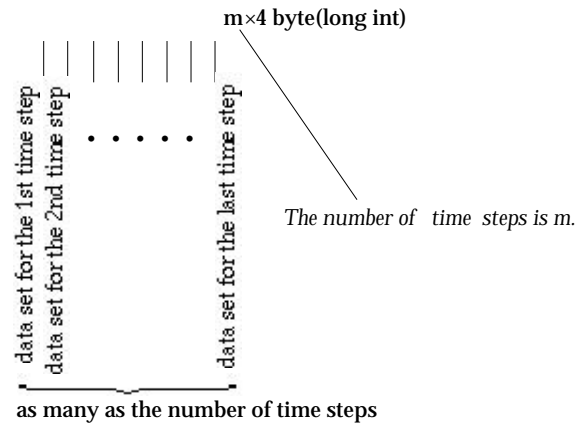
Nonlinear analysis data position record has information on the offset distance, in bytes, from the beginning of the file to the starting point of each data sets stored for the intermediate process of a nonlinear analysis. The record has entities of 4 byte long integer as many as the number of load increments. This record is included only in case of nonlinear analysis.



<Nonlinear analysis data position record>

#### ■ Dynamic analysis data position record

Dynamic analysis data position record has the information on the offset distance, in byte, from the beginning of the file to the starting point of each data sets stored for whole time steps of a dynamic analysis. The record has entities of 4 byte long integer as many as the number of time steps. This record is included only in case of dynamic analysis.

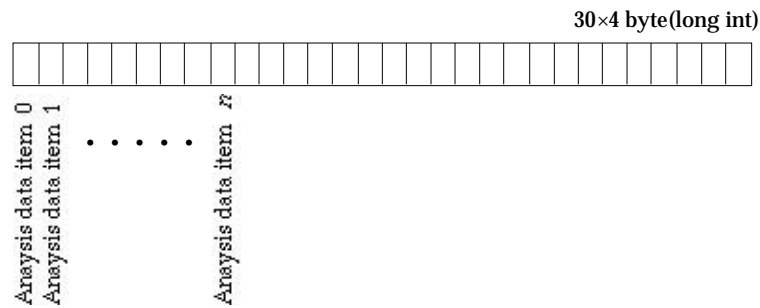


<Dynamic analysis data position record>

### ■ Analysis data position record

Analysis data position record has information on the offset distance, in bytes, from the beginning of the file to the starting point of each analysis data item. The record has 30 entities of 4 byte long integer. The blank spaces are reserved for future use.

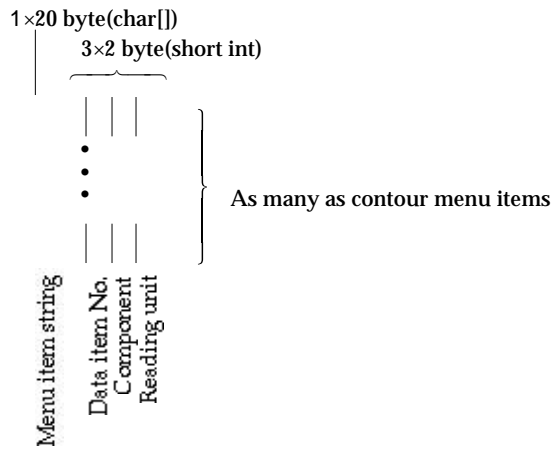
- Analysis data item 1 through  $n$  : The positions of analysis data items.



<Analysis data position record>

### ■ Custom contour menu data

The popup menu items in "Contour" dialog can be customized by the external solver. For this purpose, the solver should write the following menu data in front of the analysis data in the VisualFEA file.



<Custom contour or vector menu data>

- Menu item string: Menu string with 20 characters including blank space. The string appears as an popup menu item.
- Data item No.: The data with this item No. will be displayed when the menu item is chosen.
- Component: The component of the data item to be displayed.
- Entry No. or type: One component data consist of many data unit. Entry No. designates which entry out of a data unit is to be read and displayed. Entry type designates how a data unit is read and displayed. If this entry has a small integer value like 0,1,2,3,..., then this represent the entry No. Otherwise, this is relevant to entry type as follows ( The macro in the header file is shown within the parenthesis.):

6000 (DISPLACEMENT\_NORM) : Read all entries of each data unit, and make one norm of these values. For example,

$$\ddot{u} = \sqrt{u_0^2 + u_1^2 + u_2^2}$$

7000 (READ\_ALL) : Read all entries of each data unit.

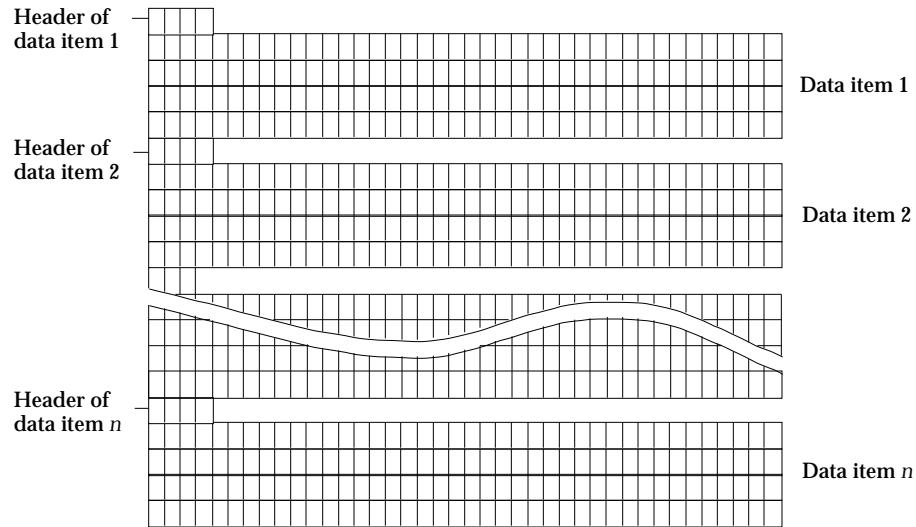
#### ■ Custom vector menu data

The popup menu items in “Vector” dialog can be customized by the external solver in the same way as the case of “Contour” dialog.

#### ■ Analysis data items

The number of analysis data items are specified in the analysis data master record described above. Each data item is obtained as the final products of finite element processing, either by VisualFEA or an external solver. The analysis data have hierarchical construction of entry, data unit, component and item. Each data item

consists of header and main body as shown in the following figure.



< Structure of analysis data >

The header record has the information on the hierarchical structure of the data. A data item is composed of a few components. A component has a number of data units. A data unit has a uniform or a variable number of consecutive data entries.

- **Number of components:** One data item has either only one component or more than one components. This entry specifies the number of components composing the data item.
- **Number of data units:** Data unit is an aggregate of consecutive data entries which are read as one unit. There are number of data units in one component of a data item. This may be , for example, the total number of elements or the number of nodes, used in the model.
- **Unit size:** A data unit may have only one entry or a number of entries. the unit size may be uniform for all data units in the component, or may vary from unit to unit. For example, if the data item is the principal directions, the unit sizes are uniformly the same as the spatial dimension. If the data item is the displacements in a mixed structure, the unit sizes may not be uniform, because the number of nodal d.o.f. are not necessarily the same for all nodes. The following macro is defined for variable nodal d.o.f. in the header file.

2000 (VARIABLE\_NODAL\_DOF) : A data unit consists of data entries as many as the nodal d.o.f. which varies from node to node.

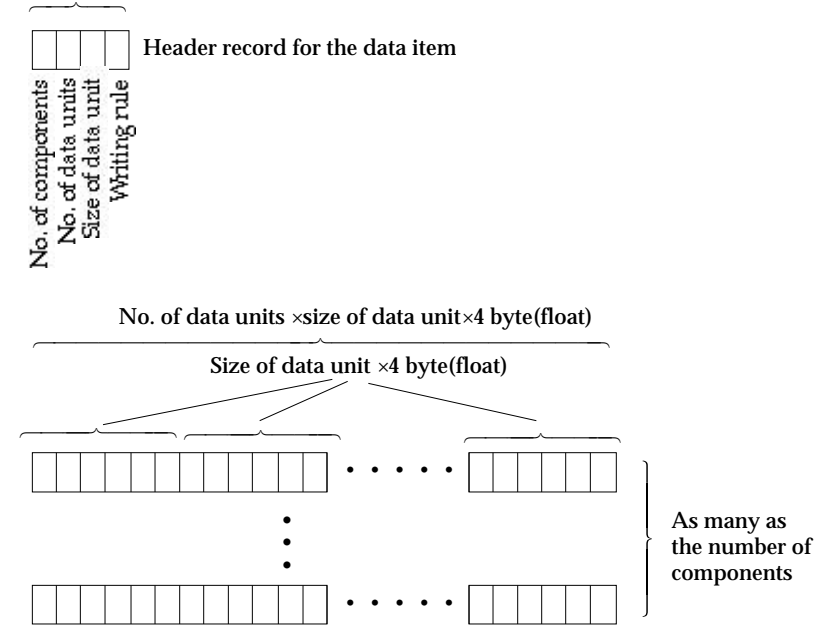
- **Writing rule:** This entry specifies what a data unit is represented by. The following macros are defined in the header file.

1 (WRITE\_NODE) : A data unit matches with one node. So, there are as many data units as the number of nodes.

2 (WRITE\_ELEM) : A data unit matches with one element. So, there are as many data units as the number of elements.

3 (WRITE\_ELEM\_NODE) : A data unit matches with each one of the nodes in an element. So, there are as many data units as the sum of the numbers of nodes in all elements.

4×4 byte(long int)



< Structure of an analysis data item >

# Index

- 2 edge mesh 4-12
  - compatible for mesh generation 4-14
  - generating mesh 4-12
  - shifting alignment 4-16
- 2 Edges 4-12
- 2-D frame 1-12, 4-63
- 2-D seepage 1-12
- 2-D seepage analysis 7-73
- 2-D truss 1-11, 4-63
- 3 edge mesh 4-17
  - compatible for mesh generation 4-17
  - generating 4-17
- 3 Edges 4-17
- 3-D cursor 2-29
- 3-D frame 1-12, 1-4-63
- 3-D solid 1-11
- 3-D truss 1-11, 4-63
- 3-D volume seepage 1-12
- 3D volume heat 7-1
- 3D volume seepage 7-1
- 4 edge mesh 4-18
  - compatible for mesh generations 4-19
  - generating 4-18
- 4 Edges 4-18

## A

- Absorb Node 4-72
- acceleration parameter 6-9
- acceleration participation 7-64
- adaptive analysis 1-12
- Adaptive analysis 1-16, 6-5
- adaptive process 6-6
  - iteration cycle 6-5
  - termination 6-5
  - visualization of 6-6
- aerial view 2-49

- analysis class 5-8
- analysis data 9-2
  - boundary condition data 9-5
  - element property data 9-5
  - member joint data 9-5
  - node data 9-5
  - temperature data 9-5
- Analysis Info 6-26
- analysis information 6-26
- analysis results 9-2
- analysis subject 2-14, 8-9
- analysis type 7-4, 8-1, 8-4
- animation 1-13, 7-83, 7-86
  - creating 7-86
  - playing 7-90
- animation script 7-86
  - commands 7-87
  - text file 7-86
  - writing 7-86
- animation speed 7-67
- arrow display 7-45
  - displaying selectively sampled 7-50
  - length of 7-50
  - line arrow in 3-D volume 7-47
  - line arrow on surface 7-47
  - options for 7-53
  - placement focus of 7-49
  - position of heads 7-49
  - solid arrow in 3-D volume 7-47
  - solid arrow on surface 7-47
  - thickness of 7-50
  - type of 7-47
- Arrow on Object 5-59
- attribute assignment 2-48
  - displaying 2-48
- attributes 5-8
- Auto Mesh 4-4

Auto Mesh (Volume) 4-39  
 Auto Mesh on Primitive 4-9  
 Auto Solve 6-20, 8-13  
 automatic tetrahedronization 4-38, 4-39  
     boundary meshes for 4-41  
     generating mesh 4-39  
 automatic triangulation 4-4  
 auxiliary plane 7-3, 7-16  
 axial force 8-2, 8-4, 8-5  
 axis of revolution 4-21, 4-30, 4-31, 4-58, 4-59, 4-67  
 axisymmetric 1-11  
 Axisymmetric heat 7-1

## B

B-spline surface 3-1, 3-13  
 back substitution 1-15  
 band width 6-22  
 base model 5-71  
 bending moment 8-4, 8-9  
 Bezier surface 3-1, 3-14  
 boundary condition 5-30  
     heat conduction 5-38  
     structural 5-30  
 boundary condition 9-14  
 boundary condition data 9-14  
 boundary curve 3-1  
 boundary lines 7-9  
 boundary surface 7-35  
 boundary surface rendering 7-8, 7-48  
     none 7-8, 7-35, 7-48  
     opaque 7-8, 7-35, 7-48  
     outline 7-8, 7-35, 7-48  
     style of 7-8  
     transparent 7-8, 7-35, 7-48  
     wireframe 7-8, 7-35, 7-48  
 bounding box 1-7, 2-32  
 bounding curve 4-23, 4-25  
 bounding primitive 4-52  
 bounding surface 3-1  
 box edge 4-43

box edge mesh 4-42  
     boundary surfaces for 4-41  
     curves forming 4-42  
 Box Edges 4-42  
 broken mesh 2-50  
 button 2-5  
     active 2-5  
     inactive 2-5  
     pressed 2-5  
 By Coordinates 7-22  
 By Ratio 7-22

## C

CAD software 9-5  
 capping 1-13  
 centering 1-7  
 central difference method 6-7, 6-9  
 Change Side 5-58  
 circle 1-9, 3-1, 3-3  
     center & radius 3-6  
     clockwise 3-6  
     counter-clockwise 3-6  
     three point 3-6  
 circular arc 3-1, 3-3  
     center 3-3  
     center & angle 3-5  
     clockwise 3-4  
     counter-clockwise 3-4  
     end point 3-4  
     starting point 3-4  
     three point 3-5  
 Clear 3-19  
 Clear Diagram 8-10  
 combined Load Case 7-69  
 cone 1-9, 3-1  
 confined head 5-41, 5-42  
 constitutive model 5-9  
     compression only 5-9  
     easto-plastic:Tresca 5-9  
     elasto-plastic:D-P 5-9



- elasto-plastic:M-C 5-9
- elasto-plastic:V-M 5-9
- linear 5-9
- tension only 5-9
- contour 1-13
  - marking 1-14
- contour band 7-3
  - aligning 7-12
  - number of 7-6
  - range of 7-6
- contour image 7-2
- contour marking 7-28
- contour scale 7-10, 7-11, 7-38
  - editing 7-10
  - format of 7-11
  - options 7-14
  - reading 7-15
  - saving 7-15
  - setting 7-10
  - symmetrically arranged 7-13
  - truncating 7-12
- Contour... 7-4
- contouring 7-3, 9-1
- contouring method 7-7
  - pixel painting 7-7
  - polygon fill 7-7
- contouring object 7-5
  - all abjects 7-6
  - cross planes 7-6, 7-9, 7-25
  - cut plane 7-6, 7-16
  - parallel planes 7-6, 7-20
  - selected objects 7-6
- control point 2-21, 2-54, 2-57, 3-1, 3-2, 3-19, 3-20
  - displaying 2-48
  - end point 3-2, 3-3
  - moving 3-20, 3-21
  - single point 3-11
- convection 5-38
- convergence 6-10
- convergence criterion 6-10
- coordinate axis 2-15, 2-22
- coordinates 2-54, 2-57
  - 3-dimensional 2-54, 2-55
  - inputting 2-54, 2-60
  - inputting by keyboard 2-60
  - of points 2-54
  - undoing input 2-61
- Copy 4-69
- crack tip 4-78
- cross plane 1-13, 7-25
  - activating 7-25
  - contouring on 7-27
  - locating 7-27
  - placing by mouse dragging 7-26
  - setting 7-25, 7-26
  - setting mode 7-25
- cross-platform software 2-1
- cursor 1-8, 2-1, 2-21
  - 3-dimensional 1-8, 2-1, 2-29, 2-54, 2-59
- curve 1-9, 3-1, 4-21
  - B-spline 1-9, 3-1, 3-9, 3-10
  - Bezier 1-9, 3-1, 3-9, 3-10
  - building boundary 3-31
  - circle 3-1
  - circular arc 3-1
  - copying 3-19
  - creating 3-2
  - cubic spline 1-9, 3-1, 3-9
  - cutting 3-19
  - deleting 3-19
  - divided 3-36
  - dividing 3-32, 3-33
  - dividing as a whole 3-34
  - division 3-1
  - duplicating 3-22
  - ellipse 3-1
  - elliptic arc 3-1
  - filleting 3-28
  - handling 3-19
  - linked 3-27
  - linking 3-27
  - moving 3-20

- mirroring 3-25
- parabola 3-9
- parametric 3-9
- pasting 3-19
- polyline 3-1, 3-10
- polynomial 1-9, 3-1, 3-10
- projecting 3-30
- rectangle 3-1, 3-11
- reshaping 3-20
- seed 4-21
- segmented 1-9
- separating 3-28
- splitting 3-29
- straight line 3-1
- type 3-2
- curve division 3-1, 3-32
- Curve Normal Mode 8-13
- Curve Plot 7-39, 7-40
- curve plotting 7-2, 7-39
  - initiating 7-39
  - modifying 7-40
  - options for 7-41
  - resizing 7-41
  - sampling line 7-41
  - terminating 7-41
- Custom Plane... 7-17
- custom view 2-40
  - getting 2-40
  - setting 2-40
- Cut 4-69
- Cut Object 7-18
- cut plane 7-16
  - activating 7-16
  - contouring on 7-17
  - setting 7-16
  - setting mode 7-16
  - slitting objects 7-18
- cylinder 1-9, 3-1
  - creating 3-17

## D

- damping 6-8
  - coefficient 6-8
  - mass damping ratio 6-8
  - modal 6-9
  - mode equivalent Rayleigh 6-8
  - Rayleigh 6-8
  - stiffness damping ratio 6-8
- data 5-1
- data assignment 1-10, 5-1
  - adding 5-35
  - boundary condition 1-10
  - checking 5-1, 5-7
  - clearing 5-7
  - creating 5-1
  - defining 5-3, 5-11
  - deleting 5-1
  - dynamic motion 1-10
  - element properties 5-8
  - element property 1-10
  - ending 5-7
  - entering data items 5-5
  - general procedure 5-3
  - heat boundary condition 1-10
  - load condition 1-10
  - Member joint 1-10
  - modifying items 5-5
  - replacing 5-35, 5-40
  - visualizing 5-1
- data edit text 2-5
- data for user interface 9-5
  - construction plane data 9-5, 9-41
  - curve data 9-5, 9-26
  - curve end point data 9-26
  - heat boundary condition data 9-5
  - load condition data 9-5, 9-33
  - primitive surface data 9-5, 9-29
  - surface mesh 9-5, 9-30
  - symbol size record 9-42
  - symbol variables 9-5

- view transformation data 9-5, 9-40
- volume mesh 9-5, 9-32
- data interface 9-1
  - with external softwares 9-1
- data item 7-4, 7-46
- data items 9-3
  - number of 9-3
- data level surface 1-13
- data offset distance 9-7
- Data sampling 1-14
- data set 5-1
  - assigning 5-1, 5-6
  - condensing 5-7
  - unassigning 5-1
- decomposition 1-15
- deformed shape 1-13, 7-2, 7-52, 8-7
  - of rigid frame 8-7
  - scale 7-53, 7-67
  - style of rendering 7-53
  - visualizing displacements by 7-52
- Deformed Shape... 7-53
- degree of freedom 5-30
- diagram 1-13, 8-2
  - 2-D and 3-D rigid frames 8-4
  - 2-D and 3-D trusses 8-2
  - axial force 8-2
  - bending moment 8-6
  - clearing 8-10
  - direction of 8-9
  - displaying more than one 8-9
  - displaying part of 8-15
  - for selected members only 8-15
  - redrawing 8-10
  - reversing directions 8-10, 8-11
  - scale of 8-9
  - shear force 8-6
  - torsional moment 8-8
  - turning on/off text 8-14
  - updating 8-9
- diagram values 8-14
  - selectively turning on text of 8-14
  - turning off text of 8-14
- dialog 2-2, 2-6, 3-17, 4-4, 5-8, 6-3, 7-1
  - 2 Edge Surf 4-12
  - 3 Edge Surf 4-17
  - 4 Edge Surf 4-18
  - Analysis Options 2-20, 6-3, 7-4
  - Arrow Display 7-45
  - Auto Tri 4-4, 4-5
  - AutoMesh Volume 4-39
  - Box Edges 4-42
  - Contour Display 7-4, 7-8, 7-28, 7-33
  - Contour Scale 7-10
  - Cut Plane Setting 7-17
  - Cylinder 3-17
  - Data Info 7-30
  - Deformed Shape Display 7-53
  - Dynamic Analysis Options 6-7
  - Dynamic Motion 5-63, 5-64
  - Extr Surf 4-21
  - Extr Vol 4-50
  - file opening 2-11
  - Iso-surface Display 7-33, 7-37
  - Load Condition 5-54, 5-55
  - Load Incremental Steps 6-12
  - Load Time Series 5-55, 5-56
  - Member Joint 5-68
  - modal 2-8
  - Modal Damping Coefficient 6-9
  - Mode Shape Control 7-67
  - modeless 2-8
  - Node Info 4-70, 4-71
  - Nonlinear Analysis Options 6-10
  - Other Divisions 3-34, 3-35, 3-37
  - Parallel Plane Setting 7-22
  - Preference 2-15
  - Prism Edges 4-45
  - Project Setup 6-7, 6-10
  - Property 5-8, 5-10
  - Revolve Surf 4-30
  - Revolve Vol 4-58
  - Save As 2-42

- Struct Boundary 5-30, 5-37
  - Surface Plotting 7-43
  - Tetra Edgess 4-47
  - Translate Surf 4-27
  - Twist Surf 4-33
  - Twist Vol 4-60
  - displacement 7-1, 8-2, 8-4, 8-7
  - display options 8-12
  - Divide 3-34
  - Divide As a Whole 3-35
  - division 3-36
    - density 3-36
    - number of 3-36
    - removing 3-36
    - setting 3-36
  - division density 3-33, 4-22, 4-24, 4-28, 4-34, 4-51, 4-53, 4-55, 4-57, 4-59, 4-61
    - weight of 3-38
  - drawing mode 8-1, 8-9
    - curve normal mode 8-13
    - instant 8-10
    - Instant redrawing mode 8-12
  - duplication 1-8, 1-11
  - dynamic analysis 1-12, 5-54, 6-7
  - Dynamic Display 7-41
  - dynamic mode 6-8
    - number of 6-8
  - dynamic mode shape 7-67
  - dynamic motion 5-63, 5-63
    - animation of 7-65
    - assigning 5-63
    - defining 5-63
    - harmonic 5-64
    - time dependency 5-64
    - transient 5-64
- E**
- editable text 2-60, 5-39, 5-45, 5-50
  - educational aid 1-14
  - educational aids 1-7
  - elasto-plastic 5-9
  - element 5-9
    - 3-d solid 5-9
    - axisymmetric 5-9
    - embedded bar 5-9, 5-23
    - frame 5-9
    - gap 5-9, 5-10
    - heat conduction 5-9, 5-38
    - interface 5-9
    - plane strain 5-9
    - plane stress 5-9
    - plate bending 5-9
    - shell 5-9
    - truss 5-9
  - element properties 5-8
    - defining 5-8
  - element data 9-11
  - Element Joint 5-68
  - element number 6-5
  - element numbering 6-1
    - optimizing 6-5, 6-23
  - element property 5-15
  - element property data 9-15
  - element type 6-1, 9-12
  - ellipse 1-9, 3-1
  - ellipses 3-7
    - full 3-8
    - half 3-8
    - quarter 3-7
  - elliptic arc 3-1
  - embedded bar 1-12
  - embedded bar element 5-9
    - linear bonding 5-10
    - nonlinear bonding 5-10
  - energy norm error 6-6
  - equation solver 6-4
    - frontal 6-4, 6-23
    - skyline 6-4, 6-22
  - equivalent nodal heat data 9-20
  - equivalent node force data 9-19
  - error tolerance 6-10

explicit method 6-7  
 Export view 2-42  
 external solver 9-2  
 Extrude to Curve 4-24  
 Extrude(Surface) 4-21  
 Extrude(Volume) 4-50  
 extrusion 4-21, 4-49  
 extrusion height 4-34

## F

file 1-7, 2-11  
   closing 1-7, 2-12  
   opening 1-7, 2-11  
   saving 1-7, 2-13  
 file contents 9-2  
 file format 9-1  
 File I.D. 9-7  
 file identifications 9-6  
 file information 9-6  
 file opening 9-1  
 file opening dialog 7-83  
 file position 9-3  
   analysis 9-3  
   modeling 9-3  
   record 9-3  
 file saving dialog 7-85  
 file structure 9-1  
 fillet 3-28  
 finite element analysis 1-1, 6-1  
   learning 1-1  
   stages of 1-2  
   teaching 1-1  
   using intuitiveFEM 1-3  
 finite element processing 6-1  
 finite element solution 6-2  
 fit-to-window 1-7  
 fixity 5-31  
 flow discharge 7-78  
 flow path 7-76  
 force symbol 5-59

  arrows off the object 5-59  
   arrows on the object 5-59  
   placement of 5-59  
   size of 5-60  
 frame analysis 8-1  
   procedure of 8-1  
   visualizing analysis results 8-2  
 frame element 4-63  
 frame elements 4-63  
   2 Edges 4-66  
   3 Edges 4-66  
   4 Edges 4-66  
   duplicating 4-67  
   Extrude (Surface) 4-66  
   Extrude to Curve 4-66  
   generating 4-63  
   Revolve (Surface) 4-66  
   Sweep (Surface) 4-66  
   Twist (Surface) 4-66  
 frame structure 8-1  
   2-D frame 8-1  
   2-D truss 8-1  
   3-D frame 8-1  
   3-D truss 8-1  
   geometry of 8-1  
 frequency 6-9  
 frontal length 6-23

## G

gap element 5-10  
 Gauss quadrature 6-25  
 geometric data 1-8  
   inputting 1-8  
 geometric model 3-1  
 geometric nonlinear 6-10  
 graph 1-13, 7-41  
 graphical modeling softwares 9-5  
 graphical visualization 7-1  
 grid 1-8, 2-1, 2-21  
   3-dimensional 1-8, 2-22

- initial state 2-15
- moving 1-8, 2-23
- on/off 1-8
- pane 2-15, 2-54
- pitch 2-15, 2-22
- plane 1-8, 1-9, 2-22
- preference 2-15
- resizing 1-8, 2-24
- scale 2-15, 2-22
- settings 2-15, 2-22
- sub- 1-8
- subdividing 2-25
- subdivision 2-22
- turning on and off 2-22
- user defined 2-26

grid plane 2-54

## H

- harmonic load 5-54
- harmonic motion 5-64
- header 9-7
- header record 9-8
- heat conduction 5-9
- heat conduction 1-12
  - 2-D plane 1-12
  - 3-D volume 1-12
  - Axisymmetric 1-12
- heat conduction boundary condition 5-38, 5-39
  - assigning 5-39
  - defining 5-38
  - representation 5-40
  - types of 5-38
- heat flow directions 7-1
- heat flux 5-38, 7-1
- heat transfer analysis 1-5, 7-1
- hidden line removal 2-50
- Hide Scale 7-31
- hiding 2-43
  - line element 2-45
  - selected meshes 2-43

- unmeshed curve 2-44
- unselected meshes 2-43

## I

- icon 2-9
- Image 1-13, 7-83
  - Capturing 1-13
  - handling 1-13, 7-83
  - Overlaying 1-13
  - Reading 1-13
  - retrieving 7-83
  - saved in a file 7-83
  - Saving 1-13, 7-85
- Import View 2-42
- incremental method 6-10
- incremental step 6-12
- Initial 2-41
- initial displacement 5-31
- initial view 2-41
  - getting 2-41
- initial water table 5-41, 5-44
- input data 9-3
  - element data 9-3, 9-11
  - element properties 9-3, 9-15
  - equivalent nodal forces 9-3, 9-19
  - equivalent nodal heats 9-3, 9-20
  - for external solver 9-3
  - heat boundary condition 9-38
  - integration scheme record 9-22
  - nodal dynamics data 9-21
  - node data 9-3, 9-10
  - number of integration points 9-3
  - state of frame member joints 9-3, 9-18
  - structural boundary conditions 9-3
  - temperature distribution 9-3, 9-17
- Instant Redrawing 6-20, 8-13
- integration method 6-7
- integration scheme 6-1
- integration scheme record 9-22
- interface element 1-12, 5-9

- gap 5-10
- linear 5-9
- no compression slip 5-10
- no tension slip 5-9
- intermediate model 6-6
- interpolation 1-15
- intersection 1-8, 3-29
  - curve 3-29
  - Curve-curve 1-8
  - obtaining 3-30
  - point 3-29
  - surface primitive 3-30
  - surface-surface 1-8
- intuitiveFEM version No. 9-7
- iso-surface 1-13, 7-32
  - boundary surface rendering 7-35
  - contours on 7-34
  - data items 7-33
  - display option 7-33
  - level 7-37
  - number of 7-35
  - number of contour bands on 7-35
  - setting 7-33
  - visualizing scalar data by 7-32
- iso-surface image 7-2
- Iso-surface... 7-33
- isotropy 5-10
- iteration cycles 6-5, 6-10
- iterative procedure 6-10

## L

- Lagrangian Surface 3-14
- launching intuitiveFEM 2-9
- level of smoothing 4-5, 4-40
- light source 1-7, 2-18, 2-51
  - direction 2-18, 2-51
  - intensity 2-18, 2-51
  - on/off 2-18
  - settings 2-18
- Limit Arrow Size 5-60

- Line Element 2-45
- linear analysis 5-9
- linear static analysis 6-3
- Link 4-69
- Load Combination 5-61
- load condition 5-45, 5-45
  - assigning 5-57
  - multiple 5-57
  - representation of 5-57
- load condition data 9-38
  - assigned objects 9-38
  - attribute record 9-35
  - header record 9-33
- load direction 5-45, 5-48
  - exchanging the reference end 5-58
  - normal to curve 5-48
  - normal to surface 5-49
  - reversing 5-58
  - tangent to curve 5-48
  - tangent to surface 5-49
  - X direction 5-48
  - Y direction: 5-48
  - Z direction 5-48
- load increment 6-11
- load incremental step 6-10
- load selection 5-59
- load type 5-45
  - bilinearly distributed 5-46
  - body force 5-47
  - hydrostatic in X 5-48
  - hydrostatic in Y 5-48
  - hydrostatic in Z 5-48
  - linearly distributed 5-46
  - mid point force 5-46
  - nodal force 5-46
  - nodal moment 5-46
  - parabolically distributed 5-46
  - self-straining force 5-47
  - thermal load 5-47
  - uniform moment 5-47
  - uniformly distributed 5-46

lofting 4-12

## M

Macintosh 9-1

Make Animation... 7-86

mass matrix 6-8

consistent 6-8

lumped 6-8

master record 9-3

material nonlinear 5-9

matrix assemblage 6-5

Matrix partitioning, 1-15

member force 8-2

member joint condition 5-68

assigning 5-68

defining 5-68

member joint data 9-18

menu 2-2, 2-4, 3-3, 4-1, 5-1, 6-7, 7-1, 8-1

assign 2-4, 5-1

Circle 3-3

Cross 7-26

Cut 7-17, 7-18

Diagram 6-20, 7-1, 8-1, 8-4

divide 2-4, 3-33, 3-34

edit 2-4, 3-19, 4-68

Ellipse 3-7

file 2-4, 2-9

head 2-4

mesh 2-4, 4-1

opening 2-9

Parallel 7-20

Plot 7-39

postpro 2-5, 7-1, 7-4, 7-33

pulldown 2-2

render 2-4

solve 2-4, 6-7

Sphere 3-15

view 2-4

mesh 1-5, 4-68

copying 4-69

cutting 4-69

deleting 4-68

hierarchical construction 1-5

merging 4-69

Moving 1-8, 4-73

pasting 4-69

remeshing 4-77

resizing 4-75

rotating 1-8, 4-74

transforming 4-73

mesh by extrusion 4-21, 4-50

generating 4-21, 4-50

up to bounding curve 4-23

up to bounding surface primitives 4-52

mesh by revolution 4-30, 4-58

mesh by translation 4-27

curves for 4-28

generating 4-27

mesh by twisting 4-33, 4-60

axis of 4-34

generating 4-33, 4-60

seed curve for 4-33

mesh editing 1-10, 4-68

Dragging node 1-10

Remeshing 1-10

mesh generation 1-9, 3-1, 4-1

automatic 1-9

automatic tetrahedronization 1-9

automatic triangulation 1-9

coordinate mapping 1-9

editing 1-10

extrusion 1-9

general procedure 4-1

lofting 1-9

nine edge mapping 1-9

on a surface primitive 4-9

revolution 1-10

six edge mapping 1-9

surface 4-3

sweeping 1-9

transfinite mapping 1-9



- translation 1-9
- triangular mapping 1-9
- triangulation on surface primitives 1-9
- twelve edge mapping 1-9
- twisting 1-10
- undoing 4-68
- Mirror 4-36
- mirroring 1-8, 1-11
- modal frequency 6-9
- modal participation factor 7-64
- Mode Shape 7-67
- mode shape display 7-67
  - animated 7-67
  - controlling 7-67
  - ending 7-68
  - scale of 7-68
  - speed of animation 7-68
  - starting 7-67
- modeling data 9-2, 9-6
  - analysis output item record 9-24
  - dynamic analysis setting record 9-24
  - master record 9-6
  - position record 9-7
  - solver option record 9-23
- modification mode 3-19, 3-20
- Move 3-22
- multi-loading view 7-69
- Multi-step Analysis 7-57
- multiple load case 7-69
- multiple load conditions 5-62

## N

- New... 2-14
- Newmark method 6-8
- nodal coordinates 4-70
  - modifying 4-70
- nodal dashpot 5-66, 5-66
- nodal degrees of freedom 5-30
- nodal dynamic properties 5-66
- nodal dynamics data 9-21

- Nodal Mass 5-66, 5-67
- nodal resultant 7-2
- nodal resultants 1-13
- node 2-48, 4-70
  - absorbing 4-72
  - coordinates 4-70
  - dragging 4-70
  - invisible 2-48
  - number 4-71
  - stick to 2-58
- node data 9-10
- node number 6-5
  - optimizing 6-5, 6-22
- node numbering 6-1, 6-22
- nonlinear analysis 1-12, 6-10
- nonlinearity 5-9
- normal direction 4-79
- Normal to X 7-20
- Normal to Y 7-20
- Normal to Z 7-20
- normalized motion 7-64
- number 2-46
  - changing 2-46
  - displaying 2-46
  - object 2-46
- number optimization 1-11
  - element number 1-11
  - node number 1-11
- numerical data 7-1
- numerical integration 6-25

## O

- object 1-7
  - hiding 1-7
  - showing 1-7
- object operation 1-11
- open head 5-41
- Open Image... 7-83
- operating system 2-1
  - Mac 2-1

- Windows 2-1
- Optimize Element Number 6-23
- Optimize Node Number 6-22
- Other Divisions... 3-34, 3-37
- OtherWeight... 3-37
- outline 2-50
- output data 9-4
  - file positions 9-4
  - for contouring 9-4
  - for vector display 9-4
  - from external solver 9-4
  - master variables 9-4

## P

- Page Setup 2-14
- pan 2-39
  - by option-drag 2-39
  - by scroll bar 2-39
  - Centering the display 2-39
- panning 1-7
- parallel plane 1-13, 7-20
  - activating 7-20
  - contouring on 7-24
  - mouse dragging 7-21
  - number of 7-21
  - orientation of 7-22
  - placing 7-21
  - rule of plane allocation 7-22
  - setting interactively 7-20
  - setting mode 7-20
- Paste 4-69
- periodic motion 7-64
- phreatic surface 7-73
- plane 3-1
- plane strain 1-11
- plane stress 1-11
- plane/surface 5-9
- plate 1-11
- Play Animation... 7-90
- plotted surface 7-43

- the datum of 7-44
- point source 5-38, 5-43
- popping up texts 8-14
- position record 9-3
- Postprocessing 1-2, 7-1, 8-1, 9-1
- postprocessor 1-3
- preference 1-7, 2-15
- Preferences... 2-15
- preprocessing 1-2, 1-7, 1-8, 8-1
- preprocessor 1-3
- prescribed displacement 5-31
- primitive surface 2-21, 2-44
- Primitive Surfaces 2-44
- principal stress 7-1
- print 1-7, 1-13, 2-14
  - screen image 1-14
  - text 1-14
- prism edge mesh 4-45
- Prism Edges 4-45, 4-45
  - curve s compatible for 4-45
  - setting 4-46
- processing 1-2, 1-7, 1-11, 6-1, 6-2
  - interactive real time 6-20
  - setting options for 6-3
- project 2-9
  - file 2-11
  - new 2-10
  - starting 2-10
- project setup 2-10
- Project Setup... 2-14
- projection 1-7, 1-8
  - Mono 1-7
  - parallel 1-7
  - perspective 1-7
  - stereo 1-7
- projection mode 2-53
  - depth cued 2-53
  - perspective 2-53
  - stereo 2-53

**Q**

quarter-point element 4-78

Quit 2-14

**R**

real time processing 8-13

reanalysis 6-5

Recovery and Smoothing 2-20

rectangle 3-11

creating 3-11

Redo 2-61, 4-68

Redraw Diagram 8-10

Remesh 4-78

remeshing 4-77

rendering 1-8, 2-19

Broken mesh 1-8

by broken mesh 2-52

by outline 2-51

by shading 2-51

by transparency shading 2-51

by wireframe 2-50

Hidden line removed wireframe 1-8

mode 2-19

model 2-50

Outline-only display 1-8

Shading 1-8

style 2-50

Transparency shading 1-8

Wireframe 1-8

Reset Arrow Size 5-60

Reset Plane 7-17

reshaping 1-8

surface primitive 1-8

Reverse 2-43

Reverse Diagram 8-10

Reverse Direction 5-59

Reverse Element 8-11

revolution 4-30, 4-49

axis of 4-30

direction of 4-30

number of divisions for 4-59

Revolve 3-23

Revolve (Surface) 4-30

Revolve (Volume) 4-58

rigid body displacement 7-54

rubber-band rectangle 2-37

**S**

sampling point 7-40

sampling value 7-30

Save 2-12

Save As... 2-13

Save Image... 7-85

Save Updated View 2-42

scalar data 7-2

contour 7-3

visualization of 7-2

visualizing by contours 7-3

scale bar 7-4, 7-29

turning on and off 7-31

scrolling 1-7

seed curve 3-1, 4-21

seed surface 4-51, 4-61

Seepage 1-12, 5-26, 7-1

seepage analysis 6-14

Seepage boundary condition 5-41

Seepage property 5-27

Selected Mesh 2-43

selection 1-8

element 1-8

node 1-8

surface mesh 1-8

surface primitive 1-8

volume mesh 1-8

selection tool 2-63

selections 2-62

all 2-67

curve 2-63

element 2-64

- method of 2-65
- node 2-63
- object 2-62
- single object 2-65
- surface mesh 2-64
- surface primitive 2-63
- unselecting 2-67
- volume mesh 2-64
- Sequential Stage 5-69
- sequentially staged analysis 7-59
- Sequentially Staged Modeling 5-69
- shading 2-17, 2-50, 8-8, 8-8
  - optimum contrast 2-17
  - settings 2-17
  - smooth surface 2-17
  - transparency 2-17, 2-51
- Shape function 1-15
- shear force 8-4, 8-6
  - component 8-6
  - for 3-D rigid frame 8-7
- shell 1-11
- Show Control Points 2-48
- Show Numbers 2-46
- Show Scale 7-31
- Show Values 8-14
- skyline solver 6-22
- slicing 1-13
- snap 1-8
- solid 5-9
- solution domain 5-8
- solution method 6-7
  - direct integration 6-7
  - modal analysis 6-7
  - mode superposition 6-7
- solution stage 6-4
- solution type 6-7
  - dynamic 6-7
- Solve 6-3, 6-7
- solver 1-3, 2-20, 6-2
  - embedded 6-2
  - external 6-2
  - frontal 2-20
  - option 2-20
  - settings 2-20
- source plane 7-43
- sphere 1-9, 3-1
  - by 2 Points 3-16
  - by radius 3-15
  - creating 3-15
- splitting objects 7-18
- spring 5-31
- spring constant 5-31
- stage model 5-71
- staged stress relaxation rate 5-80
- static load 5-54
- stepwise rendering 7-59
- Stick to Control Point 2-57
- Stick to Node 2-58
- stiffness matrix 6-11
  - constant 6-11
  - updating 6-11
- straight line 1-9, 4-64
- straight lines 3-1
  - creating 3-2
- strain 7-1
- Stress recovery 1-15
- stress singularity 4-78
- Stress smoothing 1-15
- stress-strain relationship 5-10
- stresse 7-1
- structural analysis 1-11
- Structural Boundary 5-30
- structural boundary condition 5-30
  - data items 5-31
  - defining 5-30
  - fixity 5-31
  - mixed values 5-32
  - prescribed displacement 5-31
  - representation 5-36
  - spring 5-31
- structural problem 1-5
- submenu 2-5, 3-22, 4-78

- Duplicate and 3-22
  - Special Treatment 4-78
  - Surface Normal 4-79
- support reaction 8-2
  - displaying 8-3
  - of rigid frame 8-7
- surface 1-9, 3-1
  - analytic 3-1
  - B-splines 1-9
  - Bezier 1-9
  - parametric 1-9, 3-12
  - primitive 3-1
- surface mesh generation 4-3
  - 2 edges 4-12
  - 4 edges 4-1
  - automatic triangulation 4-3
  - by mapping 4-12
  - duplication 4-3
  - mapping 4-3, 4-38
  - projection 4-3
  - sweeping 4-3
- surface meshe 4-36
  - duplicating 4-36
  - mesh duplication 4-36
  - projecting 4-37
- surface normal 4-79
- surface normal direction 4-79
  - displaying 4-79
  - reversing 4-80
- Surface Plot... 7-42
- surface plotting 7-2, 7-42
  - data items for 7-43
  - options for 7-42
  - scale of height 7-44
- surface primitive 3-1, 3-2
  - B-spline surface 3-1
  - Bezier surface 3-1
  - cone 3-1
  - copying 3-19
  - creating 3-2
  - cutting 3-19

- cylinder, 3-1
  - deleting 3-19
  - flat plane 3-12
  - handling 3-19
  - mirroring 3-25
  - moving 3-19
  - parametric 3-12
  - pastings 3-19
  - plane 3-1
  - reshaping 3-19
  - sphere 3-1
  - truncated cone 3-1
- sweeping operation 4-21
  - extrusion 4-21
  - revolution 4-21, 4-30
  - twisting 4-21
- sweeping path 4-28
- system equations 6-1

## T

- temperature 5-38
- temporary file 6-5
- termination criterion 6-5
- Tetra Edges 4-47, 4-48
- tetrahedron edges mesh 4-47
- time history 7-60
  - modal participation factor 7-64
  - of nodal value 7-63
  - plotting 7-63
  - visualizing 7-60
- time step 5-56, 6-8
  - dragging 7-66
  - ending 7-67
  - number of 6-8
  - resizing window 7-67
  - size 6-8
- tolerance 2-16
  - input range 2-16
  - intersection 2-16
  - pane 2-16

- selection range 2-16
- tool button 2-2, 2-4, 2-5, 3-2, 4-1, 7-25
  - circle 3-3
  - contour marking 7-28, 7-30
  - cross plane setting 7-5, 7-6, 7-25
  - curve selection 3-33, 4-1, 4-4
  - cut plane setting 7-5, 7-6, 7-16
  - ellipse 3-7
  - elliptical arcs 3-7
  - grid 2-5
  - input 2-5
  - line 3-2
  - object movement 7-18
  - object operation 2-5
  - object rotation 7-19
  - parallel plane setting 7-5, 7-6, 7-20
  - rectangle 3-11
  - select surface mesh 7-6
  - select volume mesh 7-6
  - selection 2-5, 2-63
  - sphere 3-15
  - torus 3-18
  - view transformation 2-5
  - volume selection 7-16, 7-18, 7-20, 7-25
- tool palette 1-7, 2-2, 2-4
- torsional moment 8-4
  - diagram 8-8
  - for 3-D frame 8-8
- torus 1-9, 3-1
  - creating 3-18
- transfinite mapping 4-12, 4-45
- transient load 5-55
- transient motion 5-64
- Translate(Surface) 4-27
- translation 4-49
- transparency shading 2-50
- triangular mapping 4-12, 4-17
- truncated cone 1-9, 3-1
- truncated model 7-36
- truss 5-9
- truss element 4-63

- Twist (Surface) 4-33
- Twist (Volume) 4-60
- twist angle 4-34
- twist axis 4-34, 4-60
- twisted extrusion 4-33
- twisting 4-49

## U

- undeformed shape 7-55
- Undo 2-61, 4-68
- Unselected Mesh 2-43
- Update File Status 2-14
- user interface 1-7, 2-1, 9-2

## V

- vector 1-13
- vector data 7-2, 7-45
  - visualization of 7-2
  - visualizing 7-45
  - visualizing by arrows 7-45
- vector image 7-2
  - overlying 7-51
- Vector... 7-45
- velocity participation 7-64
- view 1-7, 2-19
  - custom 2-40
  - direction 2-31
  - exporting 2-42
  - importing 2-42
  - initial 2-41
  - last saved 2-41
  - rotating 2-31
  - settings 2-19
- view data 2-42
  - exporting 2-42
  - importing 2-42
  - updating 2-42
- view direction 1-7, 2-35
  - rotating 2-33

- setting 2-35
- View rotation 1-7
- virtual track ball 2-31
- virtual trackball 1-7
- visibility 2-43
  - recovering 2-44
  - reversing 2-43
- visual effect 7-3
- visualization 1-5, 1-13
  - 3 dimensional 1-13
  - aids 1-13
  - frame 1-6
  - scalar data 1-6
  - truss 1-6
  - vector data 1-6
  - volume 1-6
- volume mesh generation 4-38
  - automatic tetrahedronization 4-38
  - box edges 4-42
  - duplications 4-38
  - mapping 4-38
  - prism edges 4-45
  - revolution 4-49
  - sweeping 4-38
  - tetrahedron edges 4-47
  - translation 4-49
  - twisting 4-49
- volume meshe 4-62
  - duplicating 4-62
- volume visualization 1-6, 1-7, 1-12

## W

- Wilson Theta 6-8
- window 2-2, 2-6, 7-39
  - auxiliary 2-6
  - Curve Plot 7-39
  - main 2-6
  - Time History 7-67
- Windows 9-1
- wireframe 2-50

## Y

- yield criterion 5-9
  - Drucker-Prager 5-9
  - Mohr-Coulomb 5-9
  - Tresca 5-9
  - Von Mises 5-9

## Z

- zoom 1-7, 2-36
  - dial 2-36
  - factor 2-38
  - fitting to the window 2-38
  - gradual 1-7
  - in 1-7
  - in and out 2-36
  - instant 1-7, 2-38
  - locked zoom dial 2-37
  - out 1-7
  - rubber-band 1-7, 1-15
  - using rubber-band rectangle 2-37, 2-54