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.

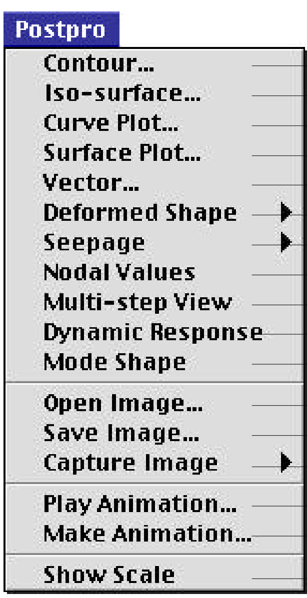
The data obtained from continuum analysis can be divided into two types. One is scalar data, and the other is vector data. Scalar data have information of magnitude only. The following functions are provided as items under **Postpro** menu for visualization of scalar data:

- 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.



Display contour image of the data
Display isosurface image of the data
Start curve plotting mode
Display the data by surface plotting image
Display vector image of the data
Display the deformed shape of the model
Display the seepage analysis results
Display the numerical values of the nodal resultants
Start multi-step view mode
Start displaying dynamic response
Start displaying dynamic mode shapes

Open an image file and display the image
Save the screen image in a file
Capture the screen image

Open an animation fole and play the animation
Create an animation and save it in a file

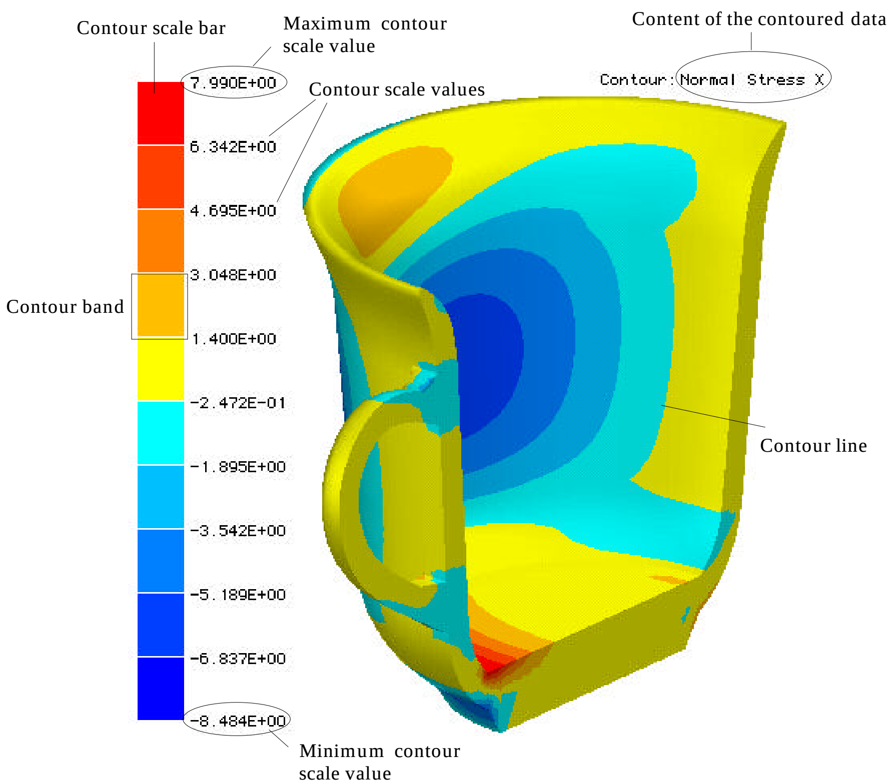
Show or hide scale bar

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.

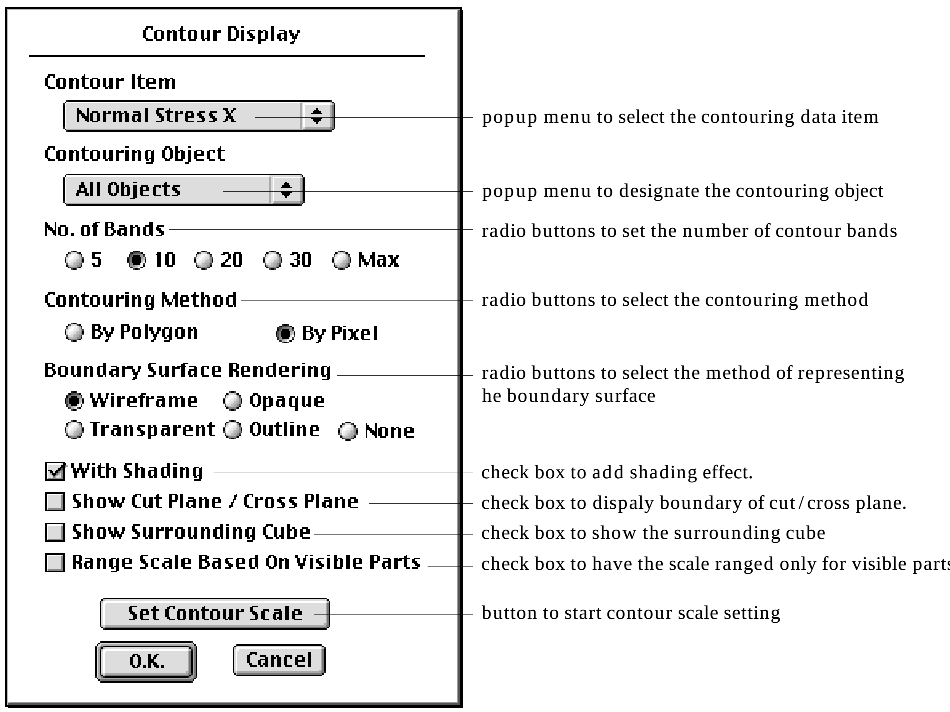
One weakness of contouring is its incapability of visualizing the data distribution inside of 3 dimensional volume. Auxiliary planes like cut plane, cross planes and parallel planes are used as volume visualization aids to make up for this limitation.



< Contour image >

Setting contouring options

In oder 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 >

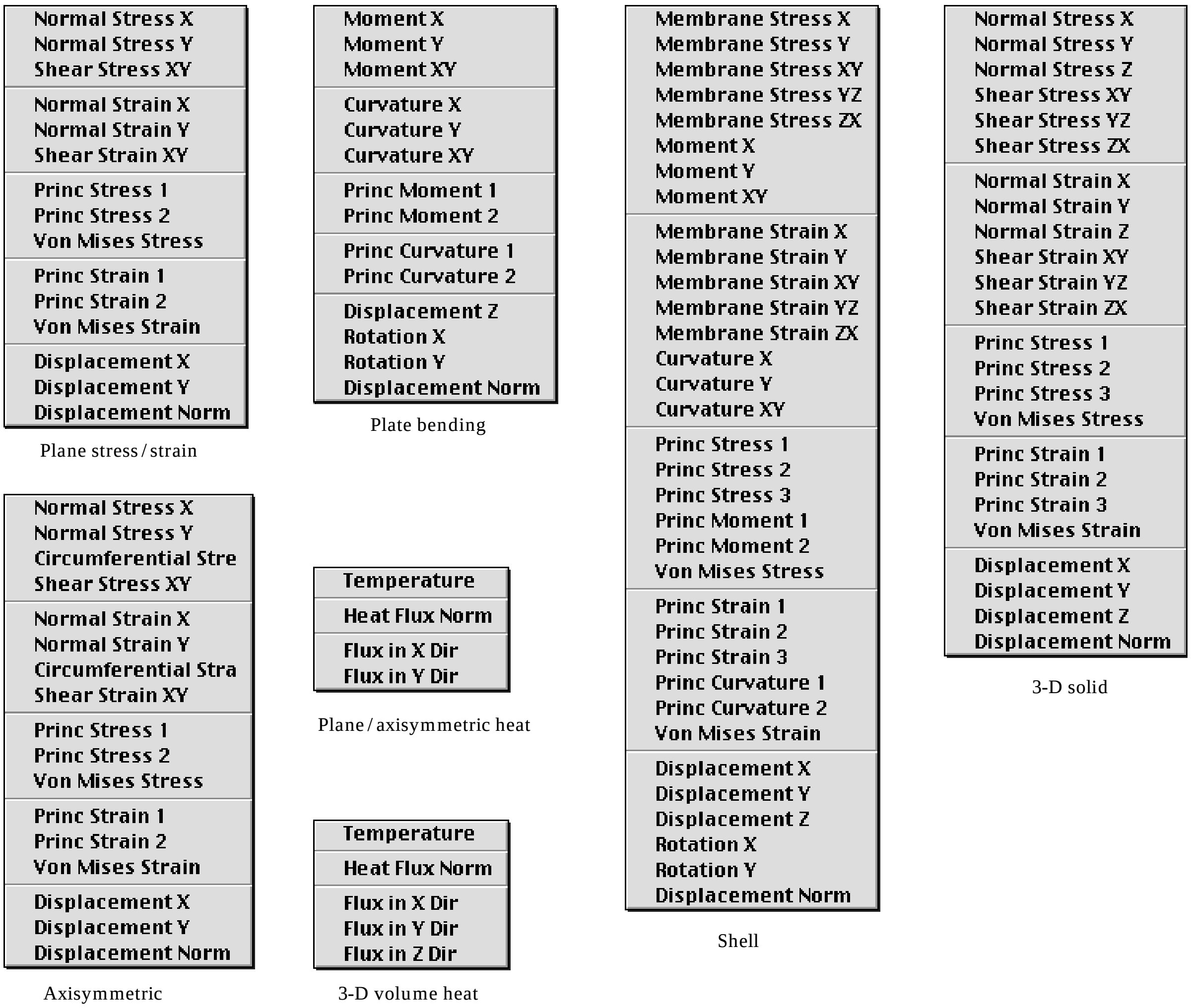
■ Selecting the data item

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.

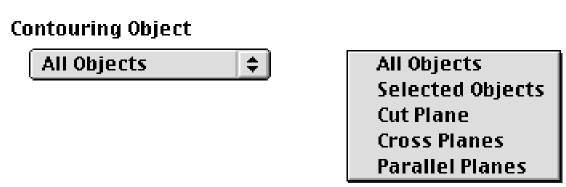
If you are using an external solver, the menu items may be different from those shown below, because the popup menu items can be customized by the external



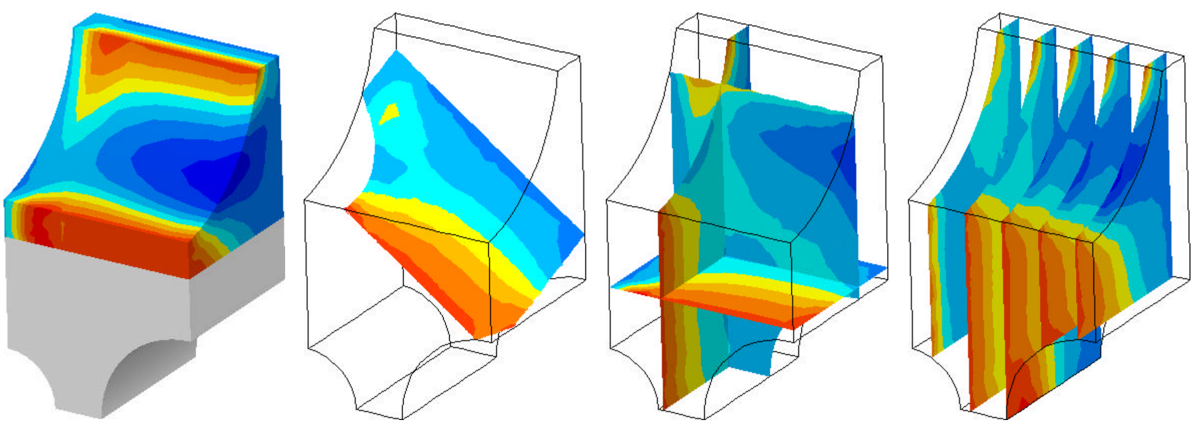
<Popup menu selecting the contouring data item >

■ 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 .



- “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.

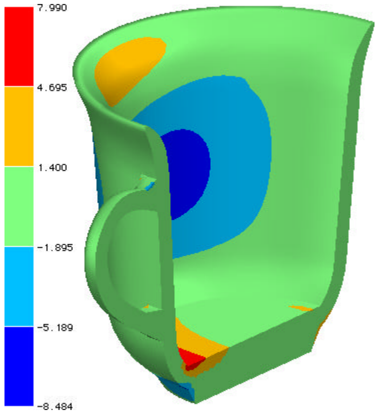


Selected objects Cut plane Cross planes Parallel planes

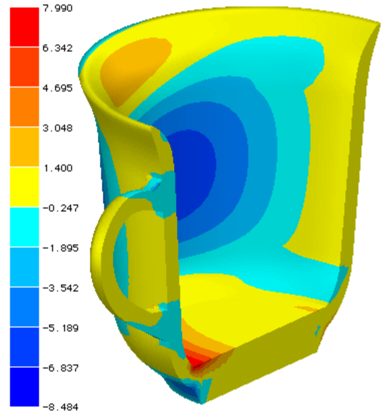
<Various contouring objects>

■ Setting the number of contour bands

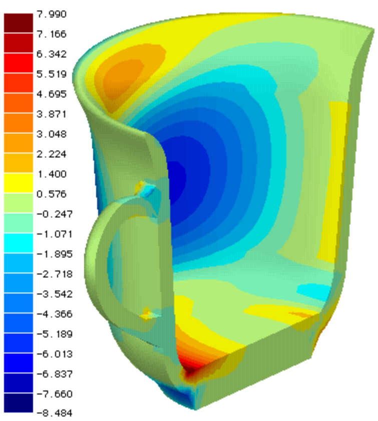
The contour image has a number of contour bands, each of which represents a certain range of data value. The number of bands is initially set as 10 by default. This default setting can be changed by clicking the radio button labeled with the desired number of bands, which should be one of 5, 10 , 20, 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.



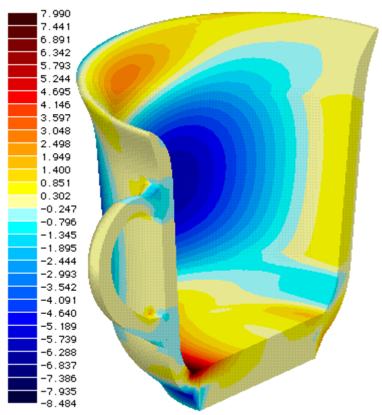
Number of contour bands = 5



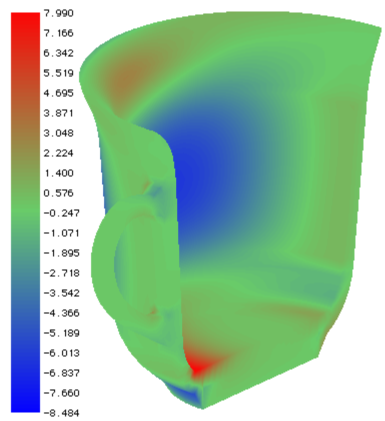
Number of contour bands = 10



Number of contour bands = 20



Number of contour bands = 30



Number of contour bands = Max

<Comparison of number of contour bands>

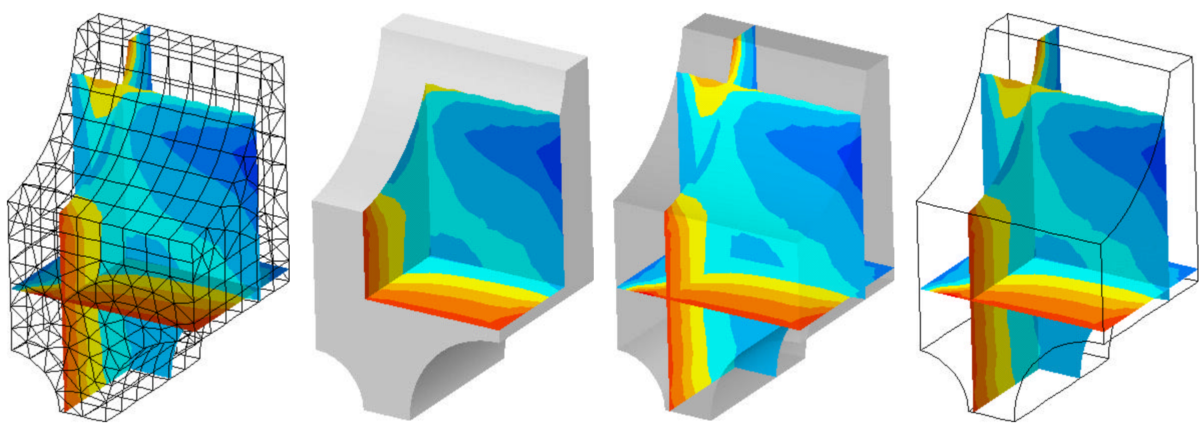
■ Selecting the contouring method

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.



With wireframe mesh   With opaque shading   With transparent shading   With outline

<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

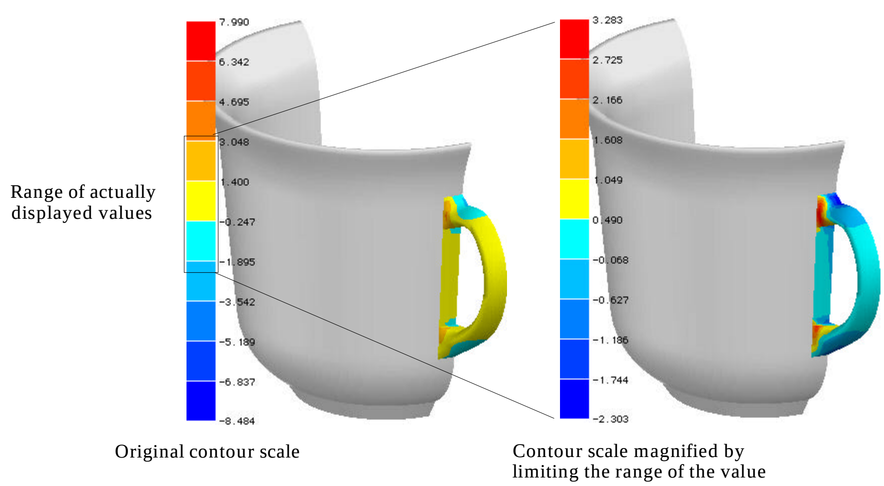
The boundary lines of a cut plane or cross planes are displayed together with the contour image, if “Show Cut Plane/Cross Plane” box is checked. This option is valid only when the contour is drawn over the cut plane or the cross planes defined prior to starting “Contour Display” dialog.

■ 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

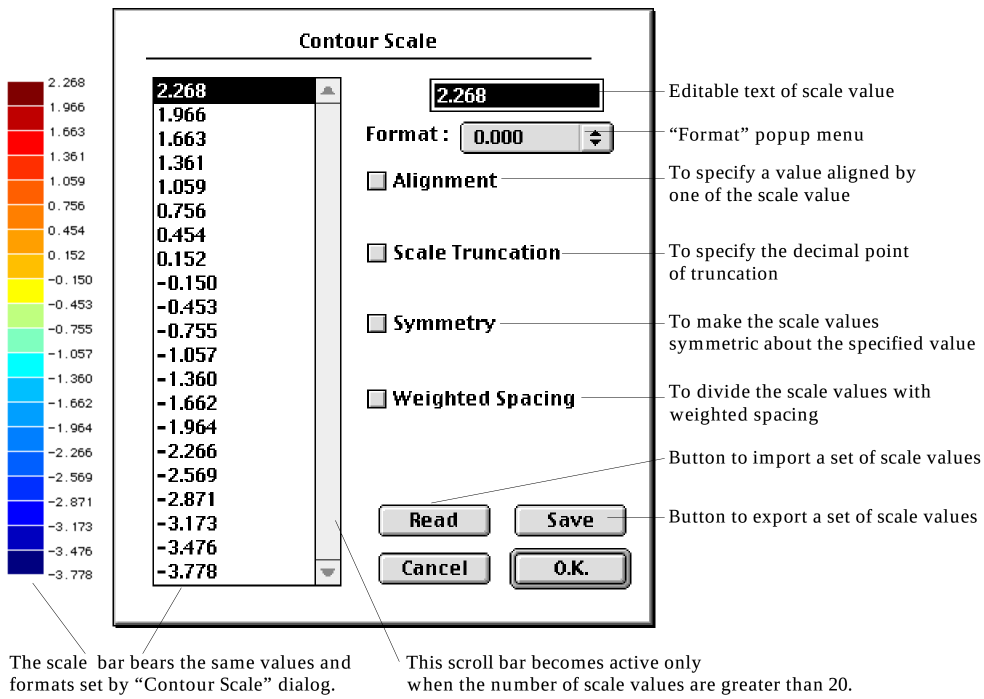
The contour image is not always drawn for the entire model. A contour image may be drawn only on the selected parts or on the visible parts. In such cases, the range of the actually contoured data may be much narrower than that of the whole data. The scale of the contour can be adjusted to the actual 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.



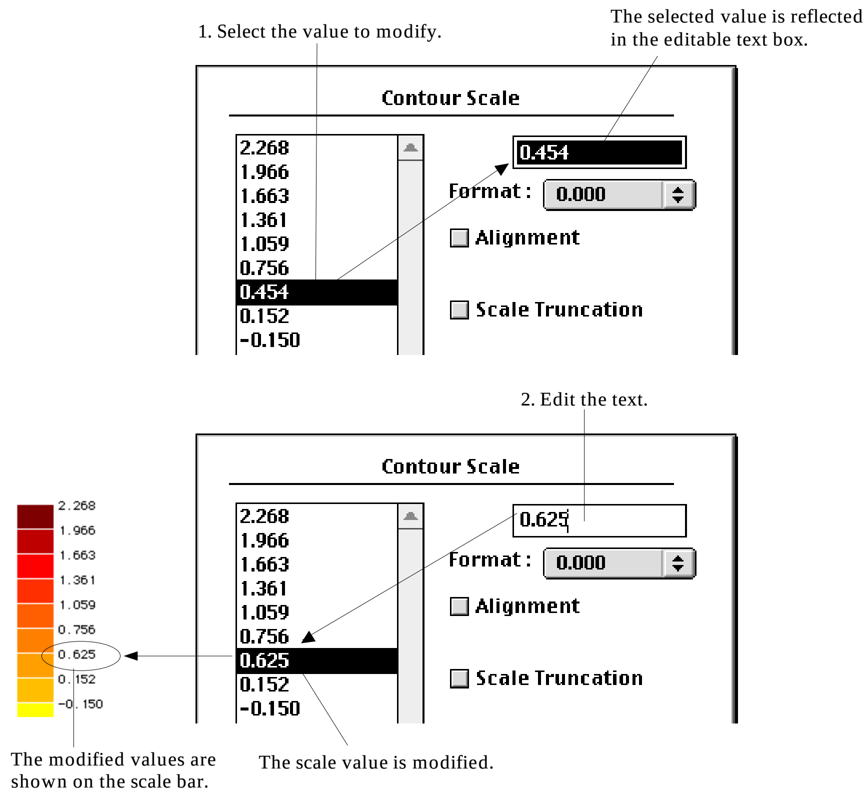
<Scale bar and “Contour Scale” dialog >

■ 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.

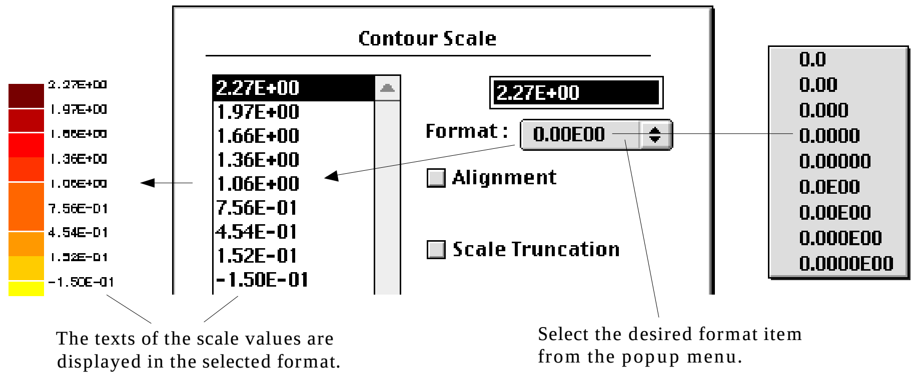
3) Repeat step 1) and 2) for all scale values to modify.
   This modification mode is terminated by clicking any other buttons, check boxes or editable texts.



<Editing scale values >

■ Setting the format of the contour scale

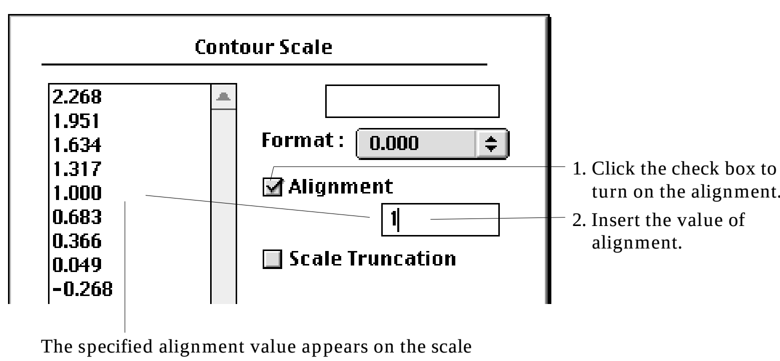
The contour scale values may be displayed in a number of different formats. The format can be set using “Format” popup menu in the dialog. The popup menu includes the various format items shown below. Select the desired item from the popup menu. Then, the text of the scale values shown in the dialog 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

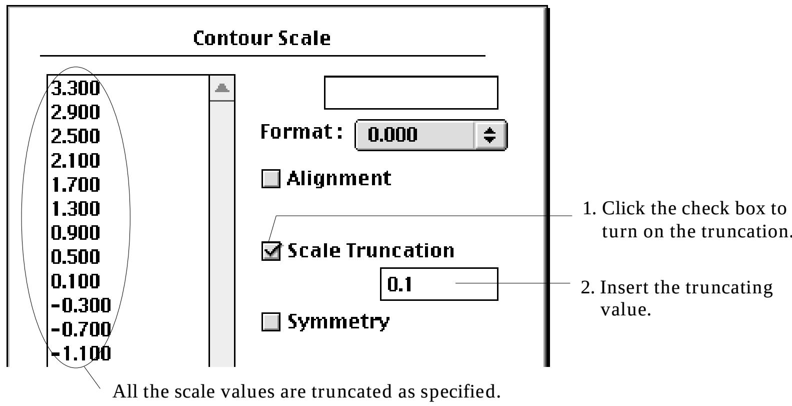
It is sometimes necessary to display a specific value by a contour line. This can be achieved by forcing one of the scale values to be aligned to the specified value. Check “Alignment” box by clicking it. Then, an editable text item appears below the check box. Insert the value specifying the alignment 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

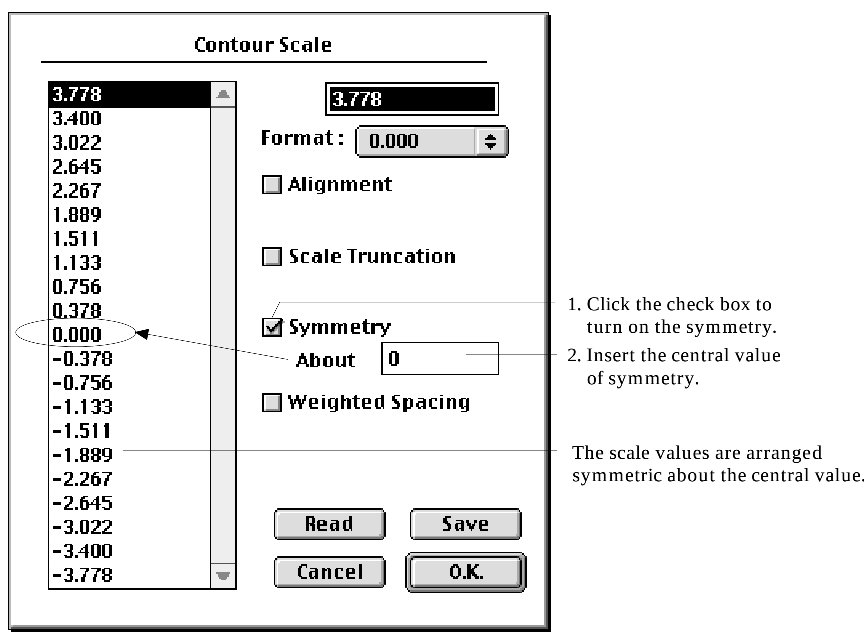
The scale values may be determined so that each of them is truncated at the specified decimal point. Check “Scale Truncation” box by clicking it. Then, an editable text item appears below the check box. Insert into this text box the value indicating the decimal point for truncation. The scale values 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.



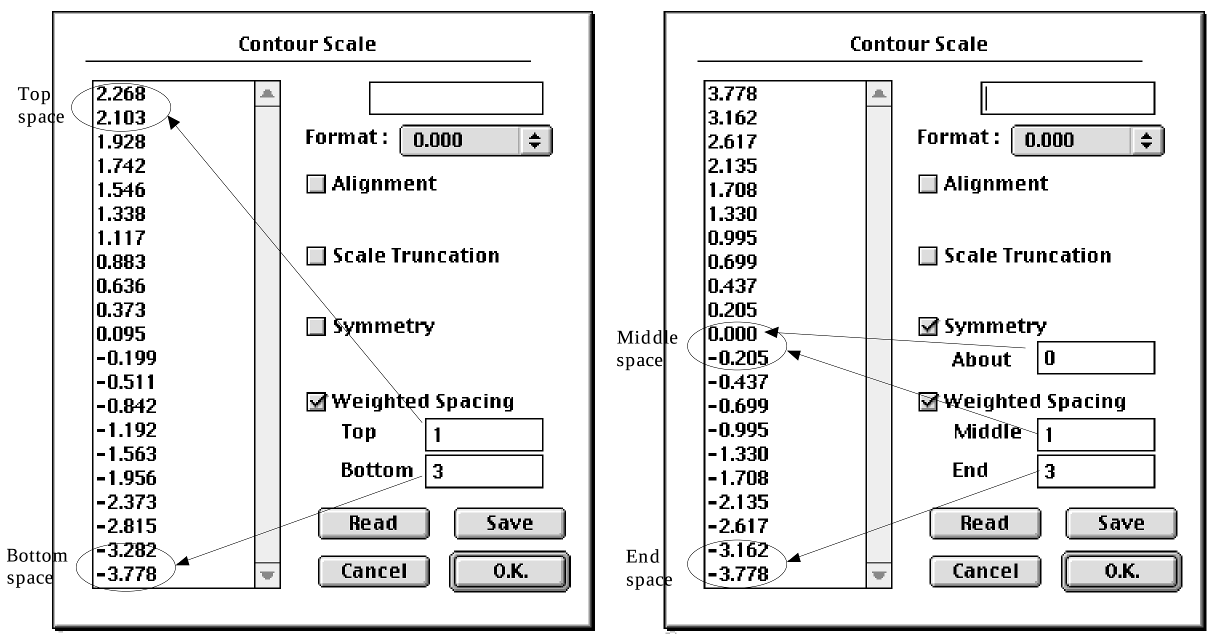
<Arranging the scale values symmetrically>

■ 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.



Without symmetric arrangement With symmetric arrangement

<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
☑	☐	☐	☐	☑	☐	☐	**Alignment**
☐	☑	☐	☐	☑	☑	☐	**Scale Truncation**
☐	☐	☑	☐	☐	☑	☑	**Symmetry**
☐	☐	☐	☑	☐	☐	☑	**Weighted Spacing**

■ Saving contour scale values

A set of contour scale values can be saved in a file for future use. Click **Save** 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 **Read** 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.

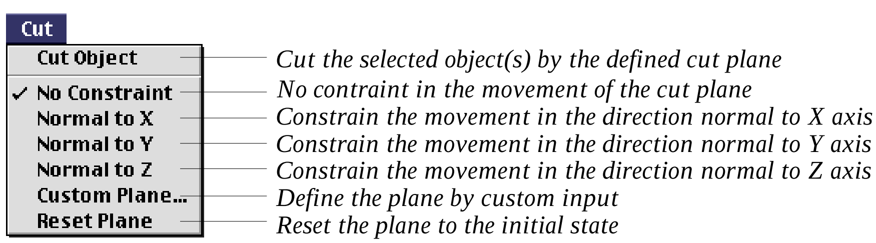
Setting a cut plane for contouring

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.

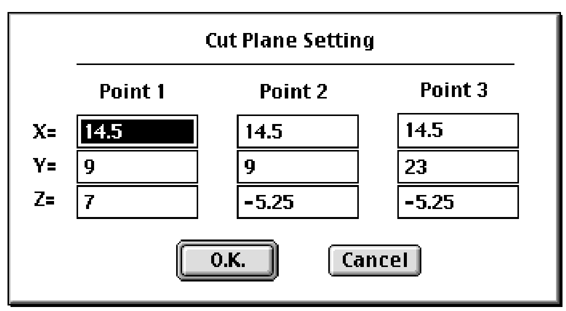


The cut plane setting mode is deactivated simply by activating other tool in the tool palette.

■ 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.

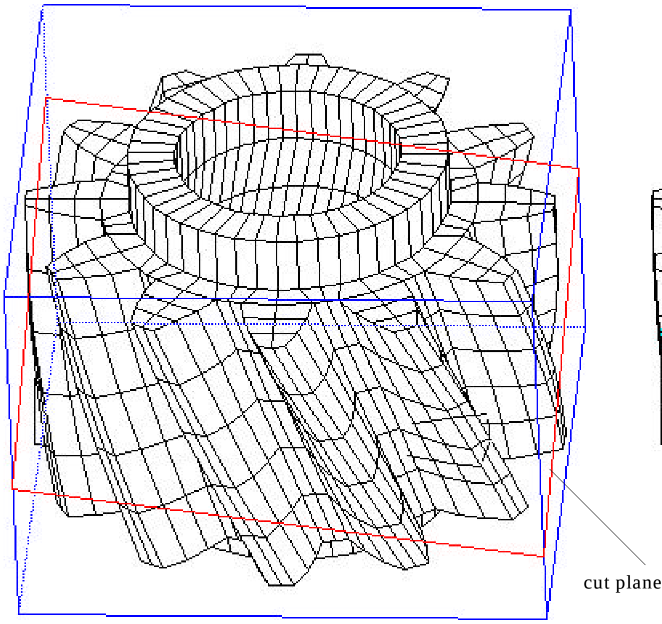


In order to reset the cut plane setting to the initial state, select “Reset Plane” menu item from **Cut** menu.

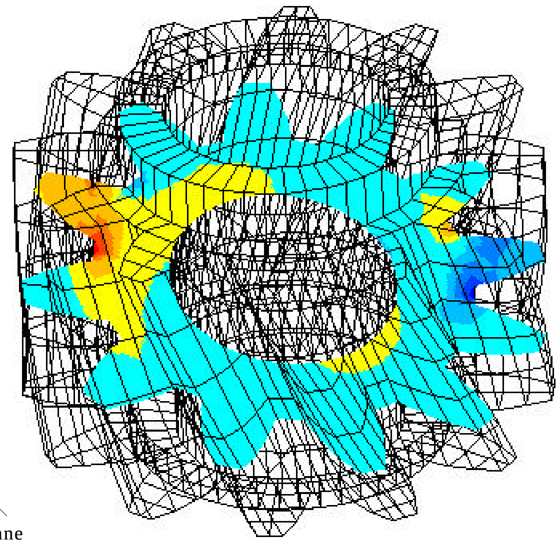
■ Contouring on a cut plane

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.

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

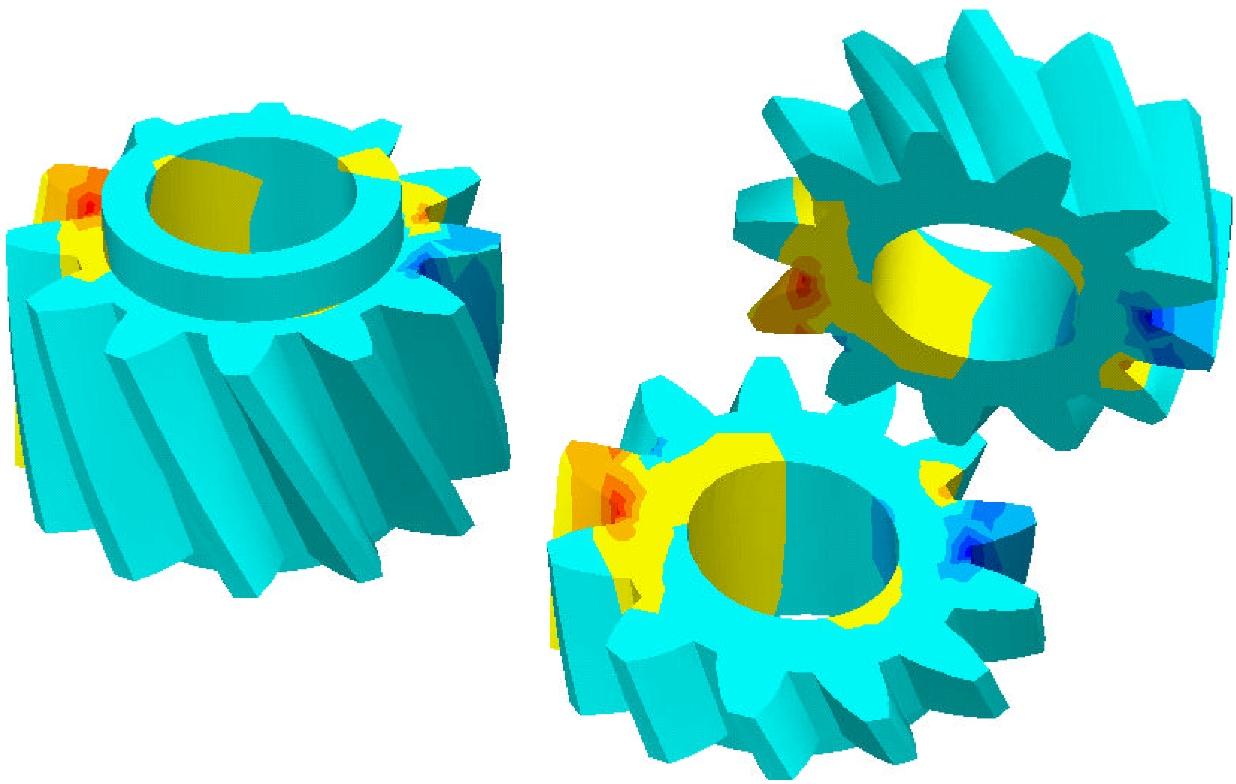
■ Splitting objects using the cut plane

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

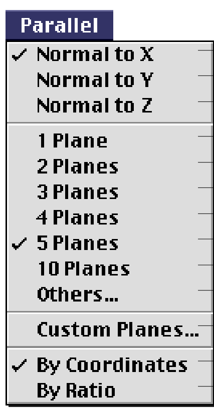
Setting parallel planes for contouring

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.



Set the parallel planes normal to X axis
Set the parallel planes normal to Y axis
Set the parallel planes normal to Z axis
1 parallel plane
2 parallel planes
3 parallel planes
4 parallel planes
5 parallel planes
10 parallel planes
Set the number of palallel planes
Define the parallel planes by custom input
Represent the location of a parallel plane by coordinates
Represent the location of a parallel plane by ratio

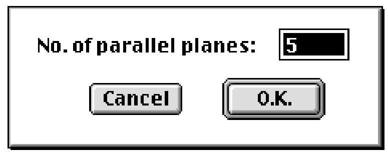
The parallel plane setting mode is deactivated simply by activating any other tool in the tool palette.

■ 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.

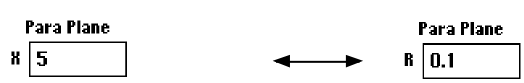


- Locating the parallel planes by mouse dragging : The parallel planes can be located at desired points by dragging each of the planes using mouse. Place the screen cursor over the image of a parallel plane, and drag the plane to the desired point. The plane moves along with mouse movement.

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.



Location of parallel plane represented by coordinates

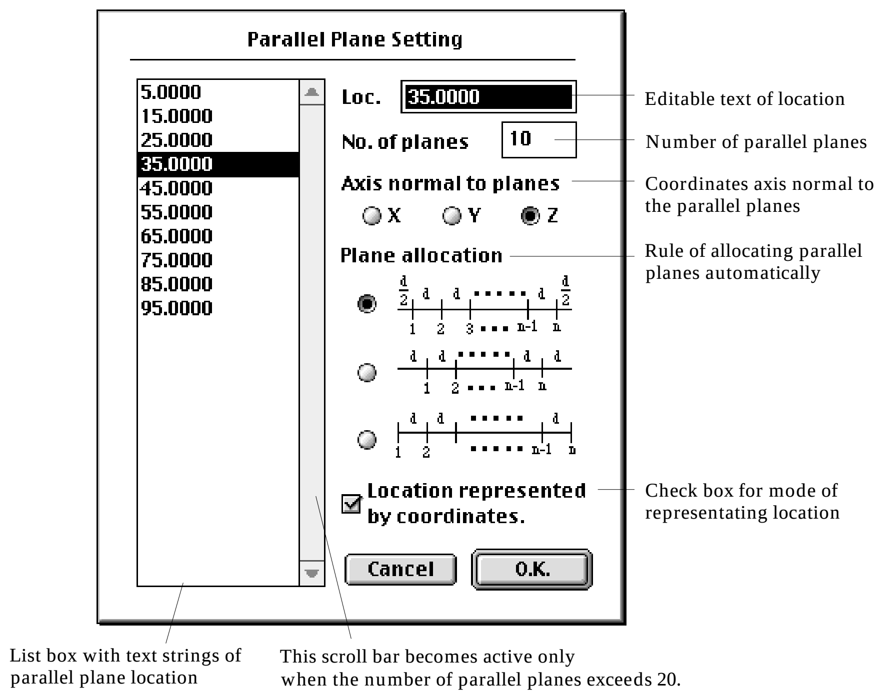
Location of parallel plane represented by relative coordinates (ranging from 0 to 1)

< 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.

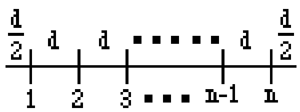
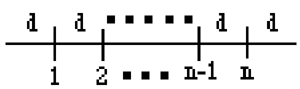
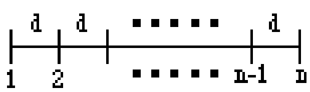


< Parallel plane setting dialog >

- 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.

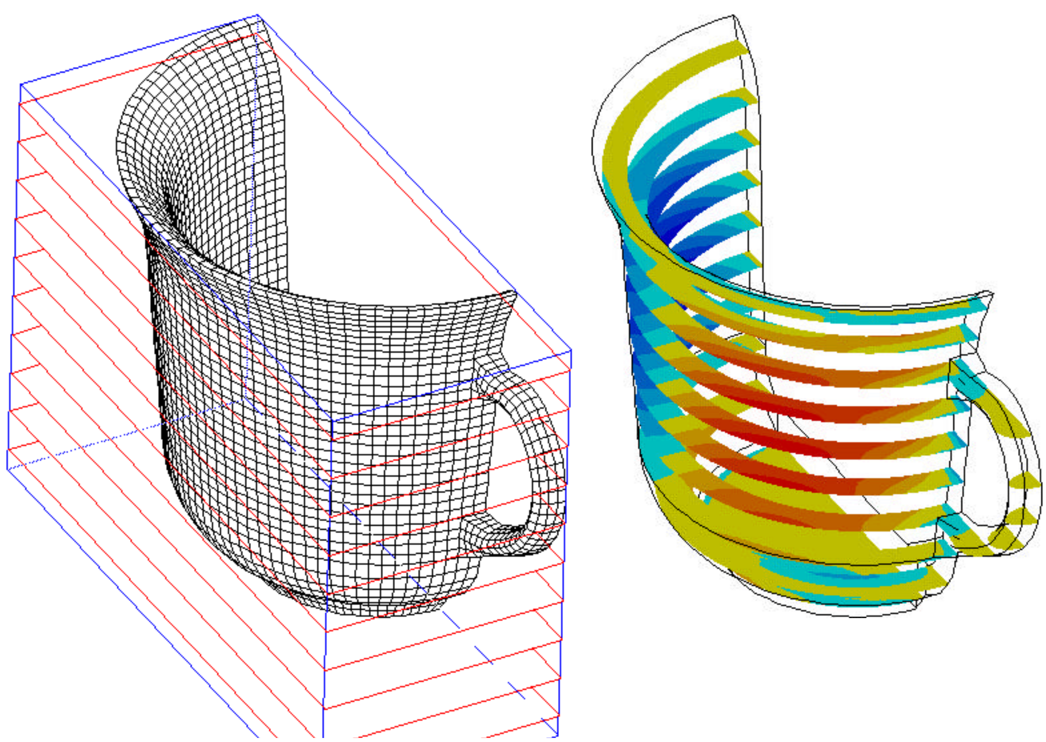
3) Repeat step 1) and 2) for all the parallel plane locations to modify.

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.

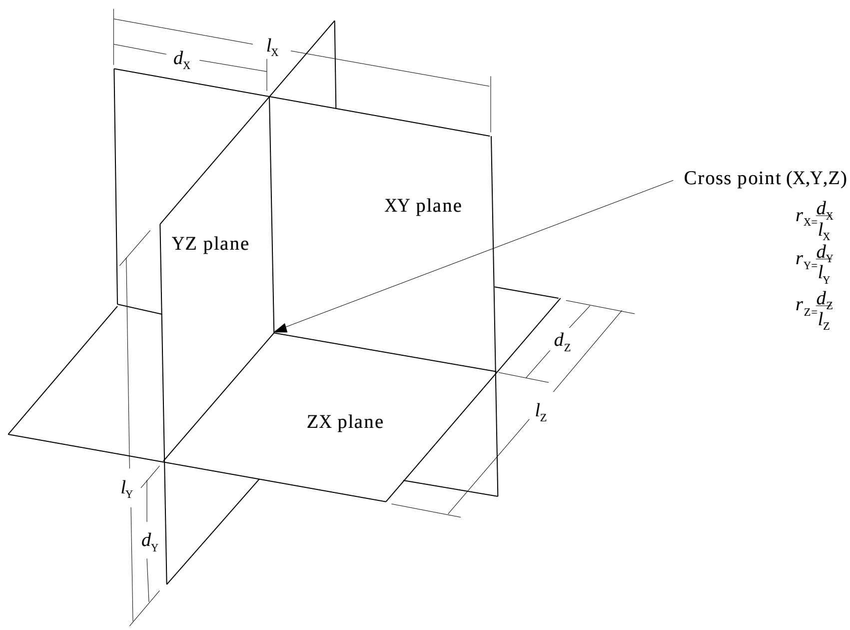
There are options setting the style of the surrounding boundary surface rendering. Refer to "Selecting the style of boundary surface rendering" for more detailed description on this subject.



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.



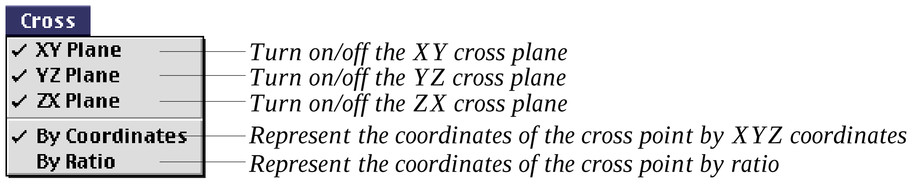
< Cross planes and cross point >

■ 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.

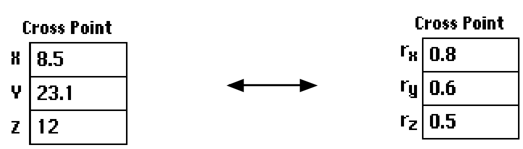


The cross plane setting mode is deactivated simply by activating other tool in the tool palette.

■ 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", "XY 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.



Location of the cross point represented by coordinates

Location of the cross point represented by ratio

< 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:

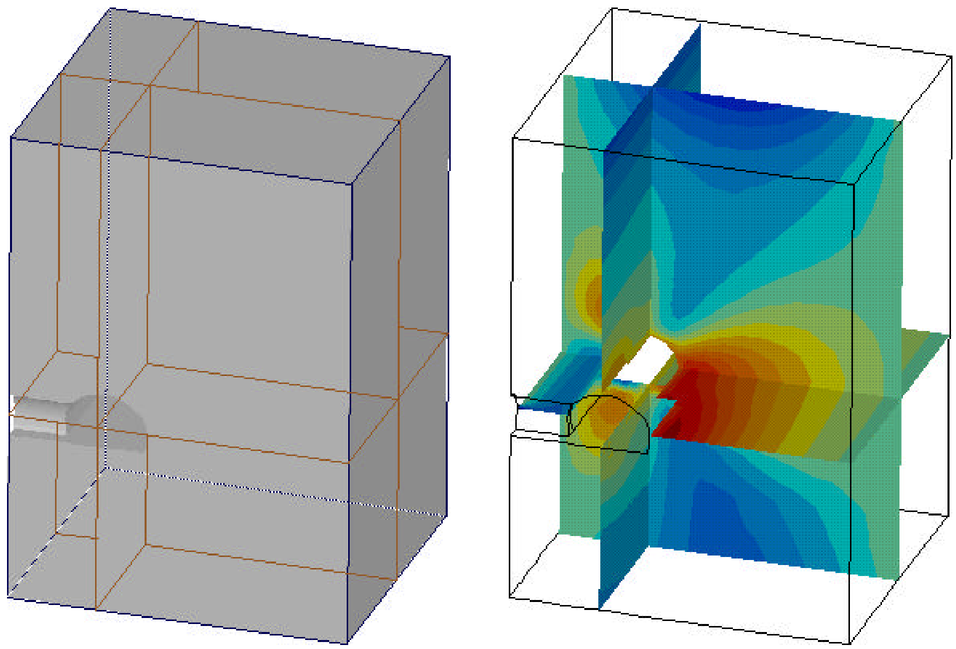
1) Set the appropriate mode of representing the location of the cross point.
   The mode can be set by choosing “By Coordinates” or “By Ratio” item in Cross menu.
2) Edit the text.
   Change the location by editing the text of location represented either by coordinates or by ratio.
2) Press return key (Windows : Enter key).

   The edited text of the location is applied by pressing return key (Windows : Enter 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.

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.



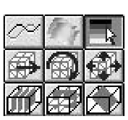
Cross plane setting

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.

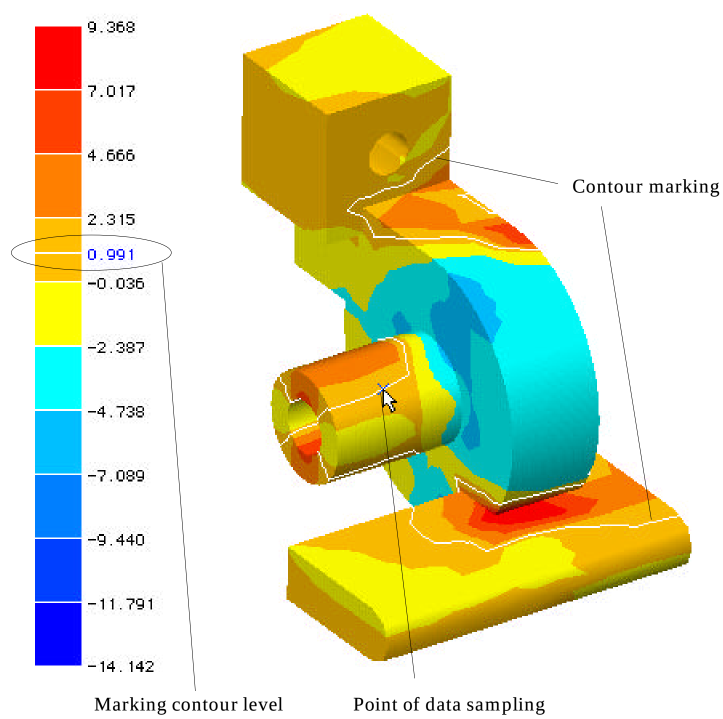
As the cursor moves into the VisualFEA window, its shape turns into . A data value may be sampled either at the point of the cursor on the contour image or on the contour scale bar. Place the cursor over the point from which the data value is to be sampled.

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 marking >

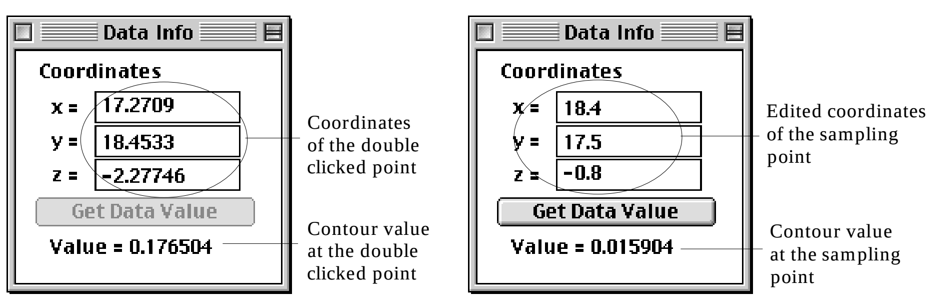
The above step 2), 3) and 4) may be repeated as long as the contour marking tool is active. The editable text box at the lower part of the tool palette shows the level of the marked contour. The contour marking may also be renewed by inserting a value in the box and pressing return *key (Windows :* Enter *key).*

The precision of the marking contour line is determined by the viewing scale of the model and the screen resolution. Therefore, you can obtained 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.

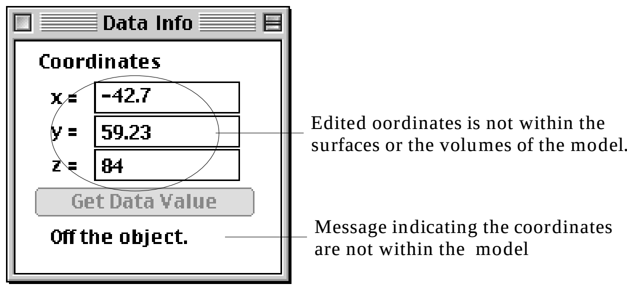


< “Data Info” dialog >

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.

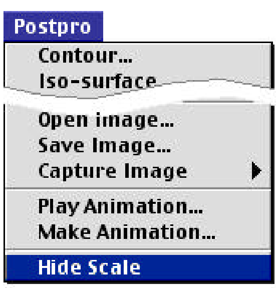
5) Press **Get Data Value** button.

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.

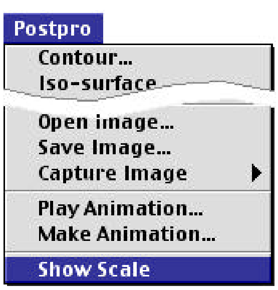


< "Off the object" message >

■ 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.

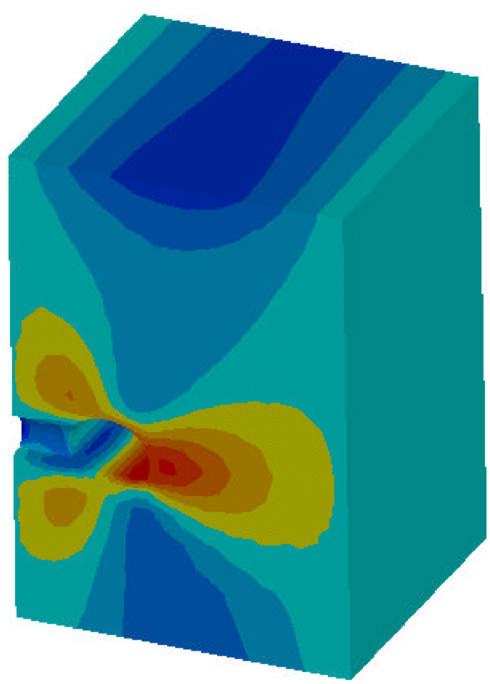


Visualizing Scalar Data by Iso-surface and others

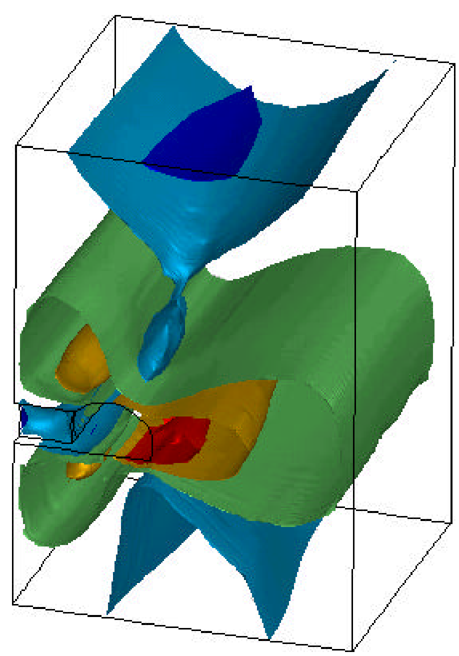
Besides contouring, VisualFEA supports other popular methods of visualizing scalar data in 2-D or 3-D space: iso-surface rendering, curve plotting and surface plotting. They are not so widely used as contouring in other finite element analysis software. But they are as efficient and convenient as contouring, 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

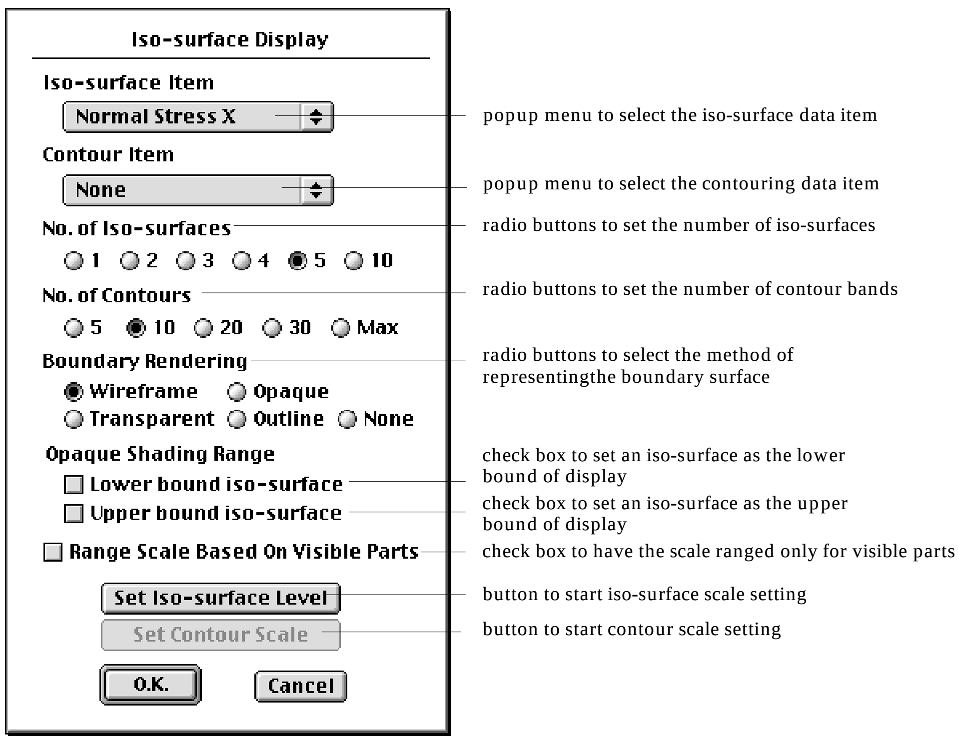


Isosurface image

<Comparison of contour and iso-surface images>

■ Setting the iso-surface display options

In oder 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.



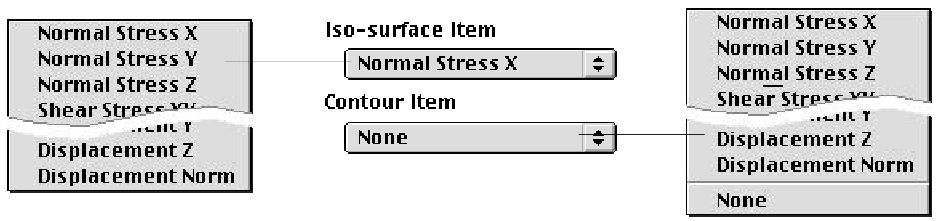
<”Iso-surface Display” dialog >

■ 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

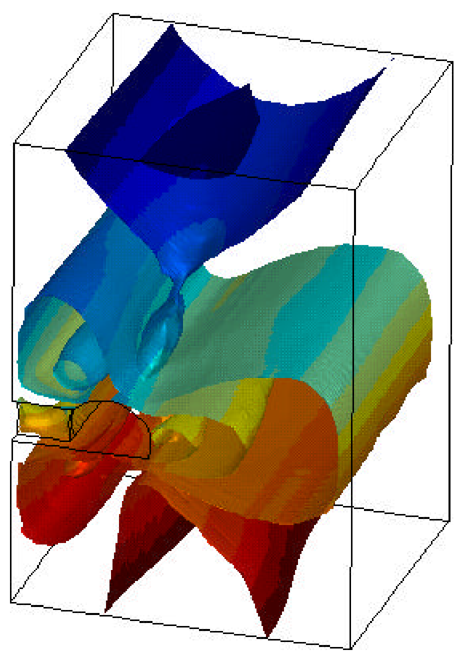
Iso-surfaces may be painted with contour bands which represent distribution of another data item. For example, iso-surfaces represent the distribution of normal stress σ_x, and contours represent the distribution of shear stress τ_{xy}.
The second popup menu has the menu items to select the contour data 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 menu to select iso-surface item

Popup menu to select contour item

< 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.

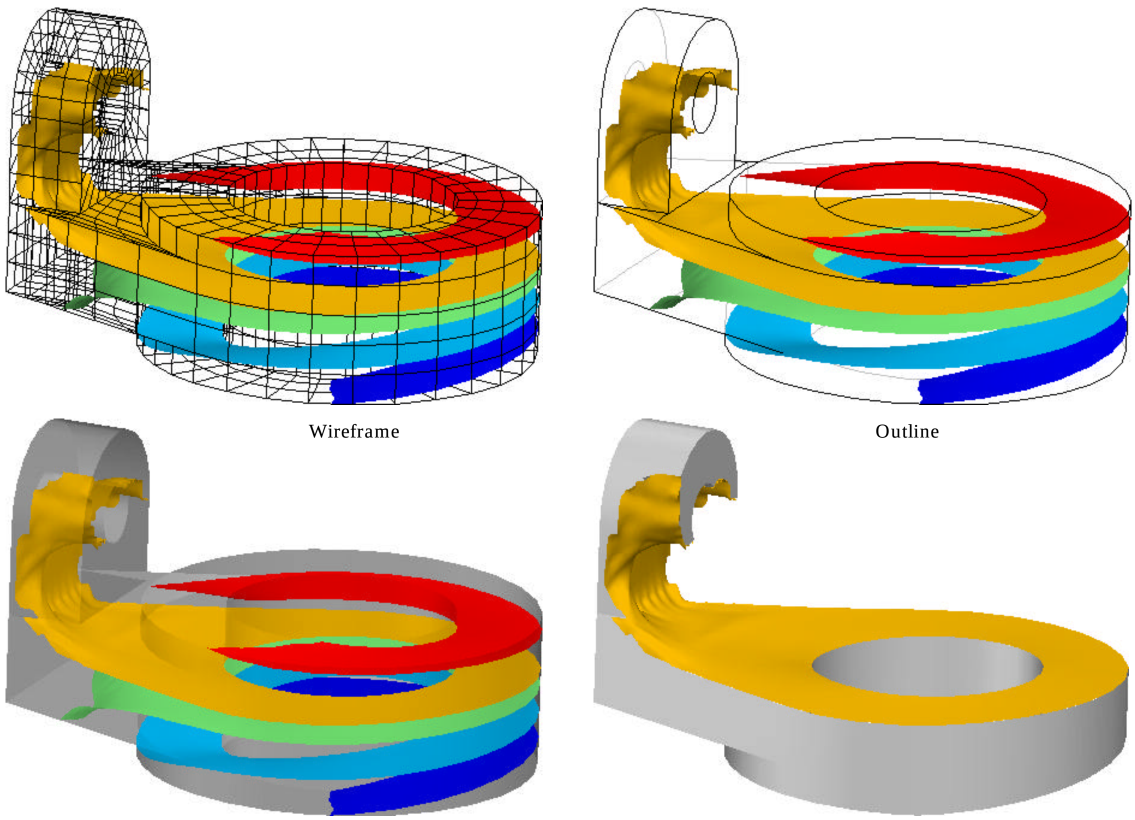
■ Setting the number of contour bands

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.



Wireframe Outline

Transparent shading Opaque shading

< Boundary surface rendering with iso-surfaces >

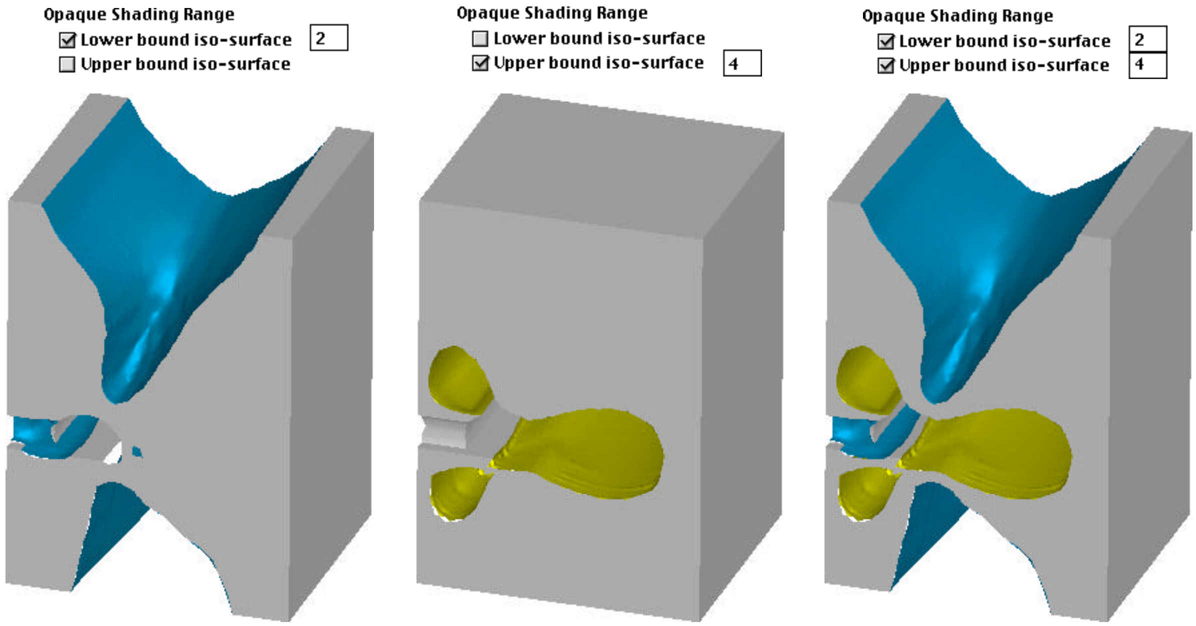
■ Designating the iso-surfaces as the boundary of a truncated model

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.



Lower bound iso-surface   Upper bound iso-surface   Lower and upper bound iso-surface

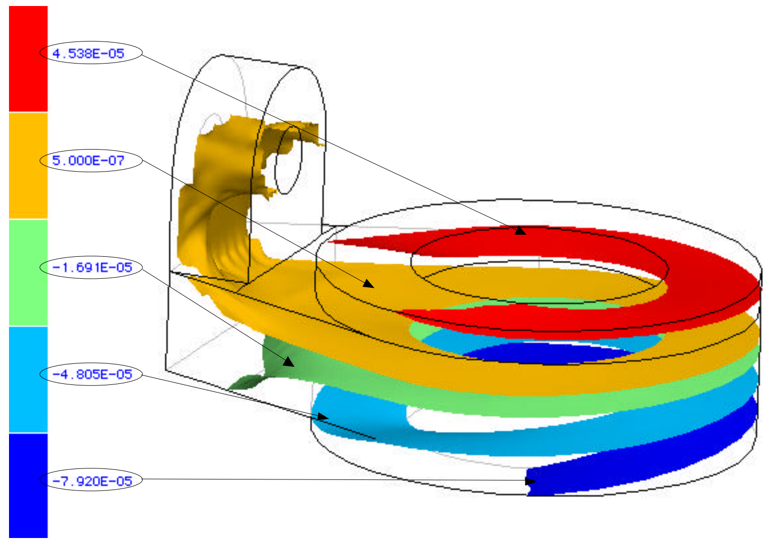
< Model truncated by iso-surfaces >

■ 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

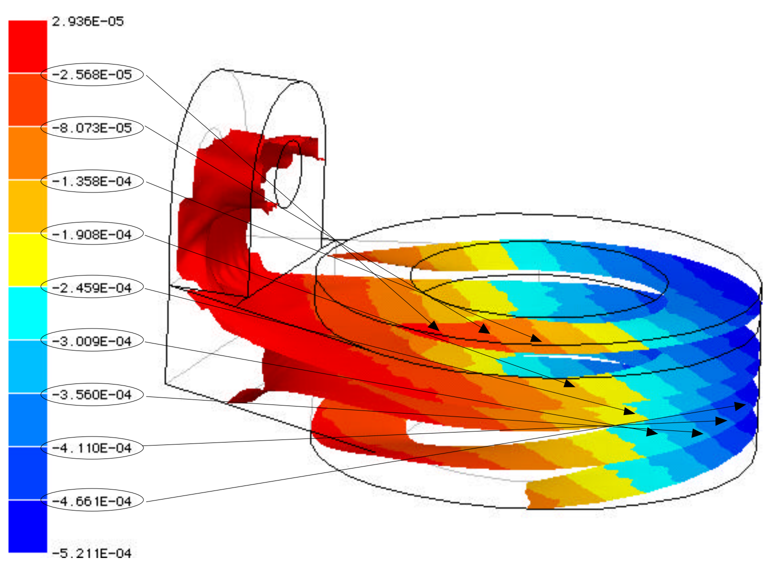
An iso-surface splits the model into two regions, one with data value just above and the other just under the data value represented by the iso-surface. This data value is termed here as iso-surface level. Each of the iso-surface levels has a value between the maximum and minimum values of the data to 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.



< Scale bar for iso-surfaces >

■ Setting the contour scale

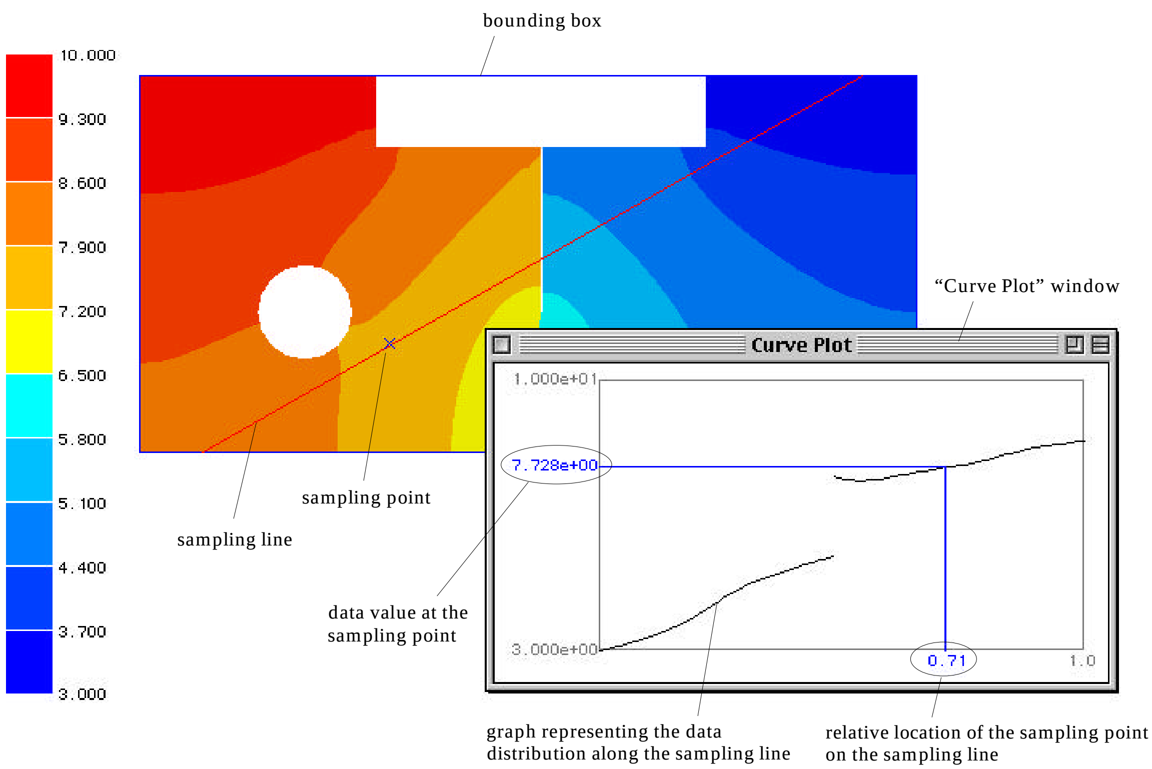
In case the iso-surfaces are displayed with contours, the data represented by the contours are scaled independently from that of the iso-surface. In order to adjust the contour scale, first click **Set Contour Scale** button in "Iso-surface Display" dialog. Then, "Contour Scale" dialog will appear on 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.



< Scale bar for contours on iso-surfaces>

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.
   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

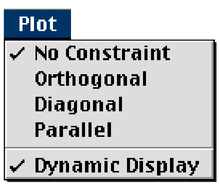
A point on the sampling line can be designated as the sampling point. Click one point on the sampling line. Then, the sampling point is set at the point, and indicated by × mark. The numerical value of the data on the sampling point is displayed in a character string on “Curve Plot” window. The relative 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.



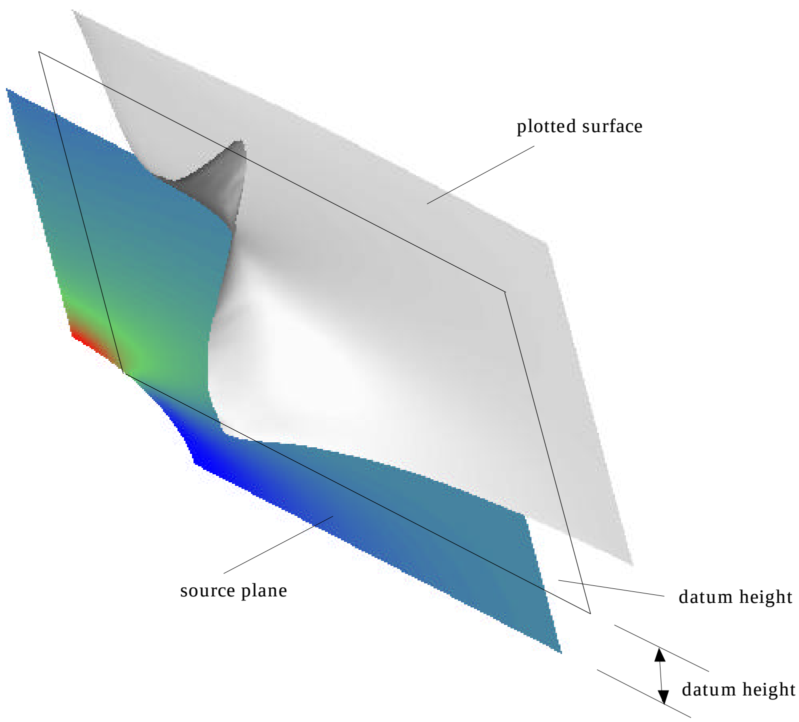
- Controlling the movement of the sampling line.
  The movement of the sampling line is controlled in the manner set by the menu items in **Plot** menu.
  - “No Constraint” : The sampling line moves without constraint. While one end of the sampling line moves, the position of the other end is not altered. 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

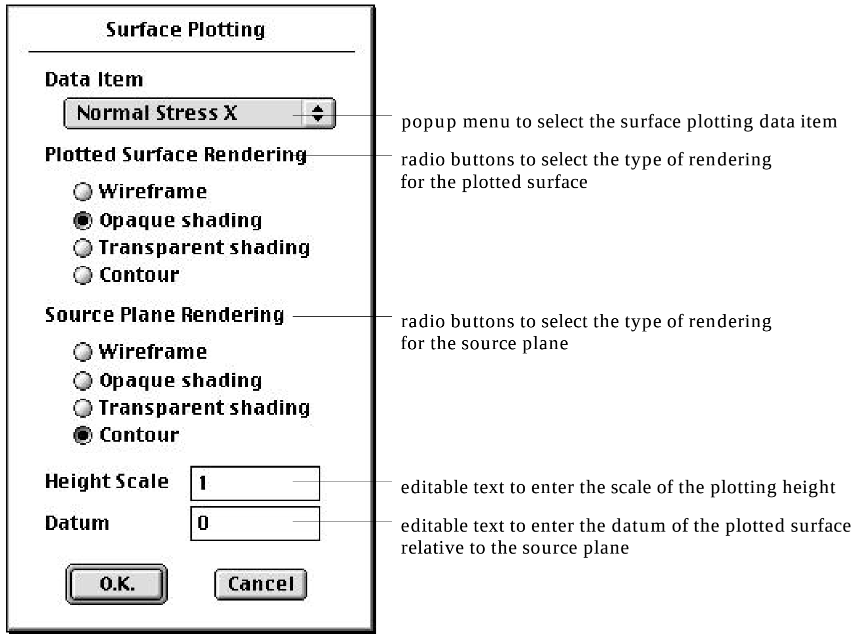
Surface plotting is a method of visualizing 2-dimensional scalar data distribution by a 3-dimensionally rendered surface over the data plane. The magnitude of the data at a point is represented by the height of the surface at that point. Surface plotting involves two surfaces, the source plane and the 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 oder 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.



< "Surface Plotting" dialog >

■ 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”:

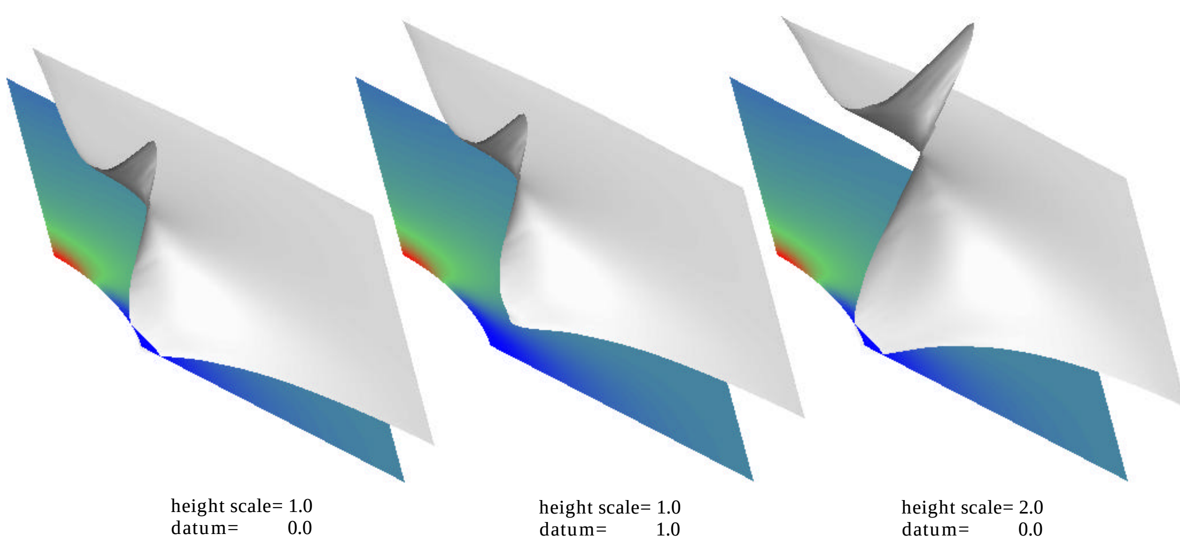
- “Wireframe”: The source plane is rendered in the form of wireframe.
- “Opaque shading”: The source plane is rendered by opaque shading.
- “Transparent shading”: The source plane is rendered by transparent shading.
- “Contour”: The source plane is rendered by a contoured surface.
- “None”: The source 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.



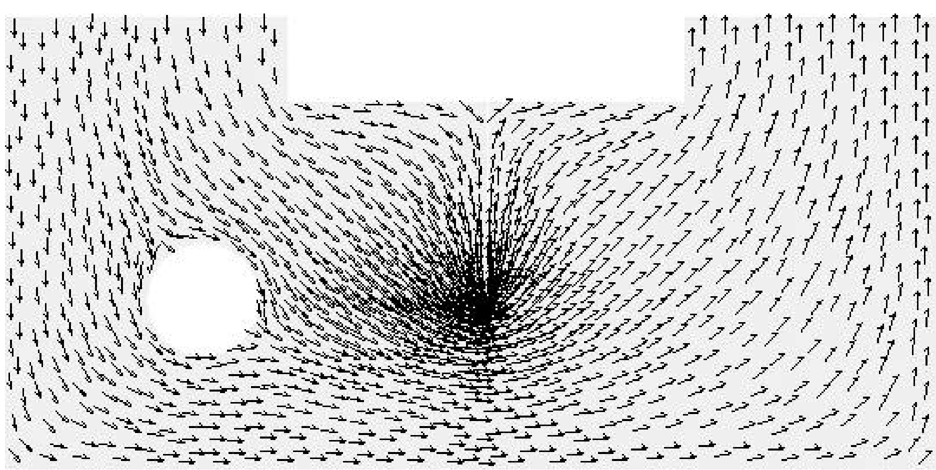
< Comparison of surface plotting with various height scale and datum >

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

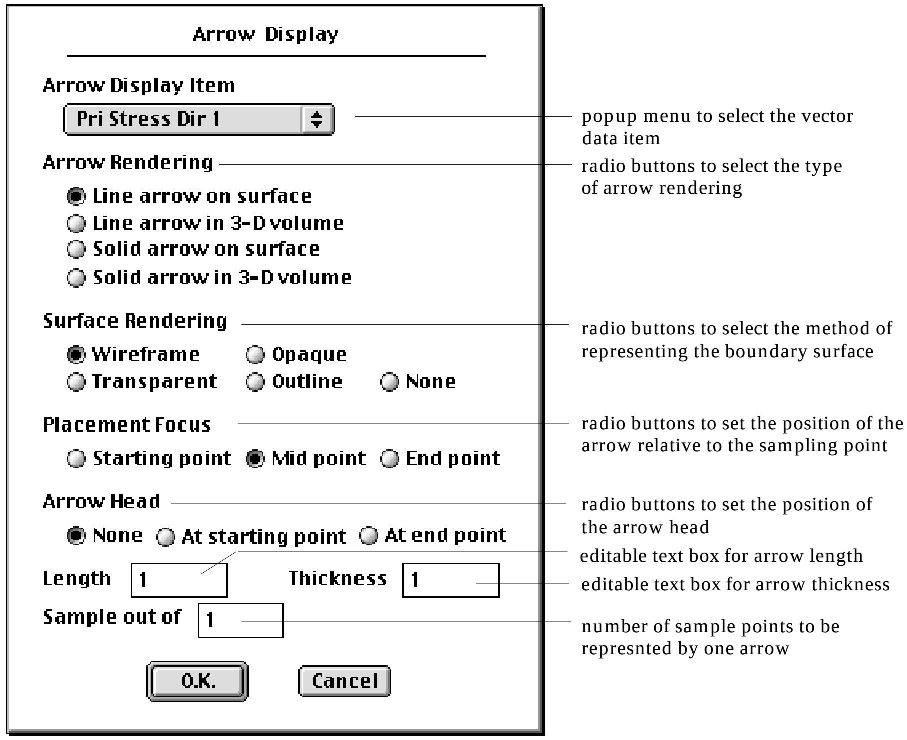
Arrows are usually used to visualize vector data. The magnitude of the data is represented by their lengths, and the direction of the data by arrow-head directions. In case of 3-dimensional solid analysis, arrows may be rendered as 3-dimensional solid objects. The following figure shows an example of 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 oder 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.



<"Vector Display" dialog >

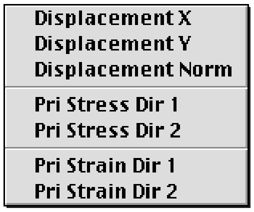
■ Selecting the data item

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.



Plane stress/strain
& axisymmetric

2-D plane heat
axisymmetric heat
3-D volume heat

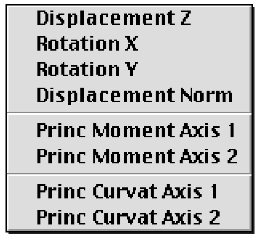
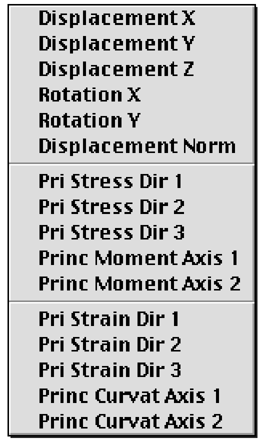
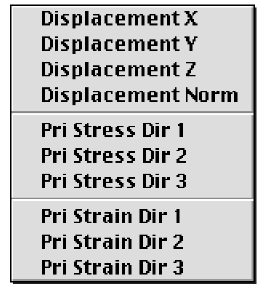


Plate bending



Shell



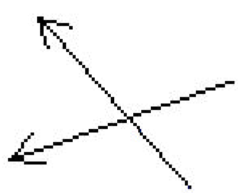
3-D solid

<Popup menu selecting the arrow display data item >

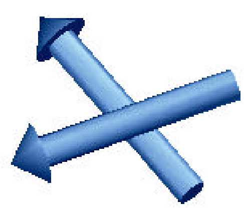
■ Selecting the type of arrow rendering

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



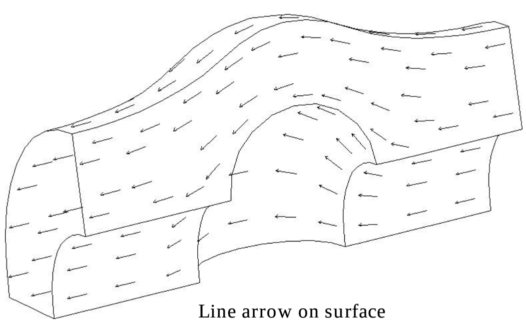
Solid arrow

< Popup menu selecting the arrow display data item >

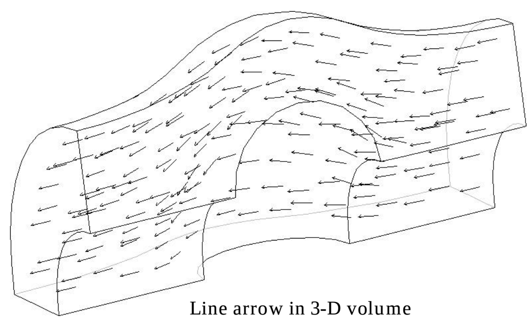
■ Selecting the style of surface or boundary surface rendering

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:

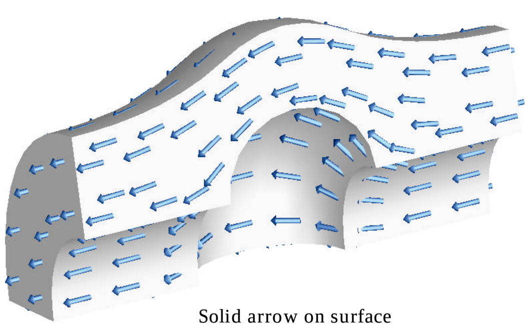
- "Wireframe": The boundary surfaces are rendered in the form of wireframe. The hidden lines are removed from the wireframe rendering.
- "Opaque": The boundary surfaces are rendered by opaque shading. If the type of arrow rendering is set to "Solid arrow in 3-D volume", this option is not applicable, 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.



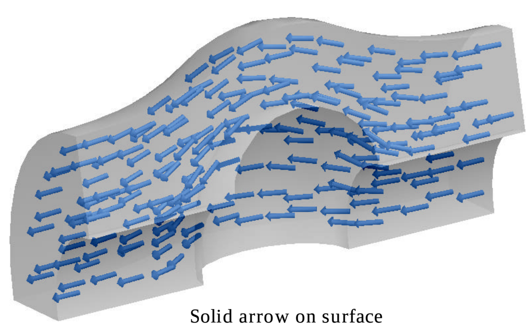
Line arrow on surface
with outline rendering



Line arrow in 3-D volume
with outline rendering



Solid arrow on surface
with opaque shading



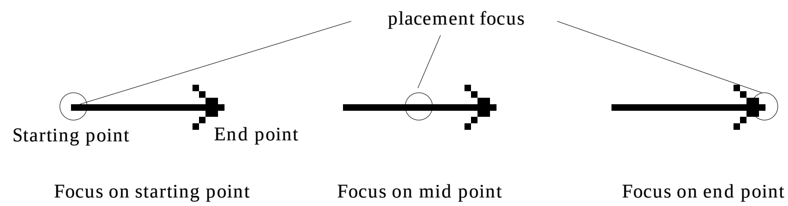
Solid arrow on surface
with transparent shading

< 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 a n 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.

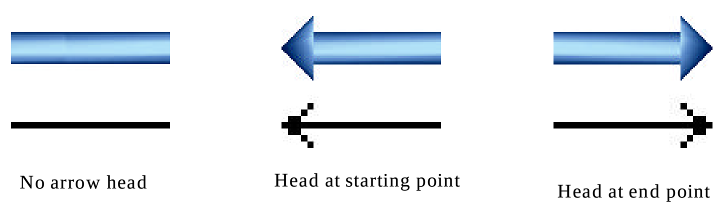


< Arrow placement >

■ 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:

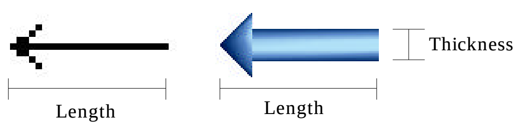
- "None" : Arrow heads are not drawn. An arrow is represented by a straight line. This option is appropriate for the case in which the sign of the direction is not important. For example, the directions of principal stresses are usually drawn without arrow heads.
- "At starting point" : The arrow head 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.



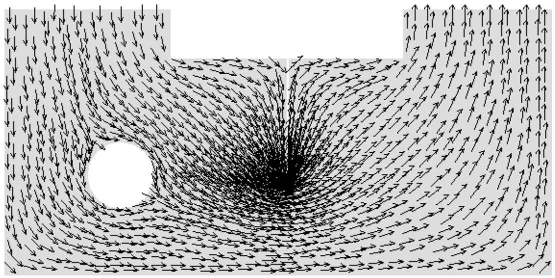
< Length and thickness of arrow >

■ Setting the thickness of arrows

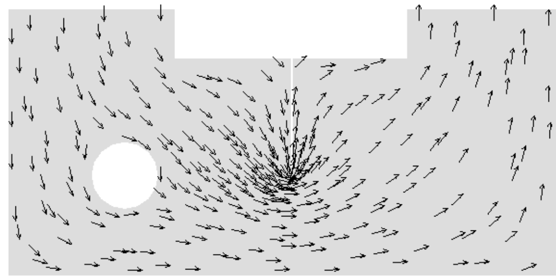
This item is effective only for “Solid arrow in 3-D volume” option. The length of the arrow rendered as 3-D solid is determined on the basis of the predefined thickness factor. The thickness factor is defined by entering its value in the editable text box labeled “Thickness” in the dialog. The entered 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 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

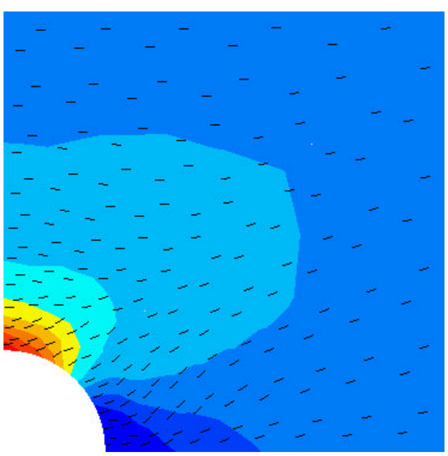


One arrow displayed at every 5 data point

< 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, σ_1 and σ_2, are desired to be displayed together in order to improve the visual understanding of the computed results. In this case, the directions of σ_1 are first drawn, and those of σ_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 condour image of σ_1 is overaid the arrow image of their directions

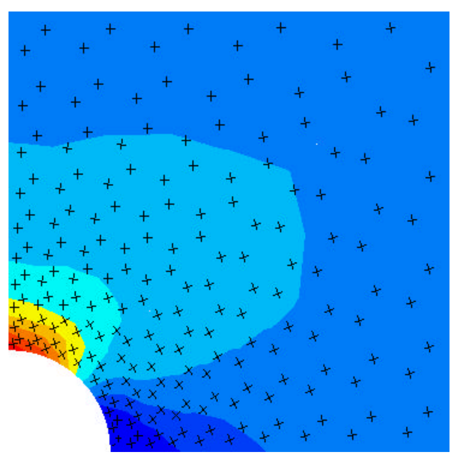
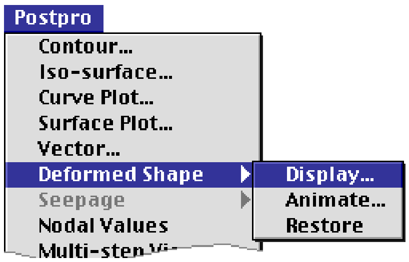


Image of σ_2 direction is added.

< Overlaying vector images >

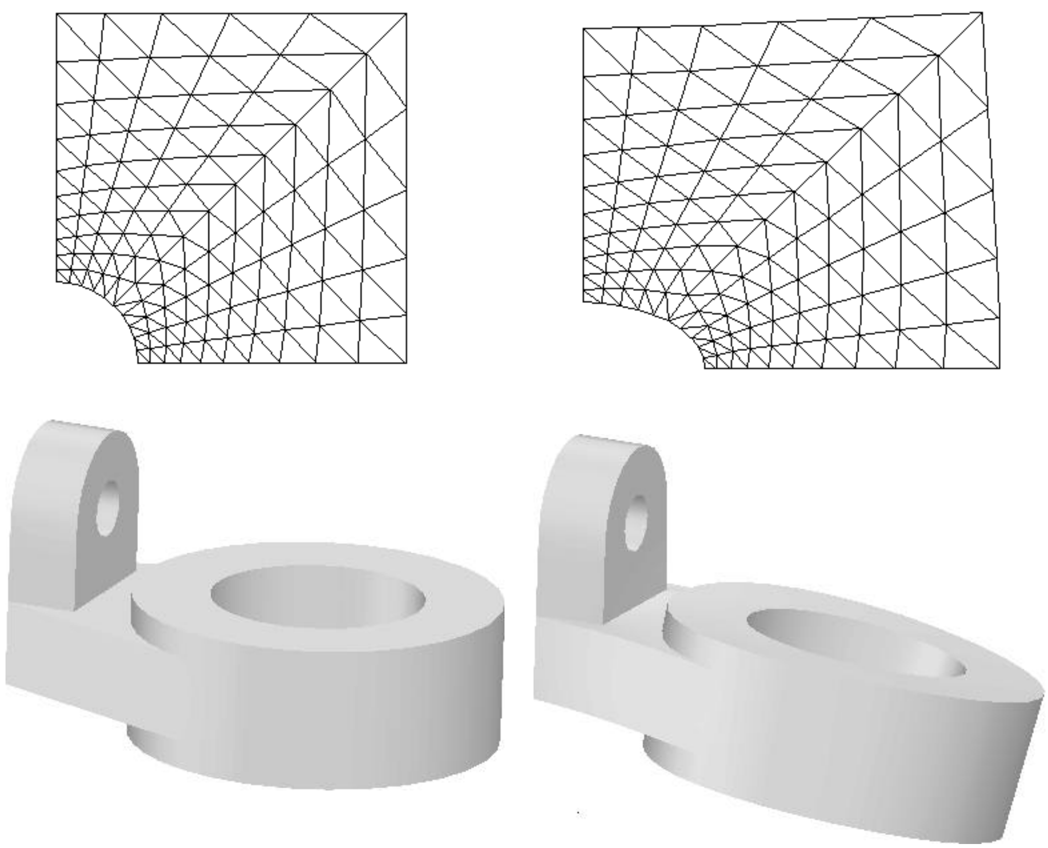
Visualizing displacements by deformed shape and animation



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.

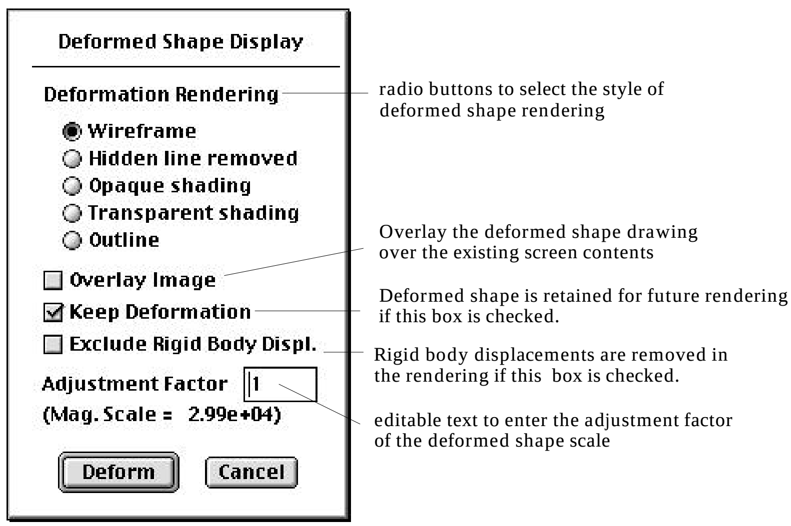


Undeformed shape Deformed shape

< Deformed shape of a planar structural model >

■ Setting the display options

In oder 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.

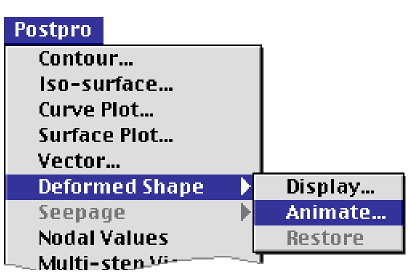
■ Excluding the rigid body displacements from deformed shape

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

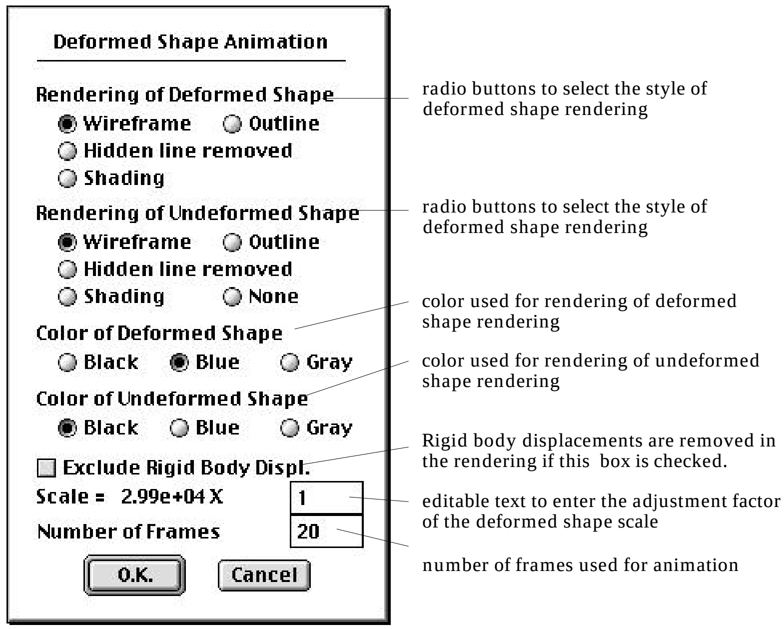
The displacements are usually very small as compared with the overall size of the structural model. Therefore, the displacements should be magnified to such a scale that their effects are discernible in the deformed shape. The deformation scale is the magnification factor that multiplies the displacements 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



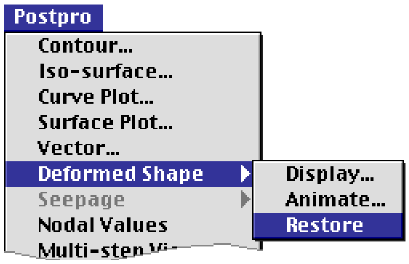
The displacements obtained from the analysis can be visualized by animation. In oder to get a deformed shape, first select "Display..." item from“Deformed Shape” submenu of Postpro menu. Then, “Deformed Shape Display” dialog appears on the screen as shown in the following figure.
The model is animated 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.

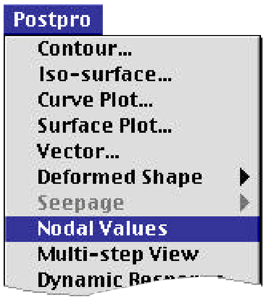


■ Restoring undeformed shape

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.



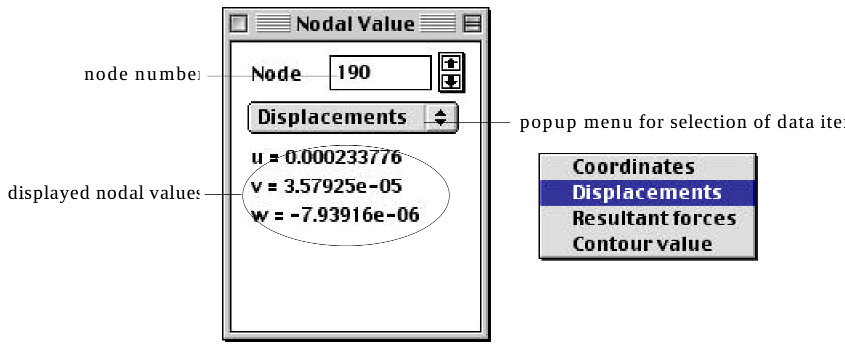
Getting numerical values of nodal data



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

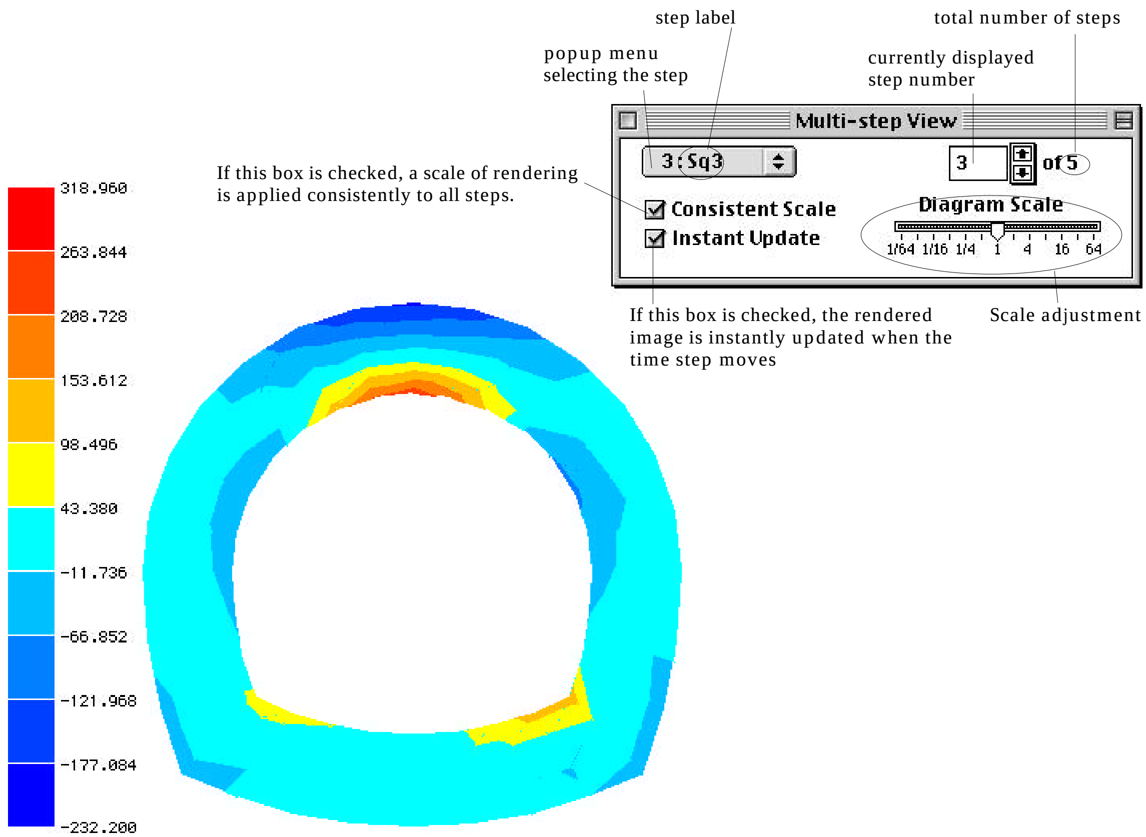
Nonlinear, dynamic,sequentially staged or adaptive solution is obtained by multi-step analysis, while linear static one is done by one-step analysis. Data obtained by multi-step analysis have information for each step of analysis. The data of intermediate steps may or may not have important meaning, 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

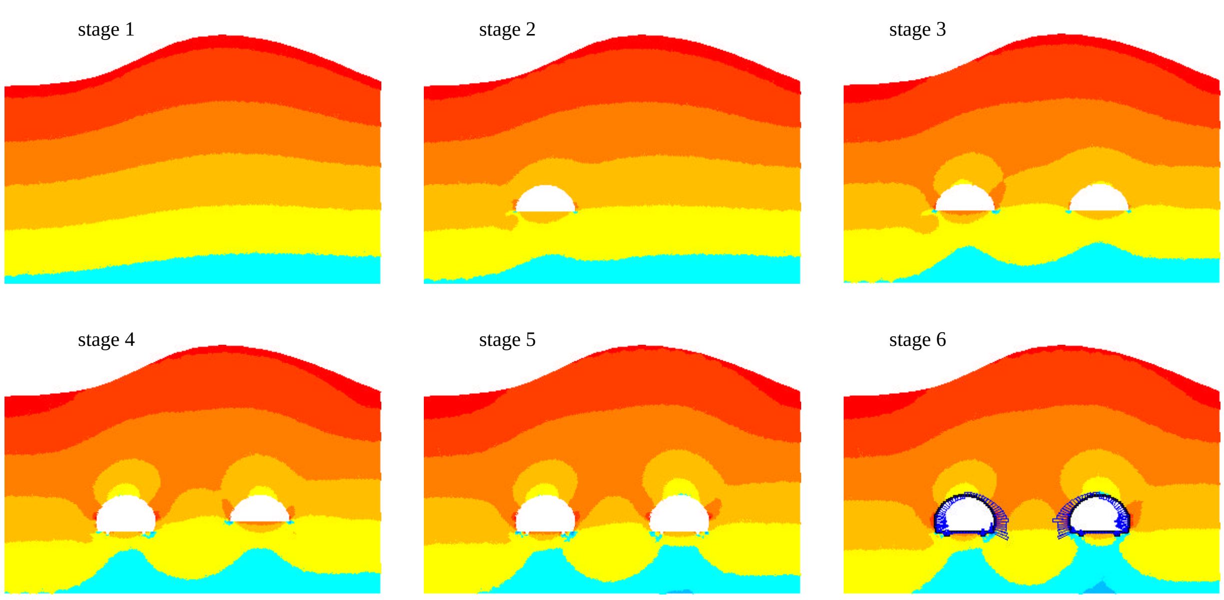
As already described above, various methods of rendering can be employed to represent multi-step analysis results. The method of stepwise rendering coincide with the method applied prior to initiating the stepwise rendering. As an example, if you start stepwise rendering after creating a contour image 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

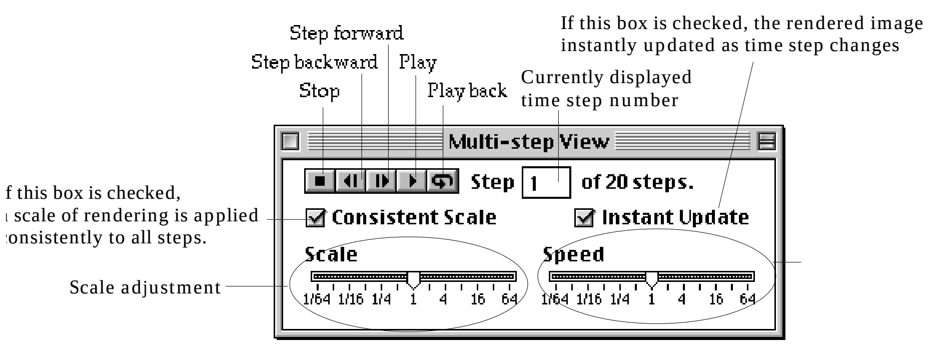
A dynamic analysis produces various data such as displacements, strains and stresses for each of the time steps involved in the solution process. The dynamic responses can be represented as functions of time on the basis of these data. The responses can be expressed using different coordinates, i.e., 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

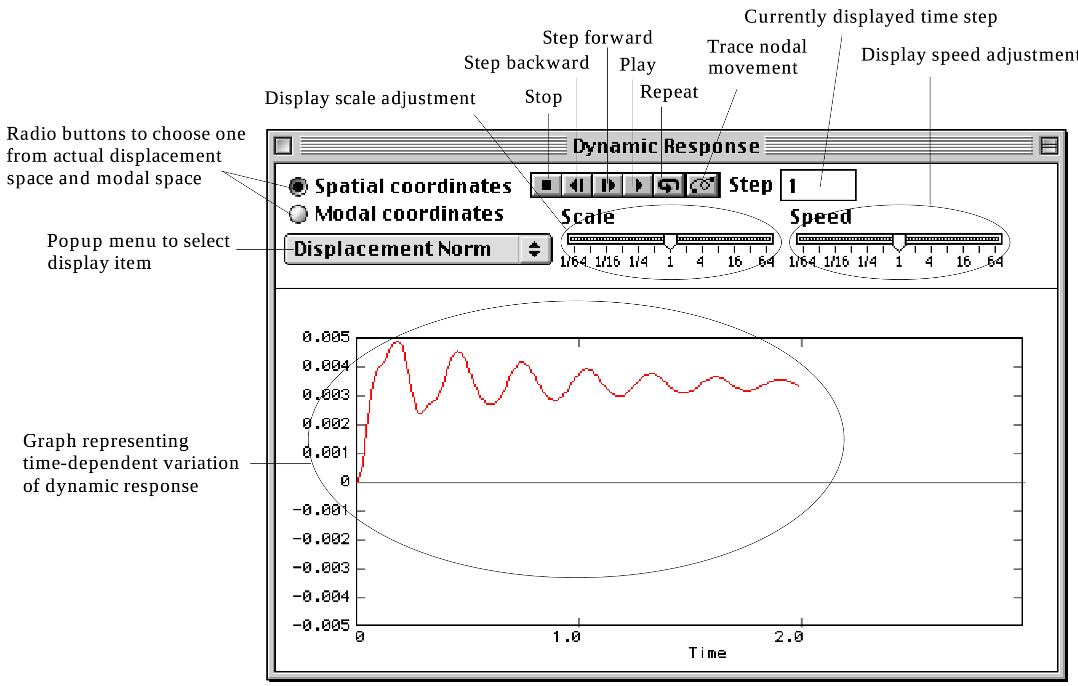
Multi-step view is used in visualizing the dynamic analysis data along the time steps. Select "Multi-step View" item from Postpro menu. Then, "Multi-step View" dialog appears on the screen. Using the control buttons of this dialog, the analysis data can be rendered step by step,



■ 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.

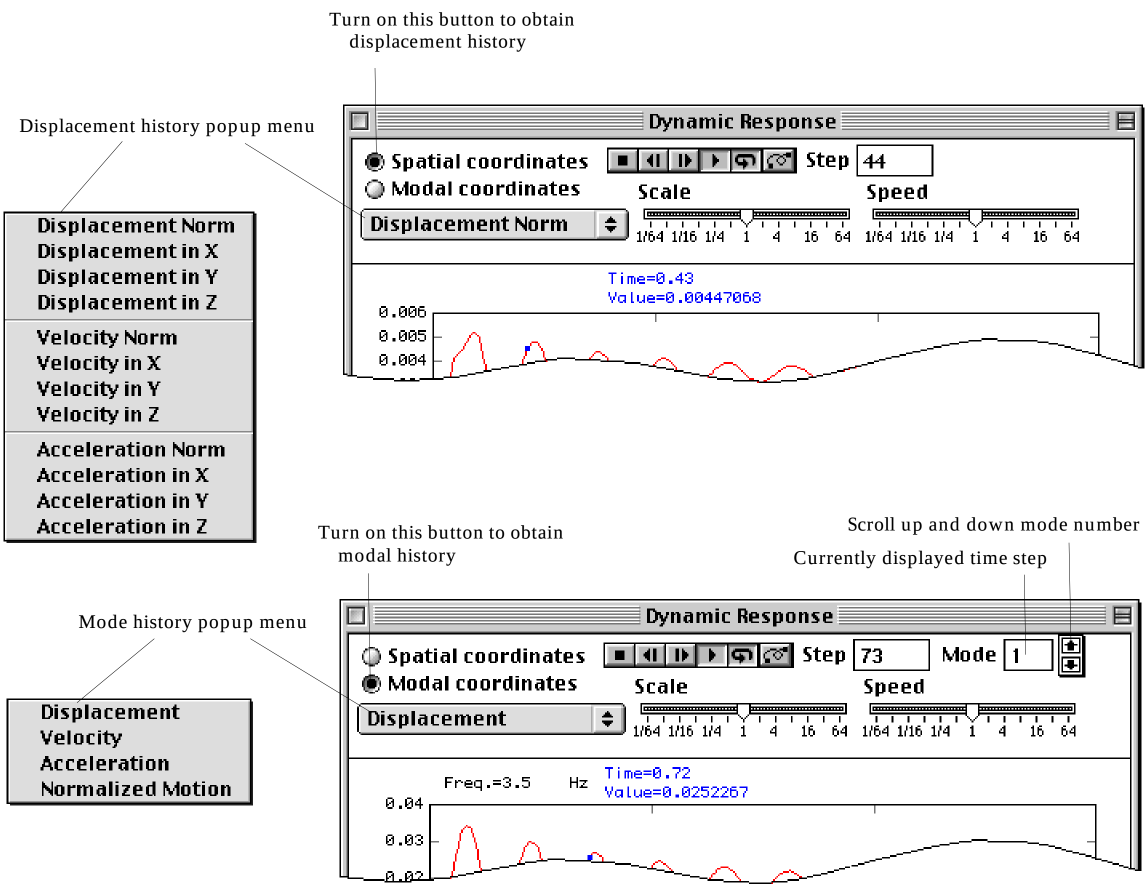


< An example of time history plot >

- 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.



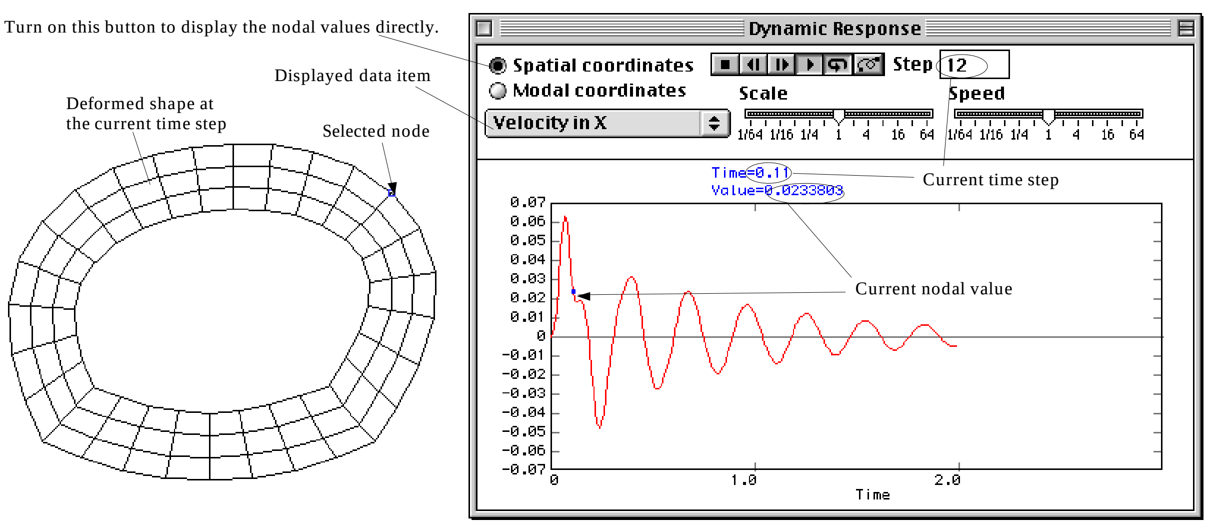
<Selection of coordinates for plotting dynamic response>

■ Plotting time history records of nodal value in spatial coordinates

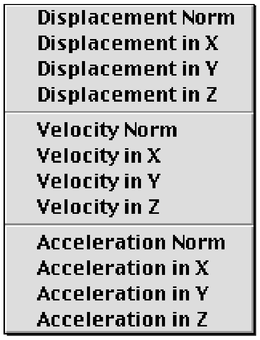
In case "Spatial coordinates" radio button is on, a nodal value changing as a function of time is plotted on "Dynamic Response" window. The nodal value represents one of displacement, velocity, acceleration, and their components which have been recorded for each time step of dynamic analysis.

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.



<Time history plot in spatial coordinates>



- “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.

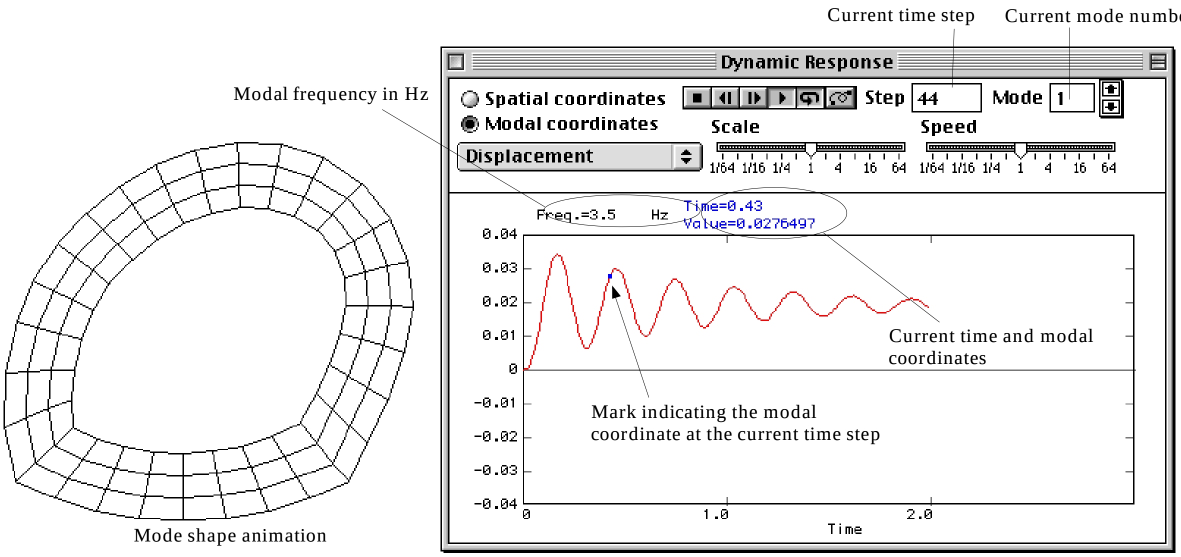
■ Plotting time history records in modal coordinates

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” : 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.



<Dynamic response plot in modal coordinates>

■ 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

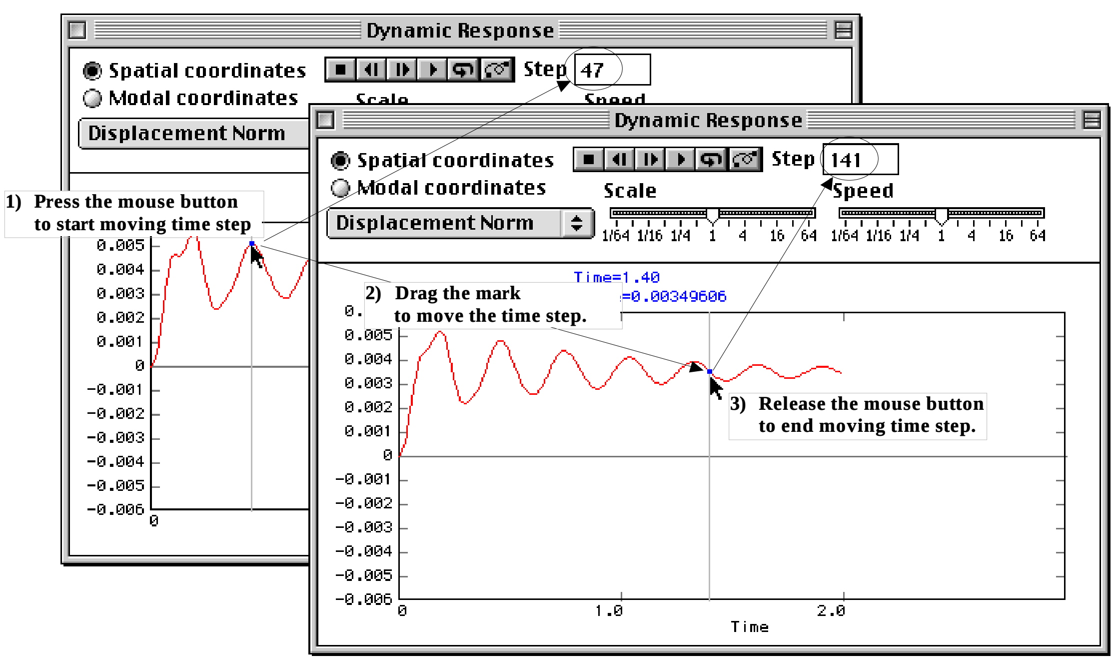
While animation of dynamic motion is going on, time also progresses and the time step number is reflected in the step text box. The time step can also be changed by using step forward button or step backward button .

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.

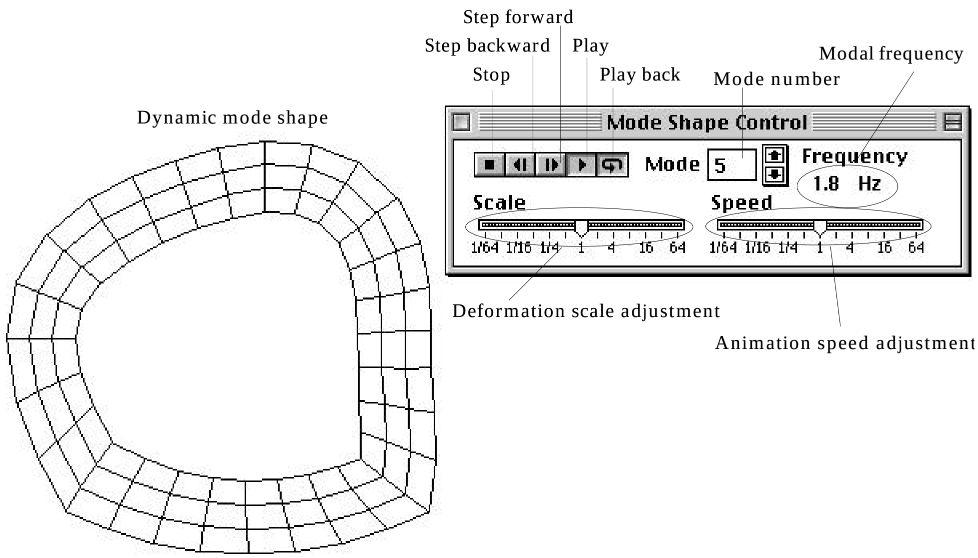
Visualizing dynamic mode shape

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.



<Display of dynamic mode shapes>

■ 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 $(\Delta_1,\ \Delta_2, \ldots,\ \Delta_n)$ are obtained respectively from the corresponding force vectors $(\mathbf{F}_1,\ \mathbf{F}_2, \ldots,\ \mathbf{F}_n)$. Each of the force-displacement relations is expressed with an identical stiffness matrix $\mathbf{K}$.

$$\mathbf{K}\Delta_i = \mathbf{F}_i \qquad (i = 1, \ldots , n)$$

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,

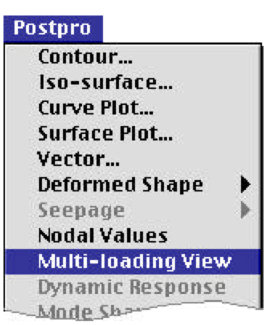
$$\mathbf{F}_c = \sum_{i=1}^{n} c_i \mathbf{F}_i$$

Then the resulting displacement vector can be obtained simply by the same linear combination,

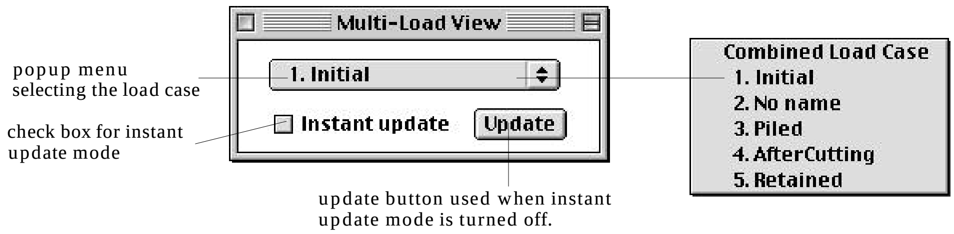
$$\Delta_c = \sum_{i=1}^{n} c_i \Delta_i$$

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.



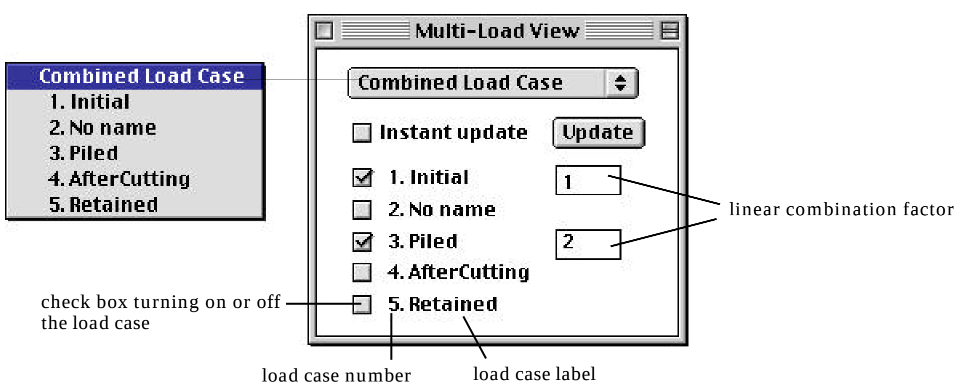
■ Visualizing the analysis results of each load case

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

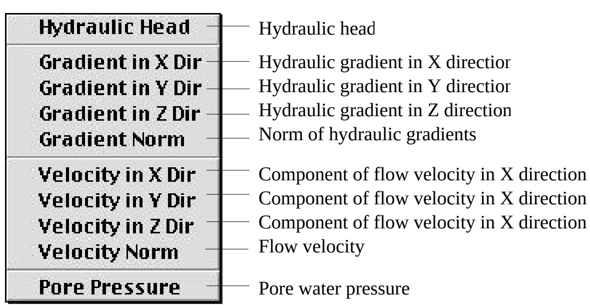
Analysis of a seepage problem gives various results including hydraulic heads, flow vectors, flow velocity, flow path, phreatic surface, and flow discharge. Some of them can be visualized by normal postprocessing functions such as contouring and vector representation. But there are other types of seepage 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

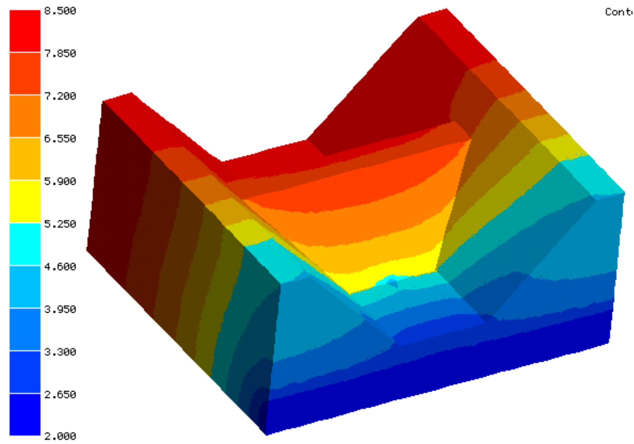
Contouring and vector representation are the most generalized method of visualizing finite element analysis results, and are appropriate for visualizing the scalar or vector data directly obtained from a seepage analysis.

■ 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.



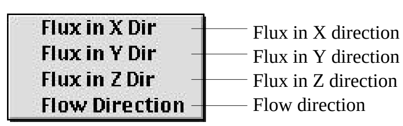
Various methods of applying contouring (for example, displaying contours on cut plane, parallel plane, etc.) are also valid for seepage analysis data. Refer to "Visualizing Scalar Data by Contours" section of this chapter.
Other variant methods of displaying scalar data such as iso-surface representation 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.

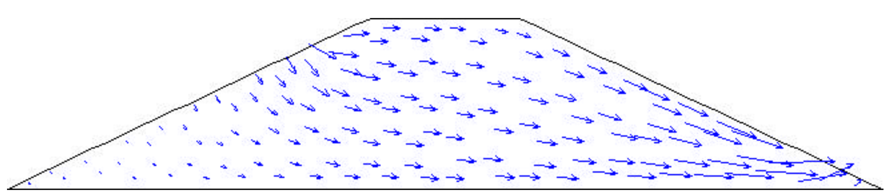


<Contour image of hydraulic head>

■ 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.





<Vector representation of flow velocity and directions>

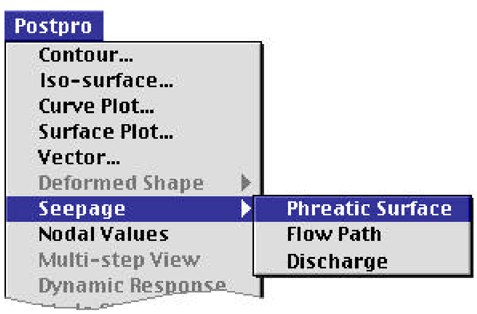
Visualization of phreatic surface

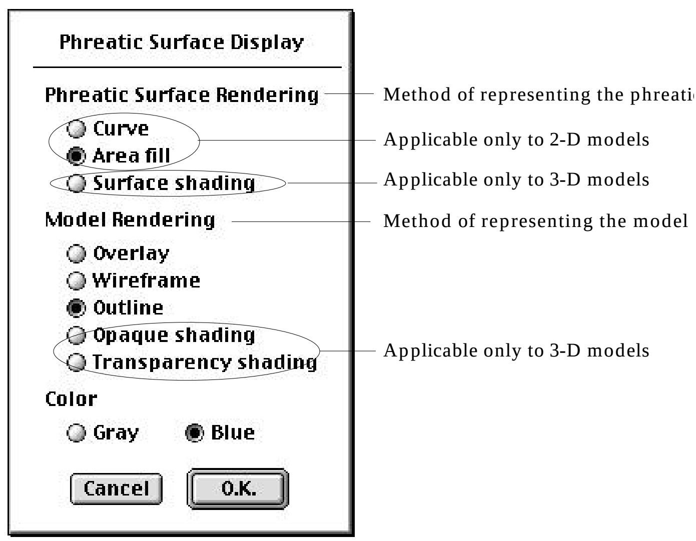
The phreatic surface is one of the most informative and useful data obtained from unconfined seepage analysis. The phreatic surface is represented by a curve in plane and axisymmetric analysis, and by a surface in 3-D seepage analysis.

■ 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.





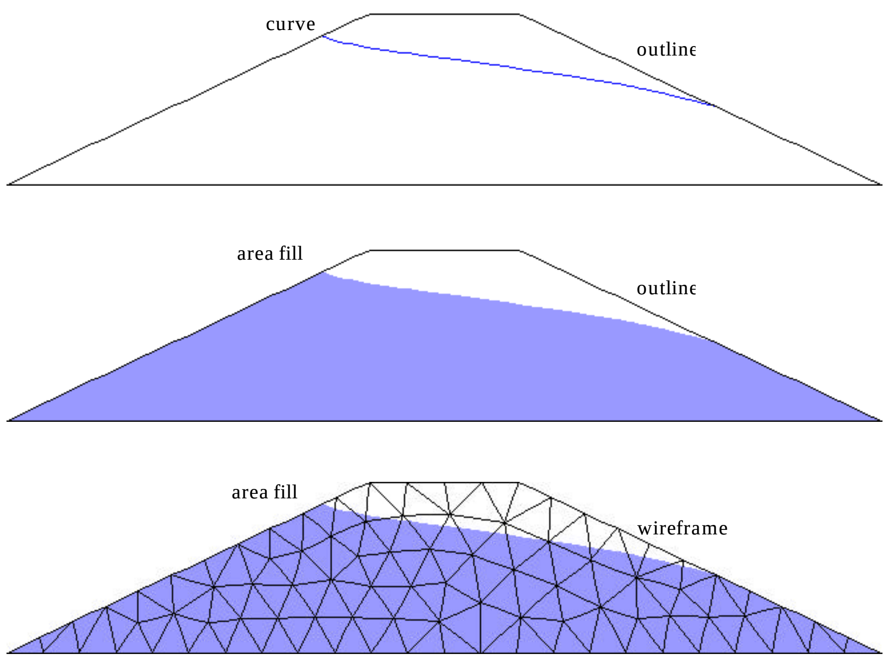
2) Set the "Phreatic Surface Rendering" option.
The phreatic surface can be represented either by line(curve) drawing or by area fill. There are 3 items of this option, but only 2 items, "Curve" and "Area fill" are enabled for plane or axisymmetric seepage analysis. The other one is used only for 3-D 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.

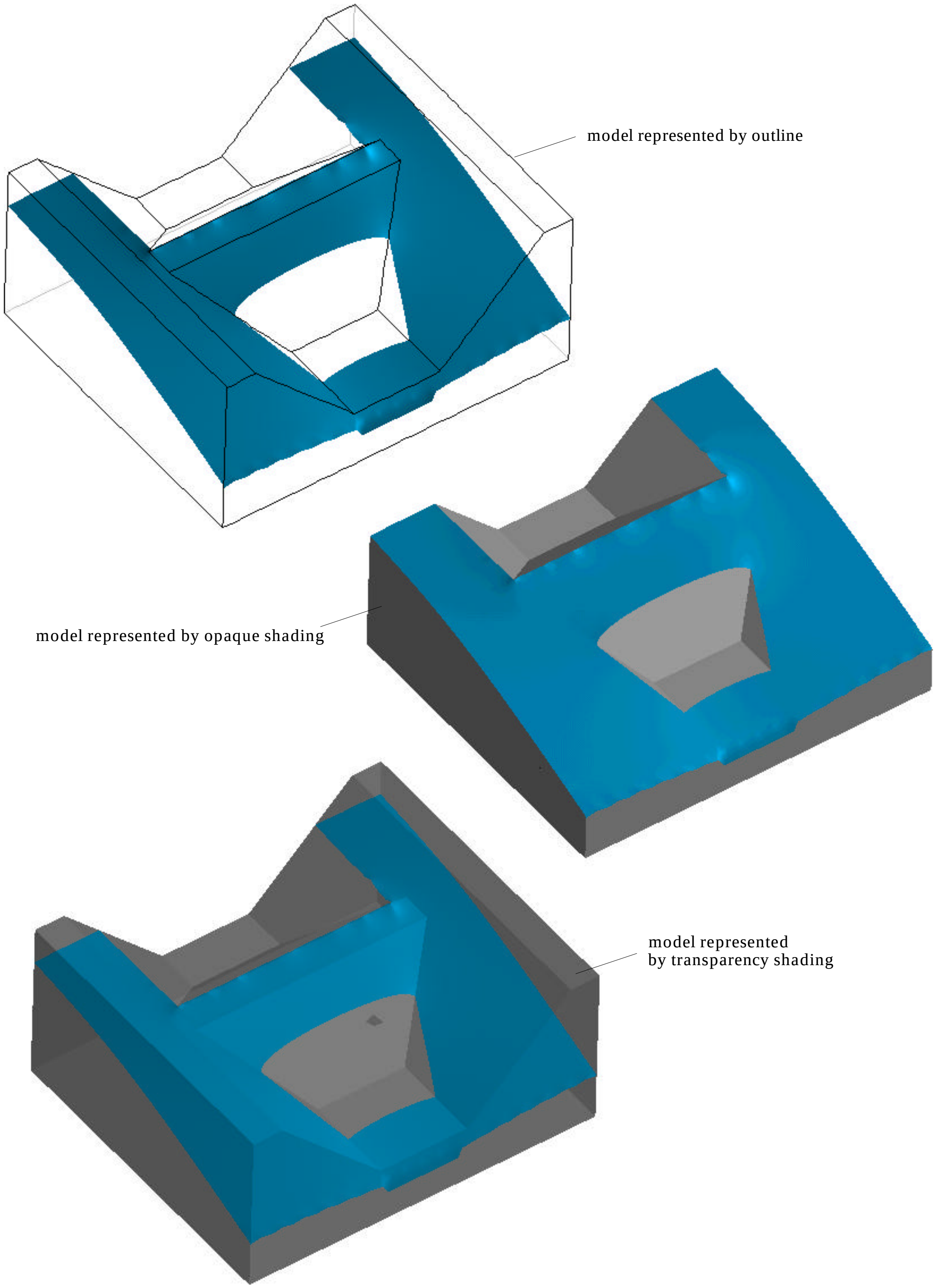
5) Click O.K. button to complete setting options of phreatic surface rendering.
   "Phreatic Surface Display" dialog closes, and the phreatic surface is rendered in accordance with the option setting.



<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 >

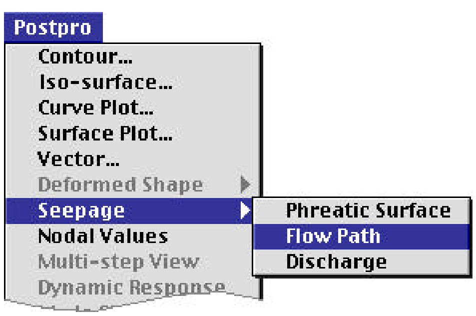
Visualization of flow path

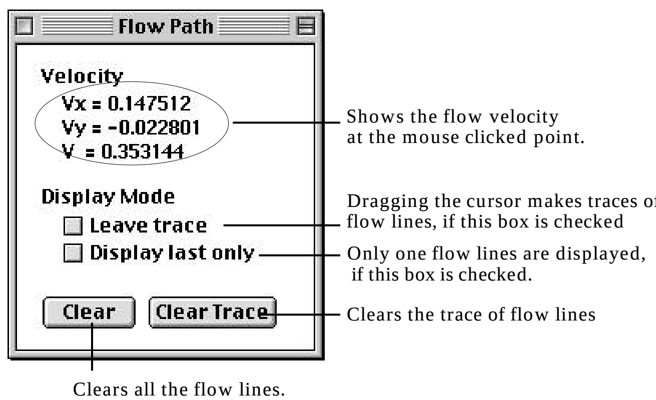
A flow path represent the route of water flow, and is obtained by tracing the velocity vector. A flow path is very close to but not identical with the flow line which can be obtained from a seepage analysis with revered boundary conditions on the basis of dual relation between hydraulic head and flux.

■ Interactively displaying a flow path

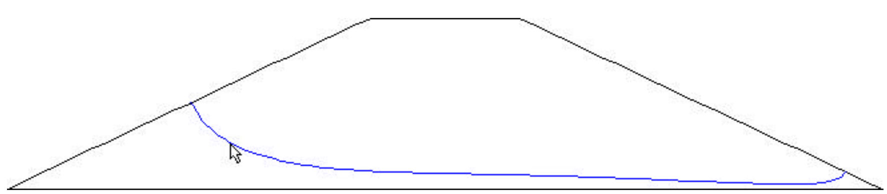
The phreatic surface is represented by a curve in plane or axisymmetric analysis. There are a number of options for rendering the phreatic surface which can be set by the following procedure.

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.



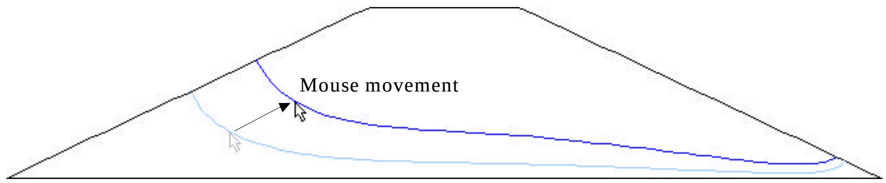


2) Select "Flow Path" item from "Seepage" submenu of **Postpro** menu.
   "Flow Path" dialog opens. The flow paths and related information can be displayed interactively using dialog and mouse clicks.
3) Click a point within the model.
   Then, the flow path passing through the point is drawn, and the flow velocity at that point is displayed on the dialog.



<Drawing flow path by clicking a point within the model>

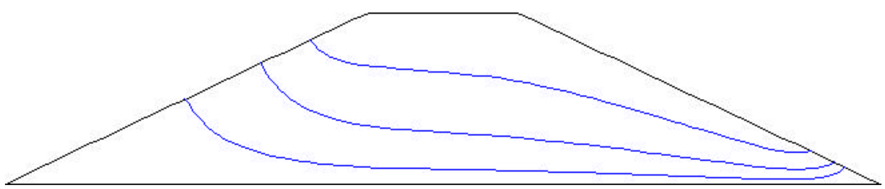
4) Move the mouse with mouse button pressed to drag the flow path.
   The flow path is dragged following the cursor point, while moving the mouse with mouse button pressed.



< Dragging the flow path >

■ Displaying multiple flow paths

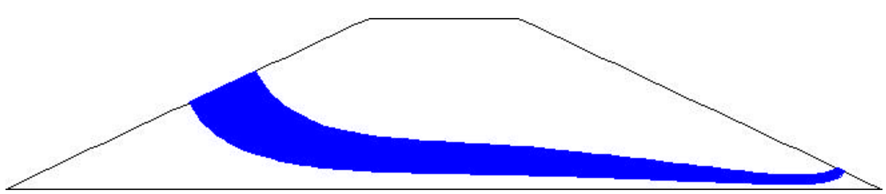
A new flow path is drawn as another point is clicked. If "Display last only" item of "Flow Path" dialog is checked, only one flow line is drawn. Otherwise, all the flow lines at the clicked points are displayed together.
Uncheck "Display last only" box, and click points within the model. The flow paths 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.

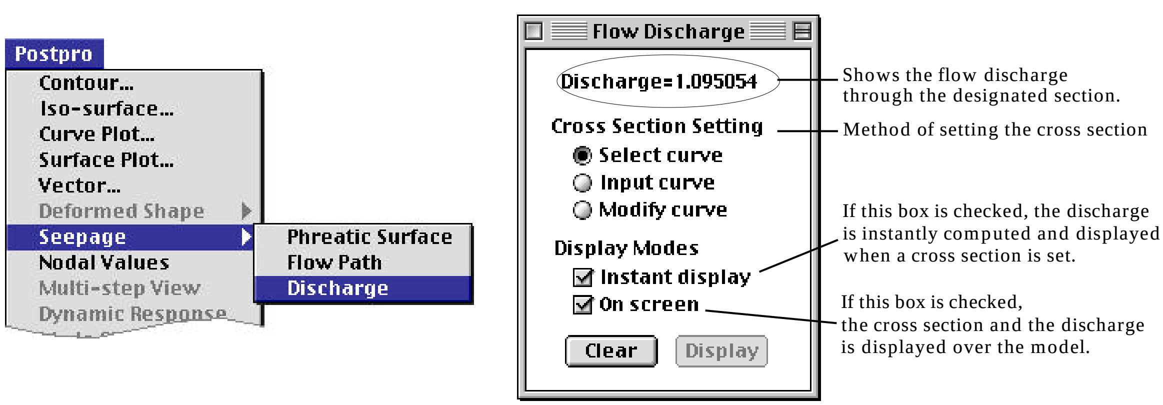
Visualization of flow discharge

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.

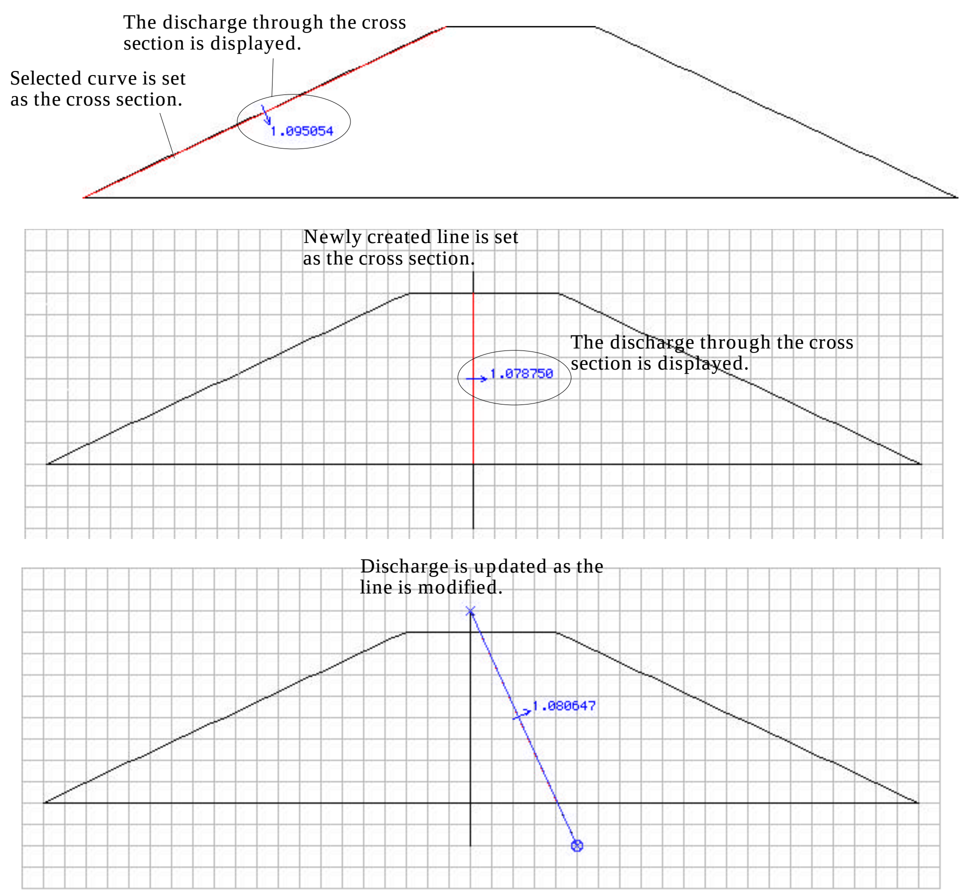
3) Choose the display mode.
   - “Instant display” : If this box is checked, the discharge is instantly computer, and displayed as soon as the cross section is designated.
   - “On screen” : If this option is on, the discharge is displayed over the cross section image on the main window.

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.

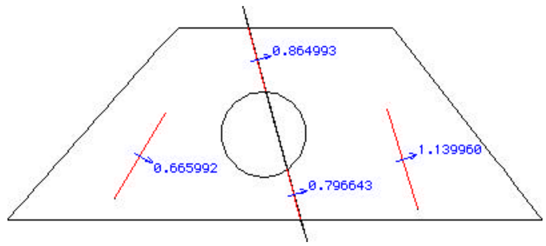
   If "Instant display" mode is on, this step is not necessary. Display button is enabled only when "Instant display" mode is on.



< 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.

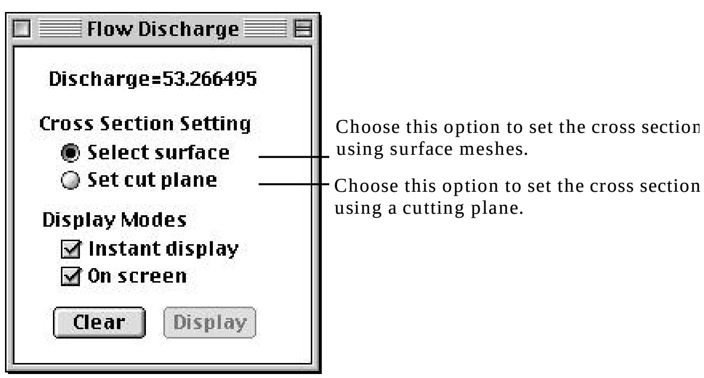


< Setting multiple cross sections for display of discharge >

■ 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.



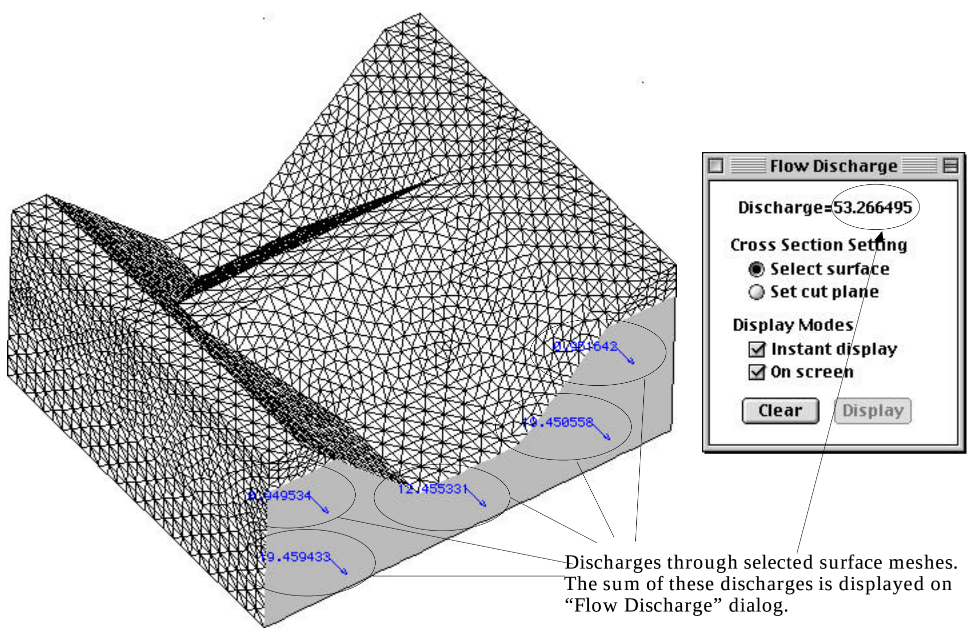
2) Choose the methods of setting the cross section in "Flow Discharge" dialog.
   - "Select surface" : The flow section is designate by selecting an existing surface mesh. The surface mesh should be part of the analysis model.
   - "Set cut plane" : The cross section is defined by the cut plane which 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).

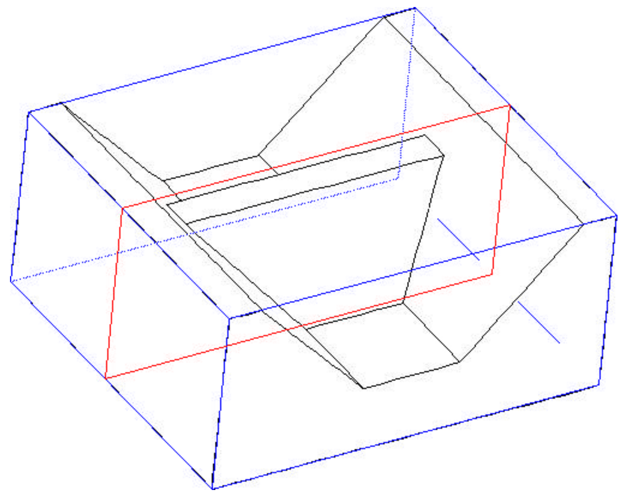
If this option is chosen, the cut plane setting mode is automatically activated, and "Instant display" mode is turned off. Instead, **Display** button is enabled. Set the cross section at the desired position by the cut plane. Refer to "Setting a cut plane for contouring" in this chapter.

6) Click **Display** button to display the discharge.

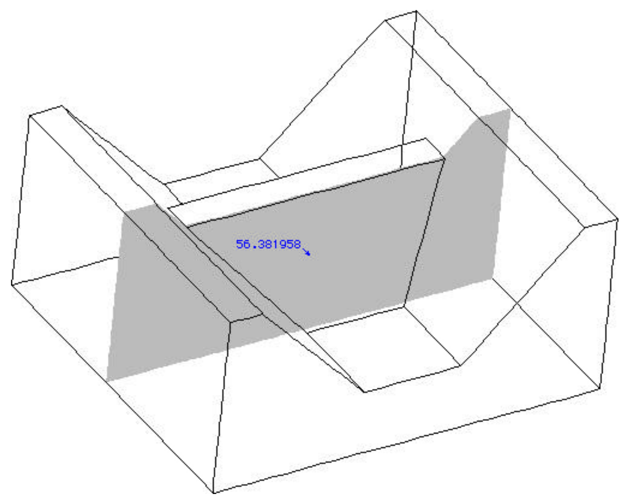
If "Instant display" mode is on, this step is not necessary. **Display** button is enabled only when "Instant display" mode is on.

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

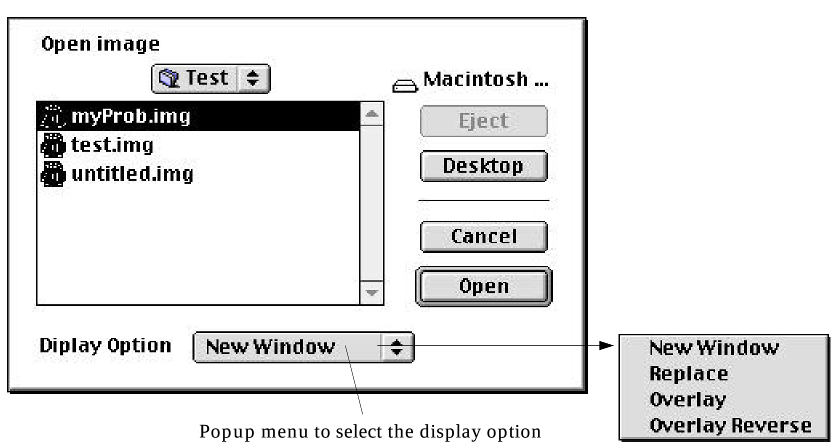
Postprocessing of continuum analysis creates various kinds of images visualizing the analysis results. The visualization can be recorded in the form of still images or animations. Images can be saved in files, and retrieved later. They can also be captured directly to the clipboard. Animations are created 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

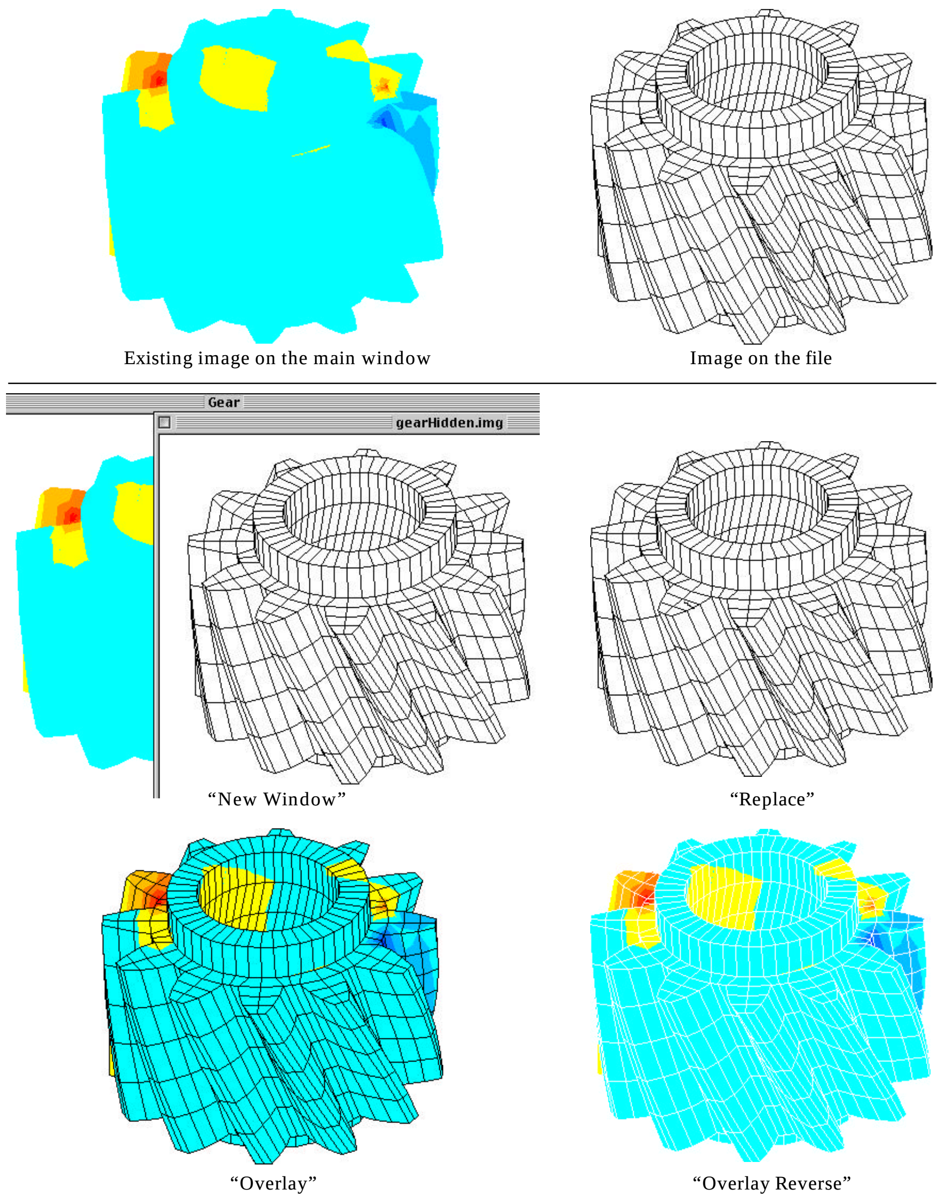
The image on the main window can be saved in a file, and can be read and displayed later. In order to retrieve an image, choose “Open Image...” item from Postpro menu. Then, the file opening dialog appears. The browser of the dialog shows only valid image files. Locate the desired image file using the 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 >

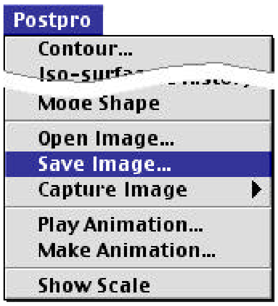
- “New Window”: New window is opened, and the image read from the file is displayed on this window. The image on the main window remain intact.

- “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.

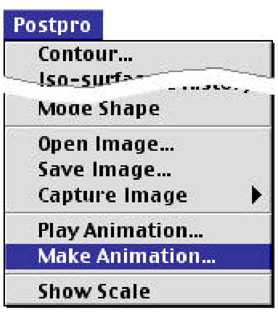
Animation

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 xy, σ_{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.

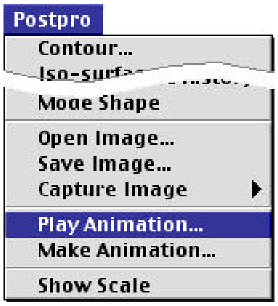
If 2 or more commands are in conflict with each other, the last one overrides the preceding ones. The use of the script commands is exemplified in the following, and their meanings are explained below.

```
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.

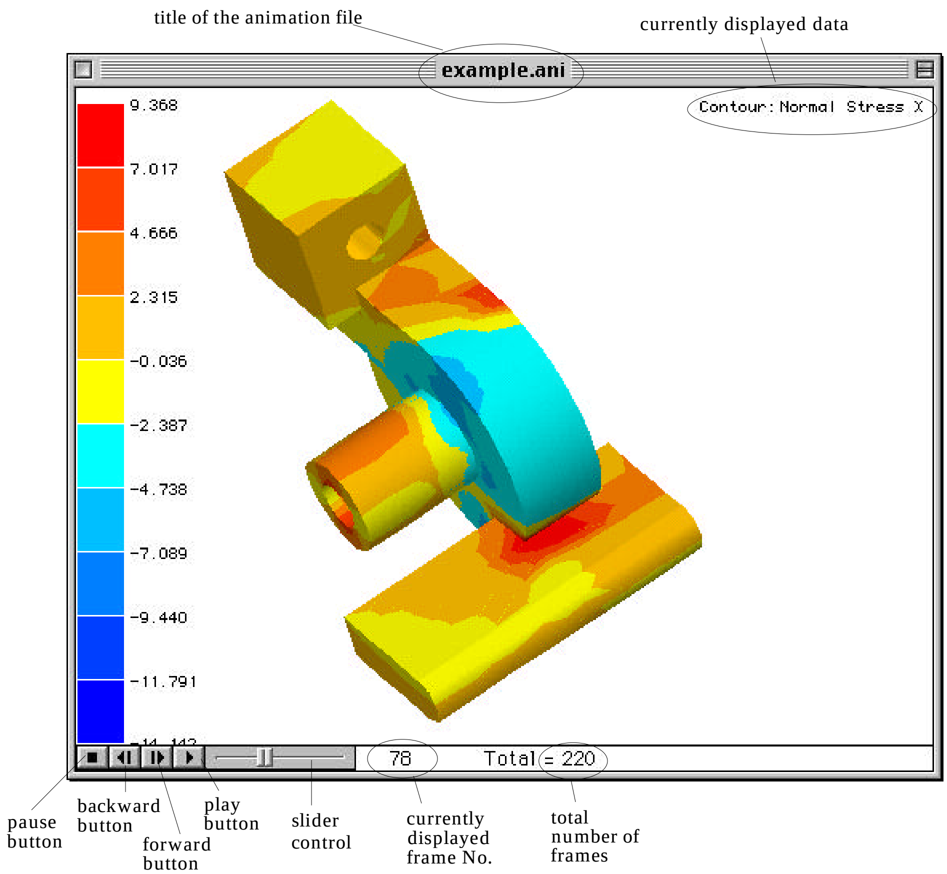
- The total number of frames in the animation is 220, because it is the maximum value of *lastFrame* appearing in the script.
- The data set designated by the popup menu item 3 is applied to frame 1 to 80.
- The data designated by the popup menu item 6 is applied to frame 81 to 200.
- Deformed shape 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 to0.0. 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



One 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 form 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 appears 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 >