

Chapter 5

Data Assignment

Chapter 5 *Data Assignment*

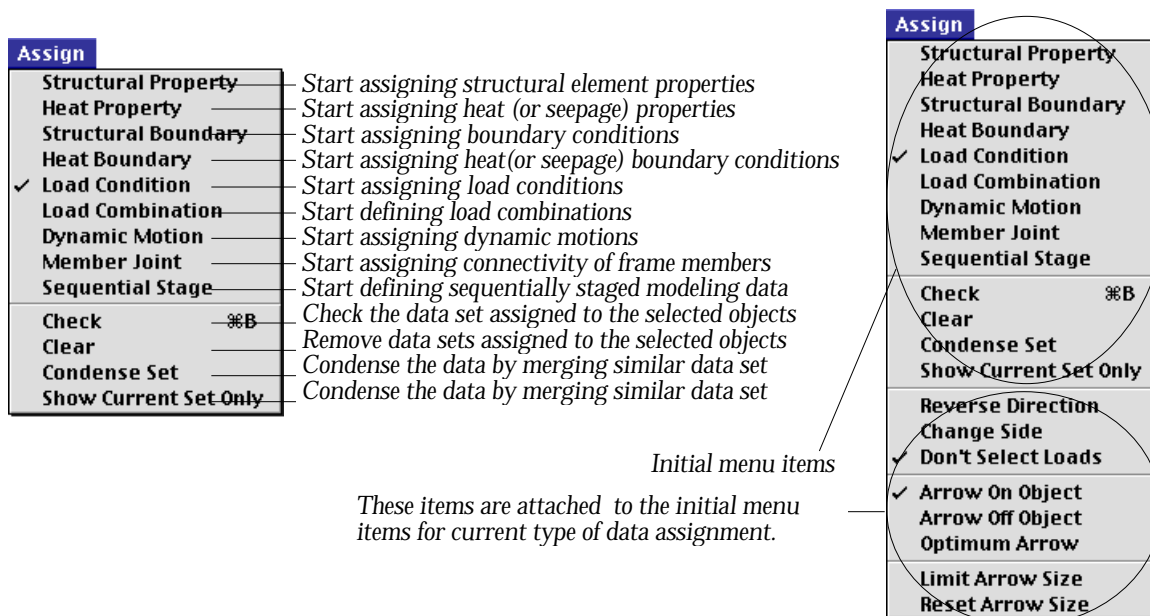
Chapter 5 Data Assignment

A problem to be solved by the finite element method is defined by the geometry and the attributes of the solution domain. The geometry is modeled by nodes and elements created by mesh generation as described in Chapter 4. The attributes consist of various data such as material properties, boundary conditions, load conditions and so on. VisualFEA provides functions for creating these data and assigning them on the nodes and elements. The attribute data handled by VisualFEA can be classified basically into 3 categories: properties of the solution domain, conditions at the boundaries, and data related to external effects such as load and temperature.

The data in each category have many items varying with the type of analysis and the geometry of the domain. Data sets are first defined with specific values of all relevant items, and then assigned to the selected nodes or elements. These actions are carried out interactively using various functions provided in the tool palette, menu or dialog box. The states of the data assignment can also be visualized and checked for their correctness. The major functions related with data assignment can be summarized as follows:

- creating and deleting data sets : define a new data set and enter values of the individual items of the data set using a dialog box, and delete a data set.
- assigning and unassigning data : assign a data set to the selected objects, and unassign a data set from the selected objects.
- checking and visualizing data assignment : visualize the assignment of a specific data set or identify the data set assigned to the selected objects.

There are also functions specific to certain data types, as shown in the following **Assign** menu.



Overview of Data Assignment

There are a few different types of data to be assigned to the finite element model. But the procedures of assigning data are similar and consistent regardless of the data type as described below.

Basic composition of data

The minimum amount of data necessary for finite element analysis is varied with the type of problem to be solved. VisualFEA can do 3 types of analysis : structural, heat conduction and seepage. And therefore, data required for these 3 types of problems are described below.

■ Data for structural analysis

The attribute data necessary for structural analysis are basically composed of the following components:

- material properties: elastic constants, unit weight, thermal expansion coefficients, etc.
- structural boundary conditions: structural constraints or supporting states such as spring.
- load conditions: various types of applied forces including uniform force, moment, body force, etc.

The above components are essential for structural analysis. Multiple load conditions can also be defined by linear combination of the load condition sets.

- load combinations: combinations of load condition sets defining multiple load condition. This item is available only for linear static structural model with more than 2 load condition sets.

The following data items are additionally required to define other characteristics of the problem.

- local member connectivity: pin ended, rigid ended.
- dynamic motion: dynamic displacement, velocity and acceleration.

When the necessary data are completely assigned, the problem are are ready for finite element solution.

■ Data for analysis of heat conduction

The data necessary for heat conduction analysis are basically composed of the following 2 components:

- material properties: thermal conductivities in X, Y and Z directions.

- heat and temperature boundary conditions: specified temperatures, specified heat flux, convection boundary conditions, heat source, etc.

In order to make the problem solvable by VisualFEA, the material properties should be assigned to all elements, and at least one component of the heat or temperature boundary conditions should be applied.

■ Data for seepage analysis

The data necessary for seepage analysis are basically composed of the following 2 components:

- material properties: flow conductivities in X, Y and Z directions, conductivity function, water content function.
- seepage boundary conditions: water head (open and confined), flux, point source, and initial water table.

In order to make the problem solvable by VisualFEA, the material properties should be assigned to all elements, and an open or confined head boundary condition should be applied at least to one point of the model, as subsequently described

General procedure of data assignment

The procedure of assigning data, regardless of type, may be divided into the following several steps:

- 1) Start the data assignment procedure.

Choose one of the items “Element Property”, “Structural Boundary”, “Heat Boundary”, “Load Condition”, and “Element Connectivity” from **Assign** menu. Then, a corresponding dialog box appears.

- 2) Define a data set.

A data set is a data unit carrying specific values of data items. There are a number of items in a data set. Specific values are entered for them using the associated dialog. Details of data items are described for each data type in the following sections.

You may define as many data sets as necessary. There is no limit to the number of data sets. Only one data set is active at a time. The currently active data set is effective for assignment. When a new data set is created, the set becomes active automatically.

- 3) Make the desired set active.

If you want to assign a data set which is not currently active, you should make the desired set become the active one by scrolling. A data set may be created, deleted or scrolled using the functions of dialog boxes, which are

explained later in this section.

4) Select objects to assign data.

A data set may be assigned to different types of objects, but not to all object types. There is a default object type for a given data type. The selection tool of default object type is automatically activated when you start a data assignment procedure. You may switch the object selection tool if necessary, but only applicable object selection tools are enabled.

5) Assign data to the selected objects.

Click **Assign** button of the dialog to assign the currently active data set to the selected objects. The newly assigned data are indicated with highlight on the mesh.

Some types of data may be assigned to surface meshes, volume meshes or curves. However, they are eventually assigned to the nodes or elements within these objects. The default and applicable object types are described in detail for each data type in the following sections.

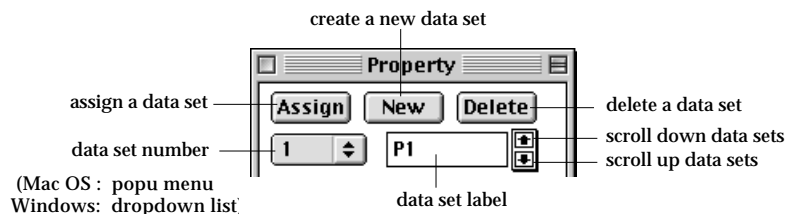
You may repeat step 2) through 5) as many times as necessary without opening the dialog again while the corresponding dialog box is on the main window. You may quit this data assignment procedure by clicking the close box of the dialog, or starting any other procedure.

Functions common to all types of data assignment

There are a number of functions related to data assignment. Some of them are provided as dialog items, and others as menu items. Among all the various functions related to data assignment, those ones commonly applied to all types of data assignment are described below. Other functions are applied differently depending on the type of data and the analysis subject. So, they are explained in separate sections for various types of data assignment.

■ Functions handling data sets

While you are assigning data, a dialog box is always displayed on the main window. The dialog boxes have various items depending on the associated data type and the analysis subject. However, there are items common to all dialog boxes for data assignment as shown below.



All of the above dialog items are for handling data sets including creating, deleting, scrolling and selecting data set. Usage of the items is explained below.



Click this button to create a new data set. The values of the data items in the new set are initially copied from the existing values if applicable, or given with the default values.



The active data set is deleted by clicking this button. The data set is also removed from all the objects assigned with it. Exceptionally in case only one data set is left. Then the set is not deleted, but only its assignment is removed.

The data set numbers are rearranged to fill the gap of the deleted set. The set next to the deleted set becomes active.



Click this button to assign the current data set to the selected objects. (Windows: It is the default push button of the dialog. Therefore, pressing key has the same effect as clicking this button.)



The number of the data set is displayed as a popup menu (Windows: drop-down list) item. You may choose the desired set using this popup menu (Windows: drop-down list).



The label of the data set. You may label a data set by entering a character string in the text box.



This button is used to scroll up the data set. The current set is scrolled up by this button. Clicking this button once reduces the current set number by one.



This button is used to scroll down the data set. The current set is scrolled down by this button. Clicking this button once increases the current set number by one.

■ Entering values of data items

A data set consists of many data items, each of which should be given with specific values. All of the data items are expressed as dialog items: some are editable texts, and others are radio buttons, check boxes, etc.

The first data set is initialized with the default values given by the program. When a new data set is created, each item of the set usually inherits the value of the corresponding item of the previously current set. Some items are interrelated.

You may freely enter or change values of the currently current data set before assigning the set to objects. However, once a data set is assigned to any object, modification of the set is restricted.

■ Modifying values of data items

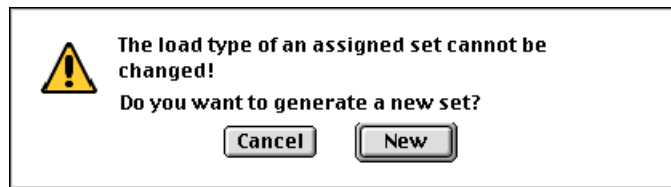
In order to modify the values of a data set, first you must make the set the current set using the dialog items , , or as explained above. Only the current

set is displayed on the dialog, and thus, their items can be accessed for modification. If the current set is not yet assigned to any object, the data items can be edited freely. On the other hand, if the set is already assigned, modification of the set either asks for confirmation or is not allowed. So, if you try to modify an item of an assigned data set, you will get a message like this



If you click **Modify** button, the current data become modifiable again so that you may freely edit any item in the set until the modified set is assigned to any object. In this case, the modification will affect all the objects assigned with this data set. If you click **New** button, the current set remains intact, and instead, a new set is created. If you click **Cancel** button, your editing action is ignored, and nothing happens.

There are also cases in which any assigned set cannot not be modified. Then, you will get a notice like this,

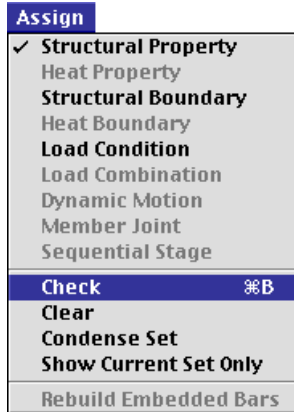


If you click **New** button, a new set is created and becomes current. So, you may edit the data items of the new set and assign the set to the selected objects. If you click **Cancel** button, your editing action is ignored, and nothing happens.

■ Assigning data sets

If new data assignment is ready, **Assign** button of the dialog is enabled. Pressing the button assigns the current data set to the selected objects. Data assignment can be canceled by “Undo” command in **Edit** menu before any other action is taken. You may assign a data set to objects which have already been assigned with other data sets. The results of such overlapping assignment depend on the type of data and setting of the related option, and are explained in conjunction with assignment of corresponding data sets.

■ Checking data assignment



The state of the data assignment can be checked using “Check” item. When you choose “check” item in **Assign** menu, the data set assigned on the selected object becomes active, and the dialog shows the data items of this set. At the same time, the assignment of the new current set is displayed with highlight.

If two or more objects are selected and they are not assigned with a single data set, the new current set cannot be determined, and thus requested checking may not be realized.

Instead of using this function, you may identify the data set by scrolling the current data set.

■ Clearing data assignment

In order to clear data assignment from certain objects, first select the objects to be cleared of data assignment, and choose “Clear” item. All data sets of the current dialog are unassigned from the selected objects. The data sets themselves are not affected by this action.

In order to delete data sets, use **Delete** button in the dialog.

■ Condensing data sets

If you want to remove data sets which are not assigned to any object, choose “Condense” item. As for structural boundary conditions, data sets with identical contents will also be merged into one by this action. If the current set is merged into another set, the merged set becomes the current set.

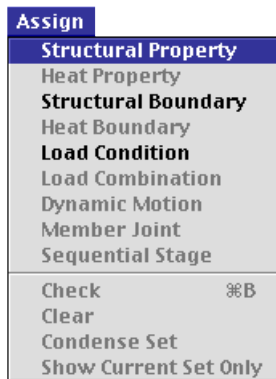
■ Ending data assignment

In order to end the current data assignment, simply click the close box of the dialog box, or start any other command.

Structural Element Properties

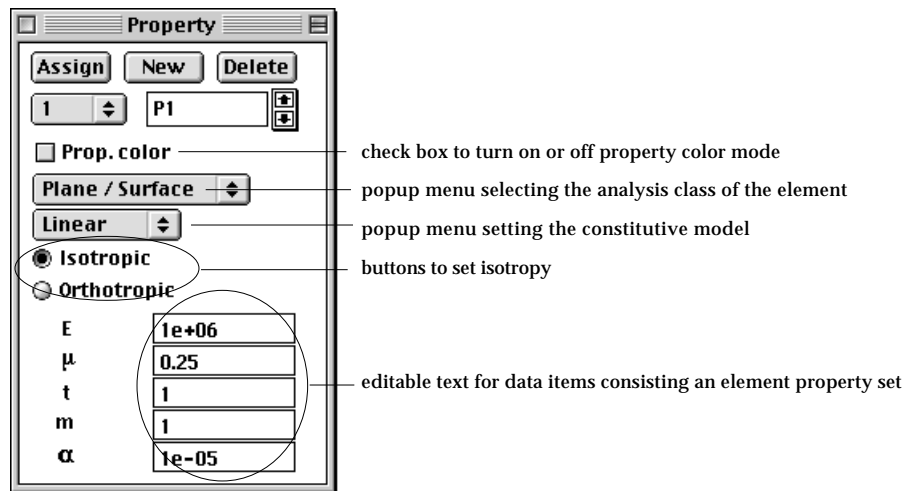
The attributes of the solution domain in finite element analysis are defined by the data representing the characteristics of each element. These data are termed here as element properties. The element property related action is initiated by one of the 2 different menu items, i.e., "Structural Property" and "Heat Property" (or "Seepage Property" for a seepage analysis) depending on the type of property for assignment. In the case of coupled analysis, it is necessary to assign two different types of properties to one model.

Defining structural element properties

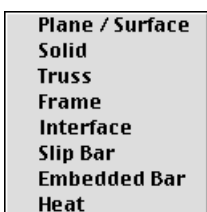


In order to start assigning structural properties, choose "Structural Property" item in **Assign** menu. "Property" dialog appears, and the current state of their assignment are displayed in the main window.

The element properties are defined and assigned by the data unit called element property set. A set consists of many data items, all of which are displayed on "Property" dialog. Structural element properties may include geometric characteristics as well as material properties. The items differ depending on the subject of analysis, or analysis class of the element as described below.



■ Analysis class of element



The first popup menu in "Property" dialog is provided to enable mixing different types of structures in one analysis, as explained in the next section. Using this menu, you can select the type of analysis-related characteristics to impose on the element. It is termed here as "analysis class of element." Each item of the popup menu represents an analysis class of an element.

- "Plane/Surface": plane stress, plane strain, axisymmetric, plate bending, and shell element.
- "Solid": 3-d solid element.
- "Truss": 2-d or 3-d truss element.
- "Frame": 2-d or 3-d frame element.
- "Interface": interface or gap element
- "Slip Bar": slip bar element
- "Embedded bar": embedded bar element
- "Heat": heat conduction element

Only classes compatible for the current project are selectable. In case of frame analysis, for example, "Truss" and "Frame" items are selectable, and others are disabled.

■ Constitutive model

The second popup menu in "Property" dialog is to select the constitutive model. This is applicable for material nonlinear analysis of structures. For linear analysis, the menu contains only "Linear" item. The current version of VisualFEA supports only those nonlinearities shown as the menu items. The items vary depending on the analysis class of element as described in the previous section. For "Plane/Surface" and "Solid", the following items are available.

Linear
Elasto-plastic:V-M
Elasto-plastic:M-C
Elasto-plastic:Tresca
Elasto-plastic:D-P
Compression Only
Tension Only

- "Linear": linear elastic model.
- "Elasto-plastic:V-M": Elasto-plastic model with Von Mises yield criterion.
- "Elasto-plastic:M-C": Elasto-plastic model with Mohr-Coulomb yield criterion.
- "Elasto-plastic:Tresca": Elasto-plastic model with Tresca yield criterion.
- "Elasto-plastic:D-P": Elasto-plastic model with Drucker-Prager yield criterion.
- "Compression Only": Linear constitutive relationship for compression, and no stress for tension.
- "Tension Only": Linear constitutive relationship for tension, and no stress for compression.

The following are the items available for interface elements.

Linear Interface
No Tension Slip
No Compression Slip
Gap

- "Linear Interface": linear elastic properties defined in the longitudinal and the thickness direction respectively.
- "No tension slip": The interface delivers compressive normal force across the element, but not tensile force. The maximum resistance against slippage between the two faces across the element is defined by the friction coefficient. The maximum resisting stress is obtained by normal stress multiplied by the friction coefficient.

- "No compression slip": The interface delivers tensile normal force across the element, but not compressive force. The maximum resistance against slippage between the two faces across the element is defined by friction coefficient. The maximum resisting stress is obtained by normal stress multiplied by the friction coefficient.
- "Gap": The interface models a gap between two faces. No force is delivered between the two faces until the gap is diminished by deformation.

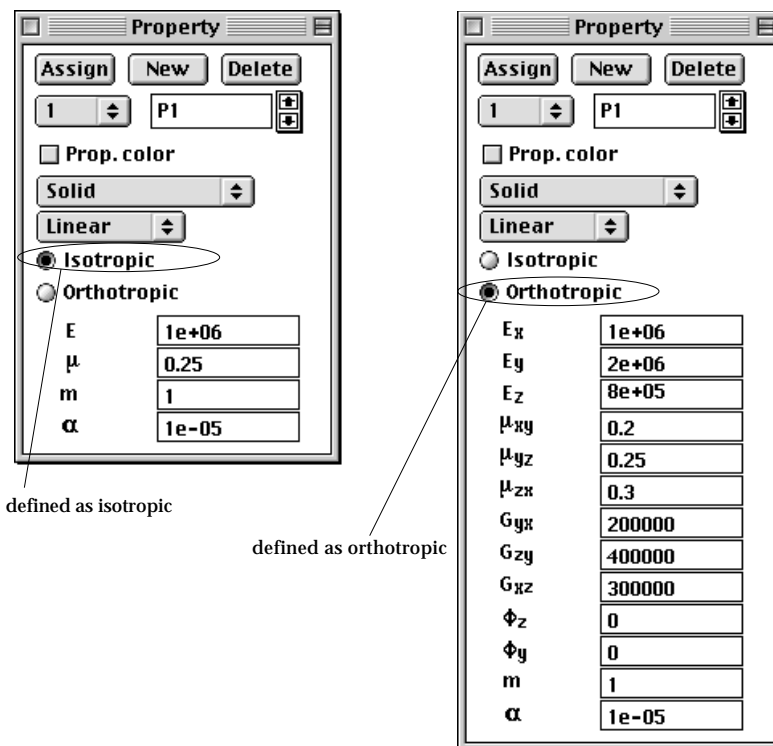
The following are the items available for slip bar elements.

Linear Bonding
Nonlinear Bonding

- "Linear bonding": The bonding between the slip bar and the surrounding body is represented by linear elastic model.
- "Nonlinear bonding": The bonding between the slip bar and the surrounding body is represented by nonlinear stress-strain relationship.

■ Isotropy of the properties

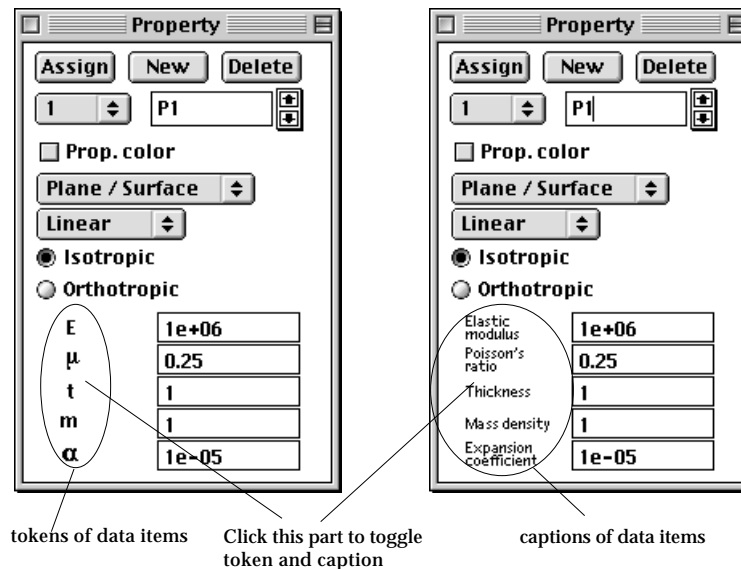
The element properties can be defined as either isotropic or orthotropic using the radio buttons in "Property" dialog. In case the properties are defined as orthotropic, there appear more items in the dialog as shown in the following figure.



<Data items for isotropic materials and for orthotropic ones>

■ Data items of element properties

An element property set consists of a number of data items, which vary depending on the analysis subject, analysis class of the element and the isotropy. As you alter the popup menu items or radio buttons on "Property" dialog, you will notice that the dialog expands or shrinks in its size to accommodate the changing data items properly. Each data item is denoted by a simple token or by a caption. This denotation of data items can be toggled by clicking the part of the dialog as shown in the figure below..

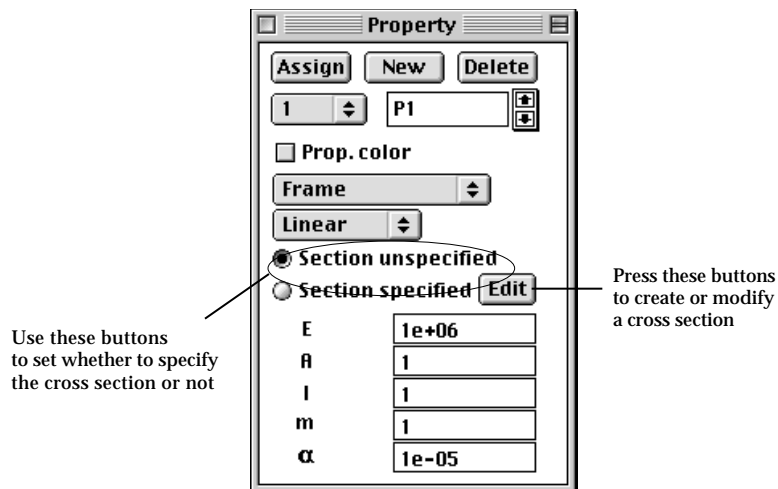


Defining element properties of truss and frame elements

Frame and truss elements are line segments with specific cross sections. The property set of such elements includes the data related to the cross section such as section area, area moment of inertia, and so on. These data can be defined either directly inserting the values to the corresponding items, or indirectly by specifying the shape and the dimensions of cross section.

"Property" dialog for frame or truss element is different from dialogs for other analysis class elements as shown below.

There are 2 radio buttons to set whether to define the element properties with or without defining the cross section. Turn on "Section unspecified" radio button to define the element property without specifying the cross section. Otherwise, turn on "Section specified" radio button to specify the cross section as explained in the following section.

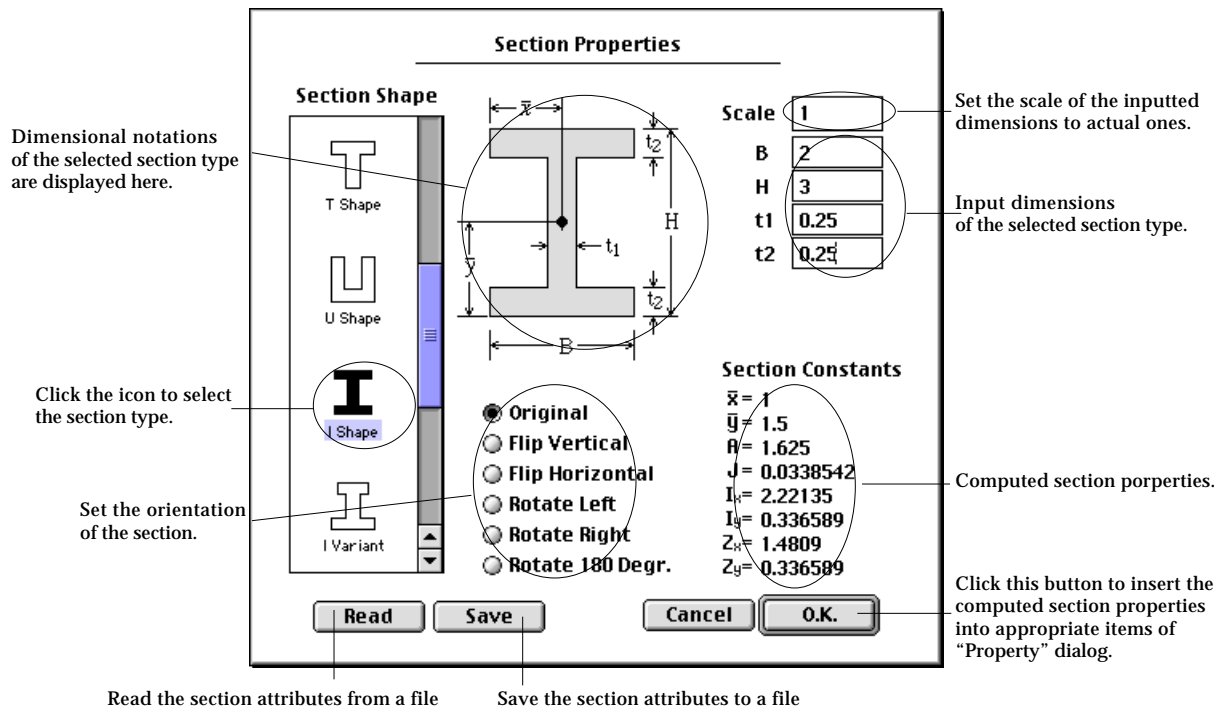


■ Defining cross sections of a truss or a frame member

The cross sectional properties of a structural member can be inputted by dimensioning the section of a selected shape in the following order.

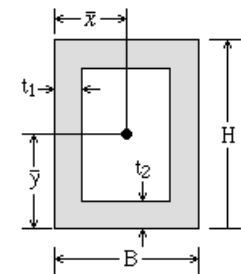
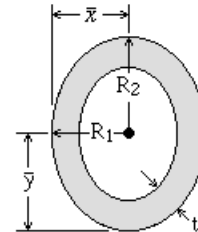
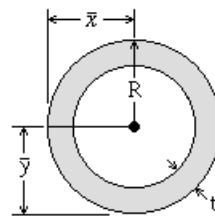
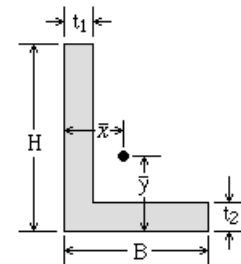
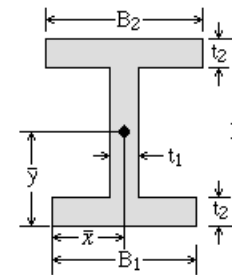
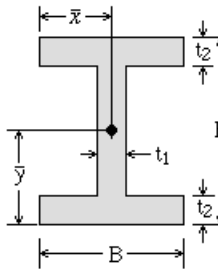
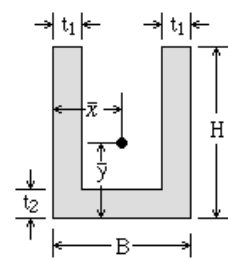
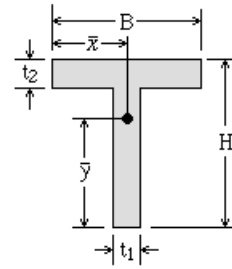
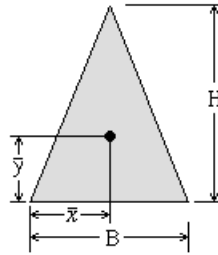
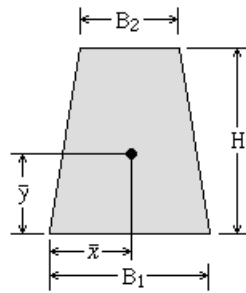
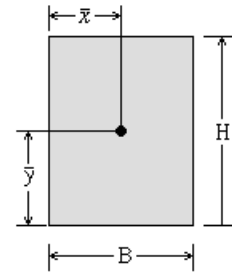
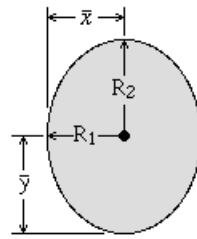
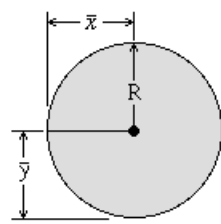
- 1) Click "Defined section" radio button or **Edit** button.

Then, "Section Properties" dialog appears on the screen.



- 2) Scroll the list view of section icons so that the desired icon may be seen.

The current version of VisualFEA has 13 shapes of defined cross section. The list view on the left of the dialog contains the icons of section shape. Only 4 of



<List view icons and dimensional notations of cross sections>

them are shown. The desired one can be made visible by using the scroll bar of the list view.

- 3) Select the desired shape of cross section.

Click the icon of the desired shape. Then, there appears the detailed view of the selected cross section with dimensional notations, and editable text boxes for inputting the dimensions.

- 4) Input dimensions of the section.

Insert the texts into the editable text items representing the dimensions of the cross sections. If all the dimensions are supplied, the computed sectional properties are displayed at the right bottom portion of the dialog.

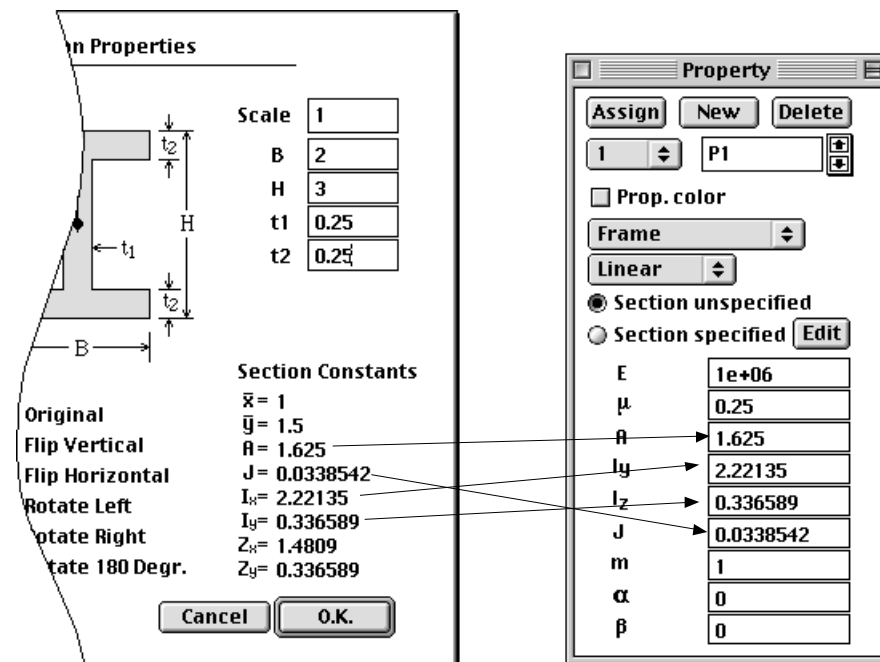
- 5) Set the orientation of the section if necessary

If the desired orientation of the section is not the same as the one displayed on the dialog, that can be altered by choosing one of the following radio buttons:

- "Original" : The orientation is maintained as it is.
- "Flip vertical": The section is flipped about a horizontal axis.
- "Flip horizontal": The section is flipped about a vertical axis.
- "Rotate right": The section is rotated 90° clockwise.
- "Rotate left": The section is rotated 90° counter clockwise.
- "Rotate 180° ": The section is rotated 180°.

- 6) Click **O.K.** button.

The computed cross sectional properties are automatically inserted into the corresponding items of "Property" dialog.



< Automatic insertion of computed cross sectional properties >

Once the cross sectional properties are automatically inserted into the appropriate text items of "Property" dialog, the cross section data is linked to the property set. Thus, "Defined section" radio button is turned on for this set. The cross section of this property set can be edited later by clicking **Edit** button and modifying the text items of "Section Properties" dialog. However, if any item automatically inserted from the cross sectional definition is manually modified, the link between the property set and the section properties is broken and cannot be recovered. Thus, "Undefined section" radio button is turned on for this property set.

Assigning element properties

Element properties are assigned to objects as a set. Only the currently active set is assigned to selected objects by clicking **Assign** button of the dialog. The general procedures of assigning data are explained in a previous section, and so will not be repeated here. Rules of object selection for property assignment and the display of the assignment are briefly explained below.

■ Selecting objects to assign element properties

Element property sets can be assigned to various objects. However, the data sets are eventually assigned to elements constituting the objects. For example, assigning a data set to a volume is equivalent to assigning the set to all elements within the volume. Therefore, the data sets can be assigned either to individual elements or to the kind of objects which contain the actual elements involved in the finite element analysis. The assignable objects differ depending on the analysis subject or the analysis class of element as summarized in the following table.

< Assignable objects for element properties >

analysis subject	analysis class of element	assignable objects
plane stress/strain axisymmetric, plate bending shell structure	plane/surface	surface mesh, surface element
3-D solid structure	volume	volume mesh, volume element
2-D truss 3-D truss	truss	curve, truss element
2-D rigid frame 3-D rigid frame	frame	curve, frame element
interface	interface	curve
slip bar	slip bar	curve
embedded bar	embedded bar	curve

■ Overriding previous assignment of element properties

If you assign a data set to objects on which another data set has already been assigned, the newly assigned set will replace the old one. Thus, one data set at most is assigned to an element.

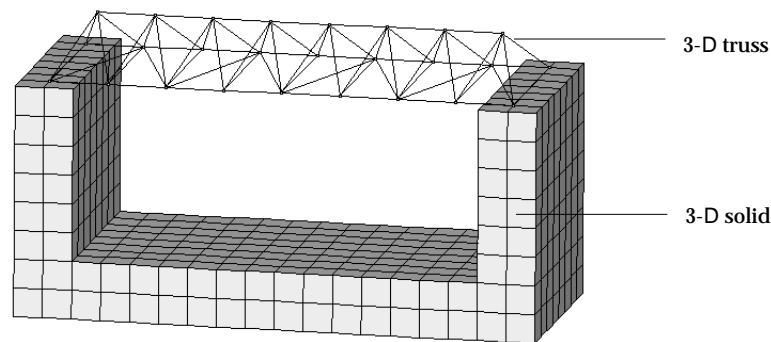
Because of this overriding behavior, it is more efficient to start assigning property sets from larger regions and to proceed to smaller parts. For example, if most of a mesh has one property and a remaining small part has another, assign one property set to the mesh as a whole first and another set to the relevant elements.

■ Representation of element property assignment

Elements assigned with element properties are drawn in red. The elements assigned with the currently active data set are highlighted in dark red. The assignment is always represented by boundary lines of each element, regardless of the rendering mode.

Mixing different structural types in one analysis

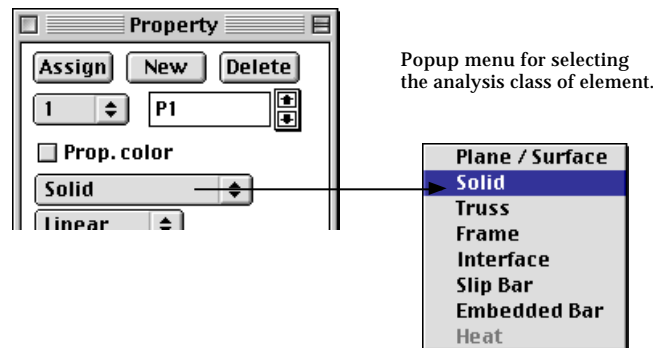
Different types of structures may be included in one analysis. For example, you may model a 3-D solid structure in which 3-D truss members exist as shown below. This can be achieved defining and assigning element property sets of different analysis classes using the popup menu in “Property” dialog.



< Example of a model with mixed structural types >

■ Changing the analysis class of element

The popup menu in “Property” dialog has a few items representing analysis classes. When the dialog appears for the first time, the popup menu is set to the item of analysis class corresponding to the current analysis subjects. If you change the item, all the dialog items are also altered accordingly. At the same time, the assignable objects are changed, and the relevant object selection tool is automatically activated.



■ Applicable analysis classes of element

In VisualFEA, not all types of structures can be mixed with each other. There are applicable types for a given analysis subject. In the popup menu shown above, only the items of applicable analysis classes are enabled. Listed in the following table are the structural types which can be included for given analysis subjects.

< Structural types applicable for a given analysis subject >

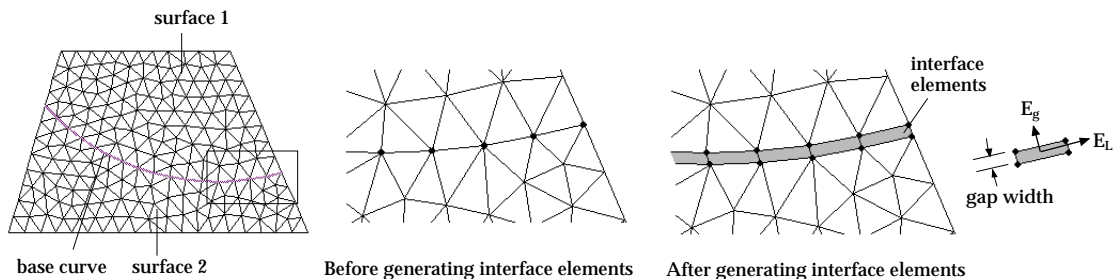
analysis subject	applicable structural types	
	structural type	analysis class of element
plane stress/strain axisymmetric	2-D truss 2-D frame interface slip bar embedded bar	truss frame interface slip bar embedded bar
3-D solid structure	2-D truss 2-D frame shell slip bar embedded bar	truss frame plane/surface slip bar embedded bar
2-D rigid frame	2-D truss	truss
3-D rigid frame	3-D truss	truss

Using interface elements

Interface elements are used in modeling the behavior of gaps with negligible width. They cannot be used alone, but may be included in plane stress, plane strain or axisymmetric problems. (The current version of VisualFEA supports only 2 dimensional interface elements.)

■ Characteristics of interface elements

Interface elements fill the gap separating two adjacent surfaces. They connect the nodes on the boundaries of both surfaces. VisualFEA supports only 4 node interface elements. The structural behavior of the gap can be controlled by the stiffness of the interface elements in width and length directions. These stiffness are determined by the moduli of elasticity in both directions, and the gap width of the elements. A pair of nodes facing with each other are separated by as much as the specified gap width. Assigning zero or very small value of gap width or large modulus of elasticity in width direction will restrain the relative movement in width direction, of the pair of nodes. On the other hand, the modulus of elasticity in length direction controls the relative movement in length direction. So, you may model the desired behavior of the gap, or relative movement of the contacting surfaces by assigning appropriate values to the properties of interface elements.



< Characteristics of interface elements >

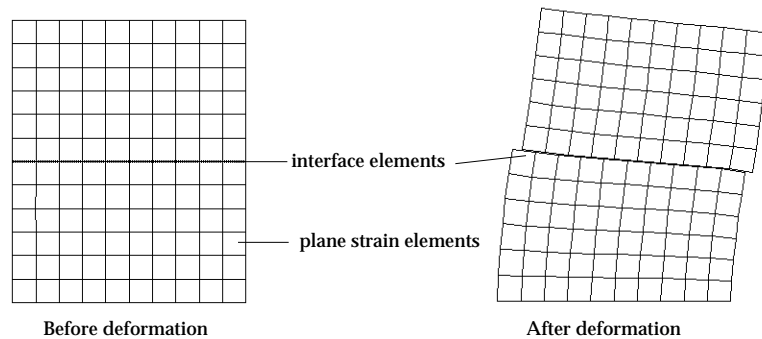
■ Creating interface elements

Interface elements are created somewhat differently from other structural types of elements. They are not created by mesh generation or curve input, but by assignment of element properties as explained below.

- 1) Choose "Interface" item from the popup menu in "Property" dialog.
The data items in "Property" dialog are altered.
- 2) Enter the value for each data item.
The interface element can be made to represent specific behavior by entering appropriate values for the data items.

- modulus of elasticity in length direction (E_L): The Young's modulus of elasticity in the direction parallel to the length of the interface elements.
 - modulus of elasticity in gap width direction (E_g): The Young's modulus of elasticity in the direction normal to the length of the interface elements.
 - gap width (**gap**): The width of interface of elements. This value should be small as compared with the dimension of the whole structure.
 - thickness (**t**): This item is applicable only for plane stress problems. Usually, this value is set as identical to the thickness of the main structure in which the interface elements are embedded.
- 3) Select one or more base curves to assign interface elements.
The selected base curves are to be used as the basis of the interface elements. Therefore, they should be serially connected, and completely embedded within the main structure.
- 4) Click **Assign** button in the dialog.
Interface elements are created on the selected curves. The active set of element properties are assigned to the newly created elements.

When interface elements are created, the basis curves together with the nodes on them are duplicated. The interface elements are composed of the nodes on the new and existing base curves.



< Example of a plane strain case with interface elements >

■ Deleting interface elements

Interface elements can be deleted only by deleting the set of element property assigned to them. The meshes are restored to the state before creating interface elements. That is, the surface meshes separated by the interface elements are joined again.

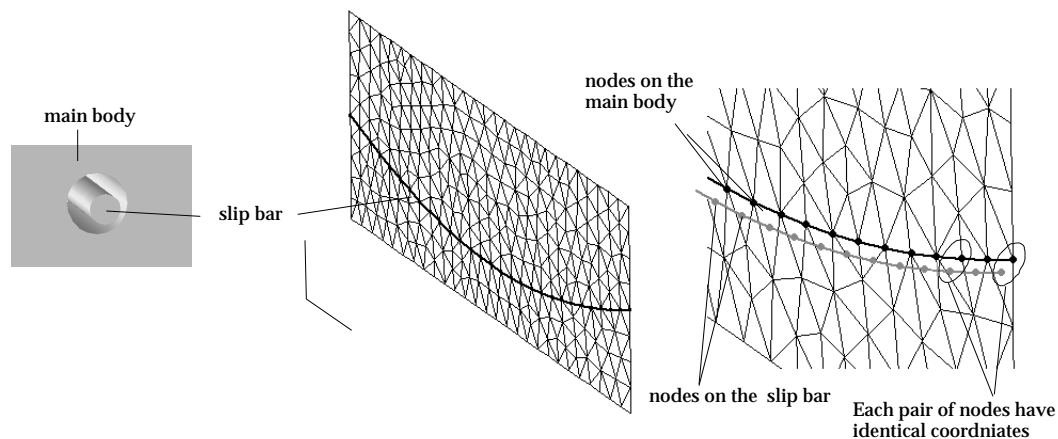
Using slip bars

Slip bar elements are used in modeling bars, cables, rods or other reinforcement in planar or 3-D solid structures. For example, tendons within prestressed concrete beams can be best modeled by using slip bars. Slip bars cannot be used alone, but may be included in plane stress, plane strain, axisymmetric, or 3-D solid structures.

■ Characteristics of slip bar elements

Slip bars are objects embedded within, but independent from, a body of planar or 3-D solid. There can be bonding effects between the slip bar and the main body. These behaviors are modeled by using slip bar elements, which are a kind of truss elements combined with bonding effects. Conceptually, nodes on slip bar are surrounded by the nodes paired within the main body. Numerically, a pair of nodes, one in the slip bar and the other in the main body, are independent from each other, but share identical coordinates.

A pair of nodes are allowed to slip against each other in the direction along the slip bar element. But they are forced to move together in its normal direction. The bonding effect between the slip bar and the main body is represented by the stiffness against the slip of paired nodes.



< Concept of slip bar elements >

■ Creating slip bar elements

Slip bar elements are created somewhat differently from other structural types of elements, but similar to interface elements. They are not created by mesh generation or curve input, but by assignment of element properties as explained below.

- 1) Choose "Slip Bar" item from the popup menu in "Property" dialog.

The data items in “Property” dialog are altered. A property set of slip bar element is always accompanied by an annexed property set. The annexed set has the attributes for truss elements representing truss behavior of the slip bar. This annexed set can be modified but cannot be assigned independently.

- 2) Enter the value for each data item.

The slip bar element can be made to represent specific behavior by entering appropriate values for the data items.

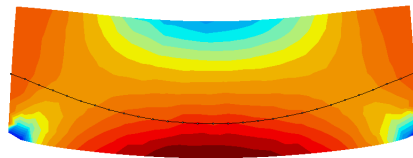
- modulus of elasticity of the slip bar (E_{bar}): The Young’s modulus of elasticity of the slip bar elements.
- modulus of elasticity representing (E_{bond}): The Young’s modulus of elasticity representing the bonding effect between the bar and the solid.
- section area (A): The section area of the bar.

- 3) Select one or more curves to assign slip bar elements.

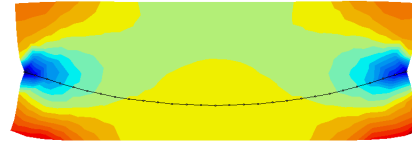
The selected curves are to be used as the basis of the slip bar elements. Therefore, they should be serially connected, and completely slip within the main body.

- 4) Click **Assign** button in the dialog.

Slip bar elements are created on the selected curves. The active set of element properties are assigned to the newly created elements.



Without prestressing



With prestressing

< Example of modeling a prestressed beam using slip bar elements >

■ Deleting slip bar elements

Slip bar elements can be deleted only by deleting the set of element property assigned to them. When a property set for slip bar is deleted, its annexed set is also removed.

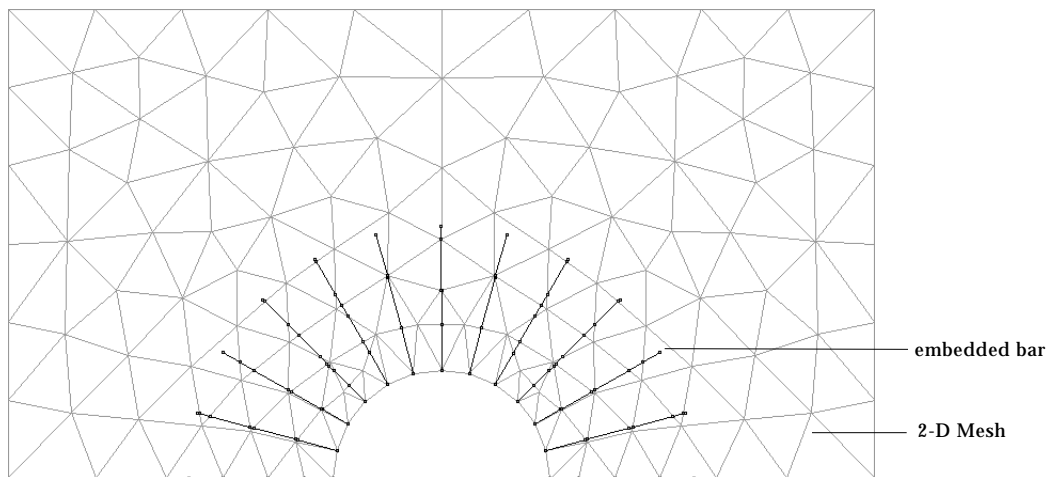
Using embedded bars

Similarly to slip bars, embedded bars are also used in modeling stiffening material embedded within planar or 3-D solid structures. Embedded bars are more convenient and flexible to use than the slip bars, because they can be arbitrarily placed within continuum. Embedded bars cannot be used alone, but may be included in plane stress, plane strain, axisymmetric, or 3-D solid structures.

■ Characteristics of embedded bar elements

Embedded bars and slip bars are similar in their usage as stiffening line segments embedded within continuum, but different in their characteristics and in their modeling method. A slip bar is an element which has nodes and its own displacement field, while an embedded bar is not an independent element but a stiffening segment embedded within continuum elements. Thus, an embedded bar has neither independent displacement field nor nodes. Its displacements are defined only through the deformation of the surrounding elements. The stiffness of the embedded bars are directly added to the surrounding elements. The strains of embedded bars are computed from the displacements of the surrounding elements.


An embedded bar can pass through a number of elements without restriction in the position or in the direction of its path. Embedded bars may be placed freely within continuum without involving nodal connectivity conditions as shown in the figures of this and the next pages. So, embedded bars can be employed conveniently in modeling complex reinforcement within continuums, especially in combination with unstructured meshes.



< An example of embedded bars placed within a 2-D unstructured mesh >

■ Creating embedded bars

Any straight line can be defined as an embedded bar by assignment of element properties as explained below.

- 1) Create straight lines using the line tool .

Many embedded bars with a certain regular arrangement may be created efficiently by using duplication of curves. The figure at the bottom of this page shows an example of thousands of embedded bars created mostly by "Duplicate and Move" and "Duplicate and Revolve". Refer to "Duplicating curves and surface primitives" section of Chapter 3.

- 2) Start property assignment procedure.

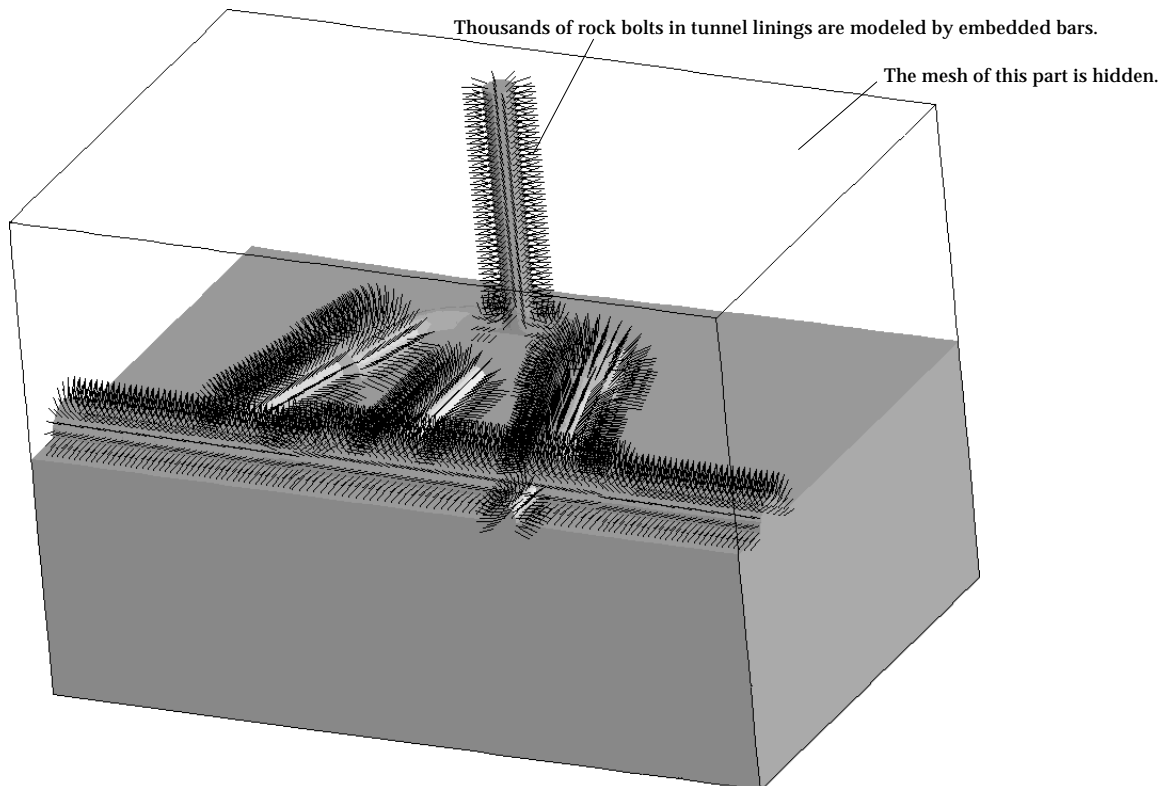
Choose "Element Property" item from **Assign** menu.

- 3) Choose "Embedded Bar" item from the popup menu in "Property" dialog.

The data items in "Property" dialog are altered to the items of embedded bars.

- 4) Enter the value for each data item.

The data items are related to the axial stiffness of the embedded bar. It is assumed that the section area of the embedded bar is subtracted from the embedding elements. Let A_e and E_e be the section area and the elastic



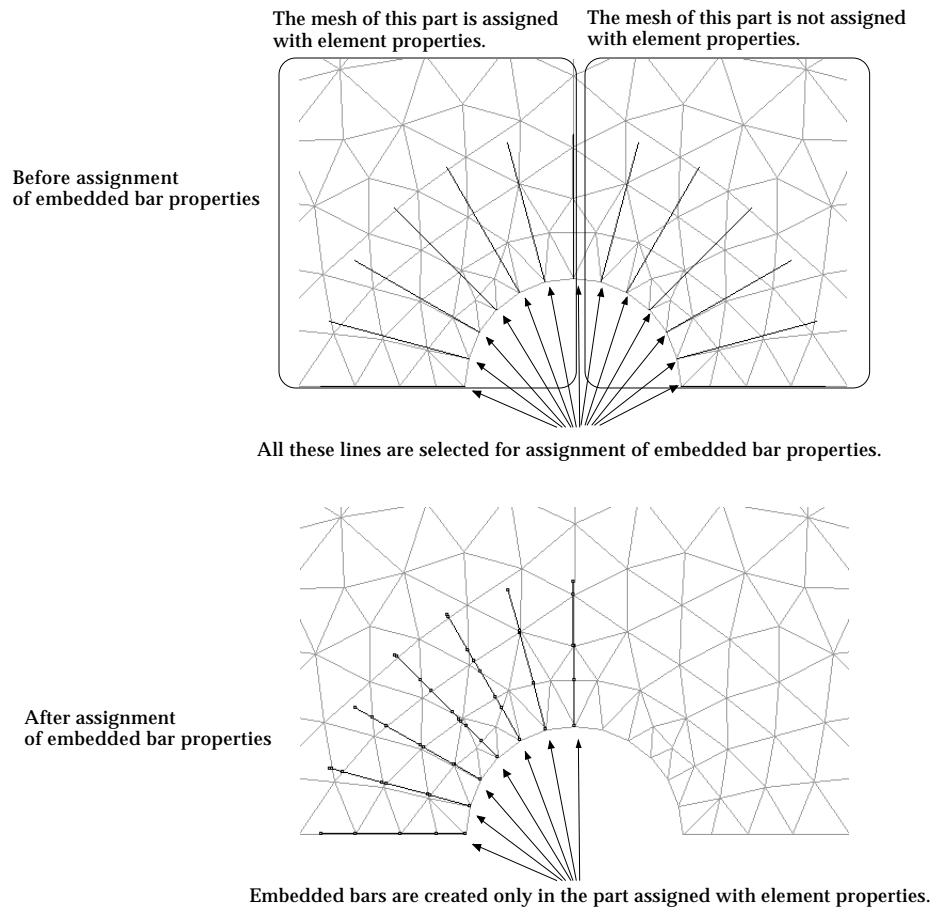
< An example of embedded bars in a 3-D solid model >

modulus of the embedded bar, and E_o be the elastic modulus of the embedding element. Then, the stiffness contribution of the embedding bar is

$$k = A_e (E_e - E_o)$$

- 5) Select one or more lines to assign embedded bar properties.
- 6) Click **Assign** button in the dialog.

The selected curves turn into embedded bars only when the surrounding continuum elements are assigned with element properties as illustrated by the figure below. The intersections of embedded bars and element edges are marked by small circles.



< Creation of embedded bars >

■ Deleting embedded bar

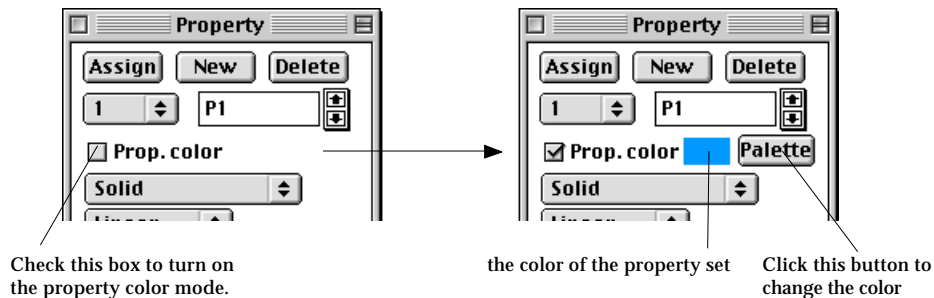
The embedded bar properties are removed by deleting the embedded bar property set, or by clearing the assigned property set. The line segments remain undeleted, although the property assignment is removed.

Color coding of property sets

Property sets can be coded by colors. If the property color mode is turned on, assignment of property set to each part of the model is distinguished by the color used for rendering the part either in wireframe or in shaded image. Any parts without property assignment are rendered in gray. The default color codes are predefined for the first 16 property sets. The color of each set can be altered using the color palette functions of the corresponding operating system.

■ Turning on the property color mode

The property color mode is turned on by checking the "Prop. color" box of "Property" dialog. The color box filled with the color of the current property set and "Palette" button appear on the dialog when the property color mode is turned on.



■ Changing or setting the property color

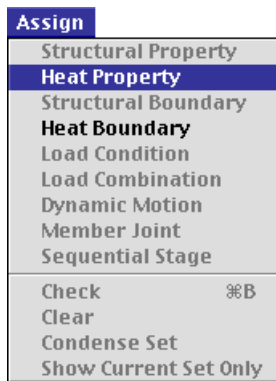
Click **Palette** button to change the property color. Then, the color palette dialog appears on the screen. The default color for the first 16 sets are shown in the palette. (Windows only) You may change the default color, or set a new color code for the property sets beyond the first 16 sets.

It should be noted that gray is the reserved color for rendering the part without property assignment, and cannot be used as a property set color.

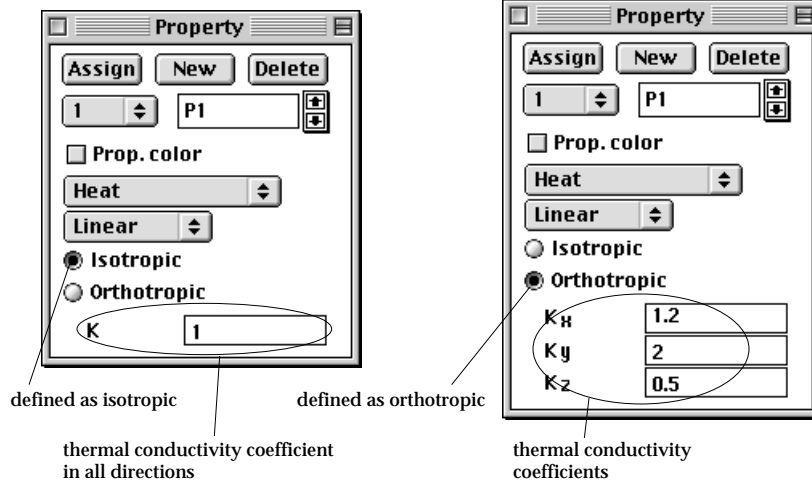
Heat Conduction and Seepage Properties

The element properties for heat conduction or seepage analysis are defined and assigned in the similar way as that of structural analysis. The related actions are initiated by "Heat Property" (or "Seepage Property") item of **Assign** menu.

Heat conduction properties



If the analysis subject is defined as a heat conduction problem, "Heat Property" item appears in **Assign** menu and is enabled. Choose the item in order to start assigning heat conduction properties. "Property" dialog appears, and the current state of their assignment are displayed in the main window.



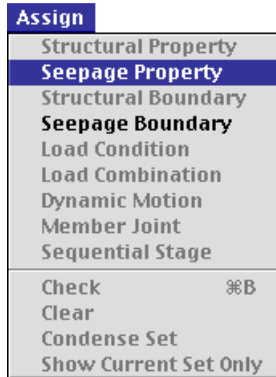
■ Defining heat conduction properties

Heat conduction properties include the thermal conductivity coefficients in X and Y directions for 2-D heat conduction problems, and X, Y and Z directions for 3-D heat conduction problems. If the conductivity is isotropic, click "Isotropic" button in the dialog. Then, the dialog shows only one editable text item "K". If the conductivity is orthotropic, click "Orthotropic" button. Then, the dialog shows the editable text item of the thermal conductivity coefficient for each of the coordinate axis directions. Inserting the value(s) in the editable text box(es) complete the defining of heat conduction properties.

■ Assigning heat conduction properties

Heat conduction properties can be assigned only to surface meshes in the case of plane and axisymmetric heat conduction problems, and only to volume meshes in the case of 3-D heat conduction problems.

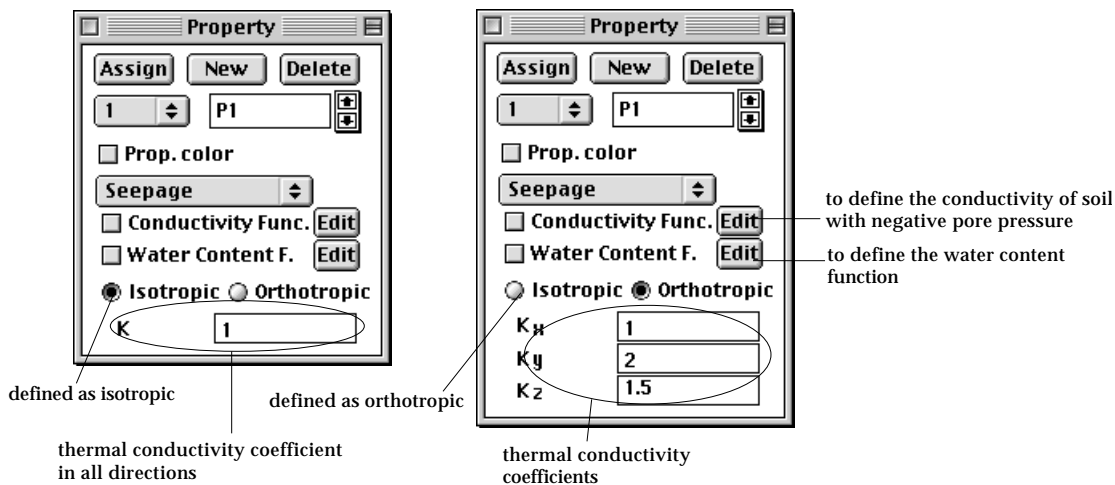
Seepage properties



If the analysis subject is defined as a seepage problem, "Seepage Property" item appears in **Assign** menu and is enabled. Choose the item in order to start assigning seepage properties. "Property" dialog appears, and the current state of their assignment are displayed in the main window.

■ Defining seepage properties

Seepage properties include the hydraulic conductivity coefficients in X and Y directions for 2-D seepage problems, and in X, Y and Z directions for 3-D seepage problems. The conductivity can be represented by one value for isotropic material as shown in the figure below. Check "Isotropic" radio button to input the conductivity coefficients as isotropic material.



Seepage properties include conductivity functions for unconfined seepage analysis. The conductivity function defines the ratio of the reduced conductivity to the full conductivity as a function of pore pressure.

■ Defining hydraulic conductivity functions

A conductivity function is associated with each one of the seepage property sets. Click **Edit** button in the "Property" dialog to input a conductivity function for the current property set. Then, "Hydraulic Conductivity Function" dialog appears on the screen. The conductivity function can be defined either by equation or by table using this dialog as described below.

- 1) Click **Edit** button of "Property" dialog to open "Hydraulic Conductivity Function" dialog.

The dialog opens with default settings. The dialog consists of 2 parts as

shown in the figure below. The first part is used to define the hydraulic conductivity function by an equation, and the other part to define the function in tabular form.

Hydraulic Conductivity Function

☒ **Defined by equation**

$$c = \frac{1}{1 + \alpha |P|^n}$$

$\alpha = 0.1$
 $n = 2$

c = conductivity factor
 α = Gardner coefficient
 P = negative pore pressure

☐ **Defined by table**

No.	Pore Pressure	Conductivity Factor
1	0	1

0 Add Delete Save Import Graph Cancel O.K.

to add or insert a new row in the table.
to delete the selected row in the table.
to read the table saved as a text file.
to save the table as a text file.
to display or edit the conductivity function in graphic mode.

- 2) Turn on "Define by equation" radio button to define the conductivity function by an equation.

The hydraulic conductivity factor c is computed as a function of negative pore pressure P by the equation,

$$c = \frac{1}{1 + a|P|^n}$$

where a is the Gardner coefficient and n is a power factor.

The conductivity factor implies the ratio of the reduced conductivity in negative pore pressure region to the full conductivity.

- 3) Insert the Gardener coefficient a , and power factor n .
The value of a is normally assumed between 0.001 and 0.1, and the value of n between 2 and 4.
- 4) Turn on "Define by table" radio button to define the conductivity function in tabular form.
The hydraulic conductivity is defined by the tabulated relation between the

pore pressure and the conductivity factor. The conductivity factor is computed by linear interpolation of the given pore pressures and the conductivity factors.

☒ Defined by table

No.	Pore Pressure	Conductivity Factor
1	-1	0.5
2	-2	0.2
3	-5	0.1
4	-10	0.05
5	-20	0.01

0.01

Add Delete

Save Import

Graph Cancel O.K.

- 5) Click **Add** button to add a new row in the table, and **Delete** button remove a row.

The buttons are enabled when "Defined by table" radio button is turned on. After creating a new row, insert the values of the row using the editable text box.

- 6) Click **O.K.** button to complete defining the conductivity function.

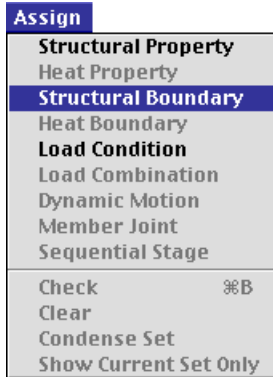
The "Hydraulic Conductivity Function" dialog closes, and the conductivity function is added to the current seepage property set. The check box of "Conductivity Func." item of the "Property" dialog is marked to indicate that the hydraulic function of the property set is defined.

■ Assigning seepage properties

In order to assign seepage properties, first select the object to assign, and click **Assign** button of the "Property" dialog. Seepage properties can be assigned only to surface meshes in the case of plane and axisymmetric seepage problems, and only to volume meshes in the case of 3-D seepage problems.

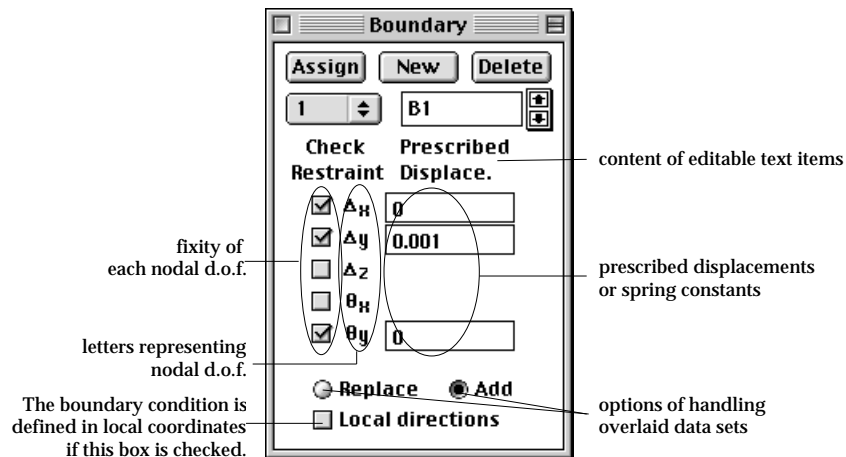
Structural Boundary Conditions

Structural boundary conditions specify the way a structure is supported. The data items of the structural boundary condition are related to the constraining status of nodes at the boundaries. The status is defined for each nodal d.o.f., and is one of the following four.



- free : free to move. In other words, no constraint is imposed.
- free with spring constant : partially constrained state like hinges with friction, elastic supports, etc. The degrees of constraint are represented by spring constants.
- fixed: No movement is allowed. The 0 displacement is prescribed.
- fixed with prescribed displacements : initial displacement, settlements, etc. The displacement is forced to be the prescribed value.

In order to start assigning structural boundary conditions, choose “Structural Boundary” item in **Assign** menu. “Struct Boundary” dialog appears, and current state of their assignment are displayed in the main window.



Defining structural boundary conditions

You must first create and define sets of boundary conditions before assigning them on the structural boundaries. All the data items of the active set are displayed on “Struct Boundary” dialog and can be entered or modified using the dialog.

■ Nodal degrees of freedom

The boundary condition dictates the state of each nodal degree of freedom. The nodal d.o.f. differ depending on the subject of analysis. If more than one structural type is involved in the analysis, the items are extended to hold all associated structural types.

2-D truss	3-D truss	2-D rigid frame	3-D rigid frame
<input checked="" type="checkbox"/> Δ_H	<input checked="" type="checkbox"/> Δ_H	<input checked="" type="checkbox"/> Δ_H	<input checked="" type="checkbox"/> Δ_H
<input checked="" type="checkbox"/> Δ_y	<input checked="" type="checkbox"/> Δ_y	<input checked="" type="checkbox"/> Δ_y	<input checked="" type="checkbox"/> Δ_y
	<input checked="" type="checkbox"/> Δ_z	<input checked="" type="checkbox"/> θ_z	<input checked="" type="checkbox"/> Δ_z
			<input checked="" type="checkbox"/> θ_H
			<input checked="" type="checkbox"/> θ_y
			<input checked="" type="checkbox"/> θ_z
plane stress plane strain axisymmetric	plate bending	shell	3-D solid
<input checked="" type="checkbox"/> Δ_H	<input checked="" type="checkbox"/> Δ_z	<input checked="" type="checkbox"/> Δ_H	<input checked="" type="checkbox"/> Δ_H
<input checked="" type="checkbox"/> Δ_y	<input checked="" type="checkbox"/> θ_H	<input checked="" type="checkbox"/> Δ_y	<input checked="" type="checkbox"/> Δ_y
	<input checked="" type="checkbox"/> θ_y	<input checked="" type="checkbox"/> Δ_z	<input checked="" type="checkbox"/> Δ_z
		<input checked="" type="checkbox"/> θ_H	
		<input checked="" type="checkbox"/> θ_y	

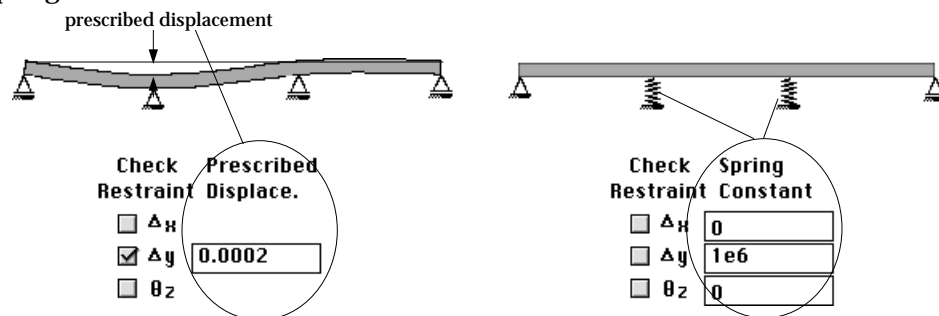
< Nodal d.o.f for various analysis subjects >

■ Data items of structural boundary condition

A structural boundary condition set has a few data items. They consist of nodal fixity, initial displacements or spring constants defined for each d.o.f.

- **fixity:** the state of nodal constraint. Either “free” or “fixed” condition is specified for each nodal d.o.f., and indicated in the check boxes of the dialog. If a box is checked, the corresponding d.o.f. is fixed.
- **prescribed displacement:** nodal displacements given prior to solving the problem. Nodal displacements can be specified only for fixed d.o.f.
- **spring constant:** This value represents the degree of partial constraint. The spring constants can be specified only for free d.o.f.

The following figure shows examples of applying prescribed displacements and spring constants.



< Nodal d.o.f for various analysis subjects >

■ Entering data items of boundary conditions

The fixity of each nodal d.o.f. is entered by clicking the check box in front of the symbol indicating the respective d.o.f. The prescribed displacements and the

spring constants are entered within the corresponding text boxes. As shown below, not all the text boxes are displayed. Some of them are hidden. Displayed items are related to the heading of the text items. There are 3 headings:

- “Prescribed Displace.”: This heading indicates that only text items for prescribed displacements are shown. Because displacements can be prescribed only for fixed d.o.f., the text items of fixed d.o.f. are displayed.
- “Spring Constant”: This heading indicates that only text items for spring constants are shown. Because spring constants can be applied only for free d.o.f., the text items of free d.o.f. are displayed.
- “Mixed Values”: This heading indicates that text items for spring constants as well as for prescribed displacement are shown. Thus, all text items are displayed.

The heading is initially set as “Prescribed Displace.” You may scroll through the above three headings by clicking any point over the heading. But, only applicable ones will be scrolled. For example, if non-zero values are entered in prescribed displacements, “Spring Constant” will be skipped.

Click here to scroll the heading.

Check Restraint		Prescribed Displace.	
<input checked="" type="checkbox"/>	Δ_H	0	indicates that the prescribed displacements are entered in the following text boxes. Enter the prescribed displacements in these boxes. Leave 0 for no displacement.
<input checked="" type="checkbox"/>	Δ_y	0.006	
<input type="checkbox"/>	Δ_z		
<input type="checkbox"/>	θ_H		
<input checked="" type="checkbox"/>	θ_y	0	

These checked d.o.f. are fixed.
These unchecked d.o.f. are free.

Check Restraint		Spring Constant	
<input checked="" type="checkbox"/>	Δ_H		indicates that the spring constants are entered in the following text boxes. Enter the spring constants in these boxes. Leave 0 for no spring constant, i.e., for completely free d.o.f.
<input checked="" type="checkbox"/>	Δ_y		
<input type="checkbox"/>	Δ_z	25000	
<input type="checkbox"/>	θ_H	0	
<input checked="" type="checkbox"/>	θ_y		

Check Restraint		Mixed Values	
<input checked="" type="checkbox"/>	Δ_H	0	indicates that either prescribed displacements or spring constants are entered depending on the fixity of the corresponding d.o.f.
<input checked="" type="checkbox"/>	Δ_y	0.006	
<input type="checkbox"/>	Δ_z	25000	
<input type="checkbox"/>	θ_H	0	
<input checked="" type="checkbox"/>	θ_y	0	

< Headings of editable text items >

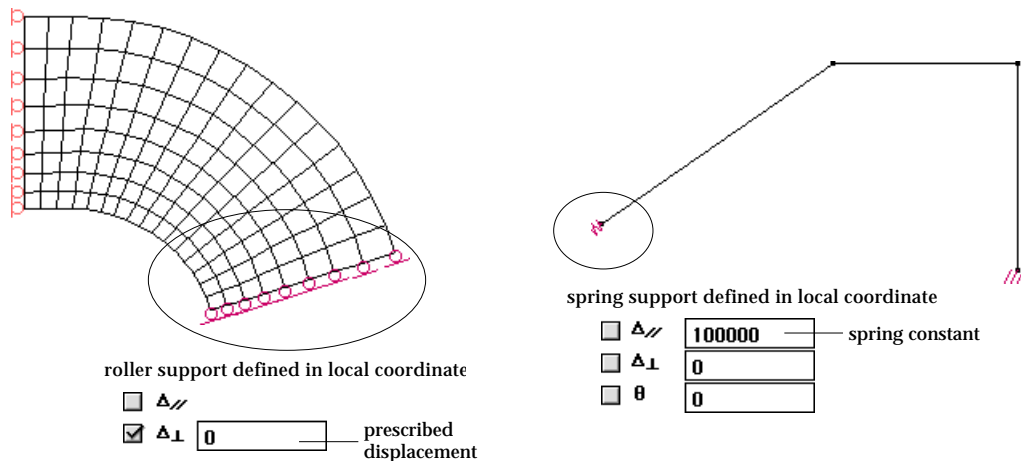
■ Defining boundary conditions in local coordinates

The structural boundary conditions can be defined in either in global coordinates or in local coordinates. The local coordinates are based on the axis of a structural member, the tangent direction of a surface edge, or the normal to a surface. Check "Local direction" box of "Struct Boundary" dialog in order to use local coordinates for a boundary condition set. At the moment, the "Local direction" is checked, the d.o.f. labels are altered as shown in the figure below.

2-D truss plane stress plane strain axisymmetric	3-D truss 3-D solid	2-D frame	
<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> Δ_{\perp}	<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> $\Delta_{/z}$	<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> Δ_{\perp} <input checked="" type="checkbox"/> θ	$\Delta_{//}$ translation in the axial direction Δ_{\perp} translation in the direction normal to axis (truss/ frame) or to surface (shell)
3-D frame	shell	plate	
<input checked="" type="checkbox"/> $\Delta_{//}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> $\Delta_{/z}$ <input checked="" type="checkbox"/> $\theta_{//}$ <input checked="" type="checkbox"/> $\theta_{/y}$ <input checked="" type="checkbox"/> $\theta_{/z}$	<input checked="" type="checkbox"/> $\Delta_{/H}$ <input checked="" type="checkbox"/> $\Delta_{/y}$ <input checked="" type="checkbox"/> Δ_{\perp} <input checked="" type="checkbox"/> $\theta_{/H}$ <input checked="" type="checkbox"/> $\theta_{/y}$	local coordinates not defined	$\Delta_{/H}$ translation in the local x direction(shell) $\Delta_{/y}$ translation in the local y direction $\Delta_{/z}$ translation in the local z direction $\theta_{//}$ axial rotation (shell) $\theta_{/y}$ rotation about the local y axis(shell) $\theta_{/z}$ rotation about the local z axis(shell)

< Boundary constraints in local coordinates >

Use of local coordinates may be most appropriate for roller support and spring boundary condition which are illustrated in the figure below. The prescribed displacements and the spring constants are entered for the local coordinates in the same way as for the global coordinates.



< Examples of boundary conditions defined in local coordinates >

In most cases, only the axial direction or the normal direction is concerned with the boundary conditions in local coordinates. The axial and normal directions at every node is determined in association with the object to which the boundary condition is assigned.

- **curve:** In the case the boundary condition is assigned to a curve, the axial (//) direction is defined as tangent to the curve at every node on the curve. For planar models, the normal () direction is uniquely determined from the axial direction. For 3-D models, the 2 normal (/y and /z) directions are determined by vector operations of the unit axial vector, \vec{t} and the unit vector in the global Z direction, \vec{k} .

$$\vec{v}_y = \vec{k} \times \vec{t}$$

$$\vec{v}_z = \vec{t} \times \vec{v}_y$$

where \vec{v}_y and \vec{v}_z represent unit vectors in the local y and z directions respectively.

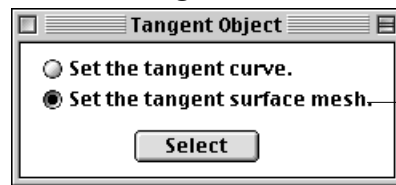
- **surface mesh:** In the case the boundary condition is assigned to a surface mesh, the normal direction () is first determined at every node on the surface mesh. And the 2 other (/x and /y) directions are determined by vector operations of the unit normal vector \vec{n} and the unit vector in the global Y direction, \vec{j} .

$$\vec{v}_x = \vec{j} \times \vec{n}$$

$$\vec{v}_y = \vec{n} \times \vec{v}_x$$

where \vec{v}_x and \vec{v}_y represent unit vectors in the local x and y directions respectively.

- **node:** If the node belongs to only one curve or only one surface mesh, the normal or the axial direction is determined by the curve and the surface. But, if the node shared by multiple curves or by multiple surfaces, then the program asks to choose the curve or the surface mesh to be used as the basis for determining the local directions.



This item is dimmed in the case of a frame, or a planar problem.

First, choose one of the two radio buttons, "Set the tangent curve", or "Set the tangent surface mesh. (For a frame or a planar problem, the second button is dimmed, and the first button is turned on.) And select the curve or the surface mesh for the basis of the local directions. Then, **Select** button becomes active. Click the button to assign the boundary condition in local coordinates.

Assigning structural boundary conditions

The structural boundary conditions are assigned to objects as a set. The currently active set is assigned to selected objects by clicking **Assign** button of the dialog. It is assumed that all the nodes are initially free. That is, none of the nodal d.o.f. are constrained fully or partially at the beginning. Assigning a boundary condition set is equivalent to modifying the unconstrained state of each nodal d.o.f. Thus, the boundary condition sets are assigned only to the nodes, one or more d.o.f. of which are fixed or free with spring constant. It is meaningless to define or assign a boundary condition set with all d.o.f. free. If you define such a set and assign it to the selected objects, the assignment is ignored.

■ Selecting objects to assign structural boundary conditions

Structural boundary conditions may be assigned to nodes, curves and surface meshes. However, the data sets are eventually assigned to nodes within the objects. For example, assigning a boundary condition set to a surface mesh is equivalent to assigning the set to all nodes within the surface mesh.

For a 3 dimensional solid, boundary conditions are usually assigned only to the outer surfaces, or to nodes on these surfaces. Therefore, it is sometime necessary to prevent assigning boundary conditions to nodes inside the volume inadvertently. This can be done by checking “Shut Invisible Nodes” item in **View** menu so that the inside nodes become unselectable.

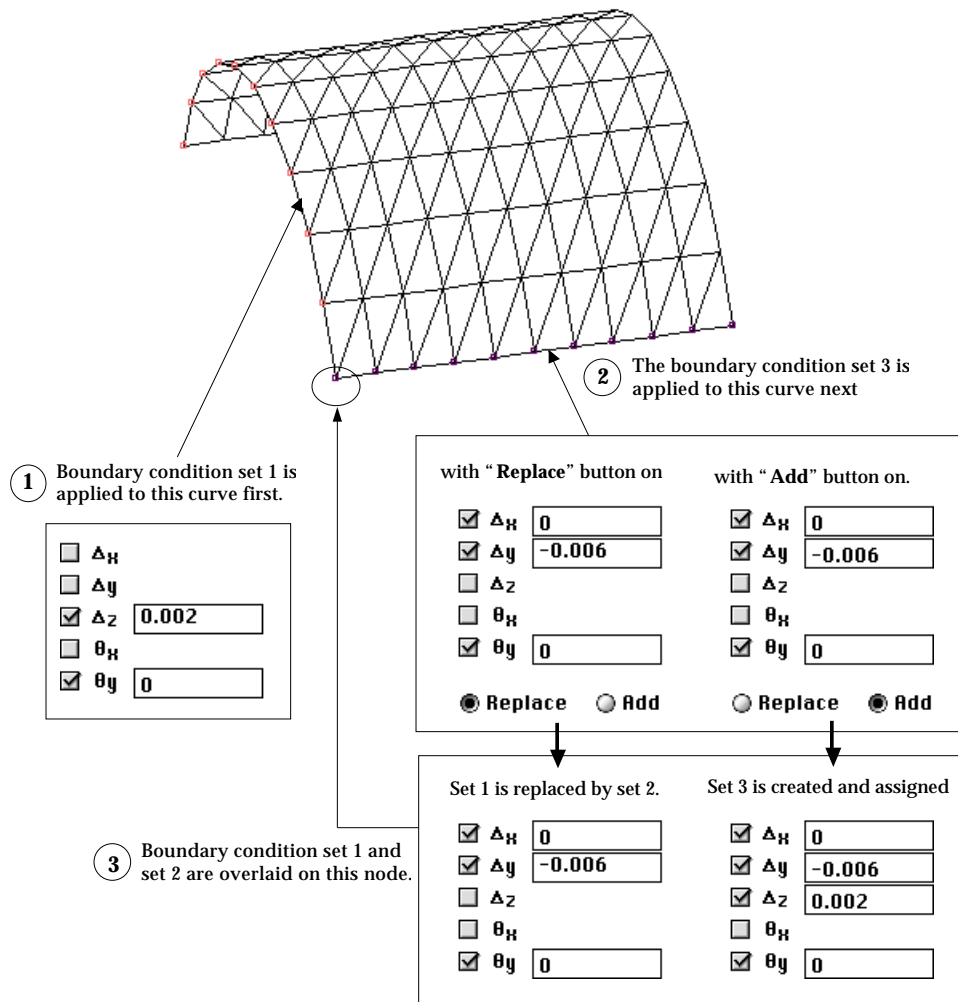
■ Replacing or adding previous assignment

When a new structural boundary condition set is assigned to a node with previously assigned boundary conditions, the old set will be either replaced by or added to the new one, depending on the mode of overlaying data sets.

In “Struct Boundary” dialog, there are two radio buttons, “Replace” and “Add”. If “Replace” button is on, the old assignment is replaced by the new assignment. The text data items are also replaced by the corresponding values of the new set.

If “Add” button is on, the resulting assignment is the union of the old and the new assignments. The state of a d.o.f. is determined by bit ORing with ‘0’ for free d.o.f. and ‘1’ for fixed d.o.f. If the editable text data of both old and new sets are of same kind(both prescribed displacements or both spring constants), the item is filled with the one whose absolute value is larger.

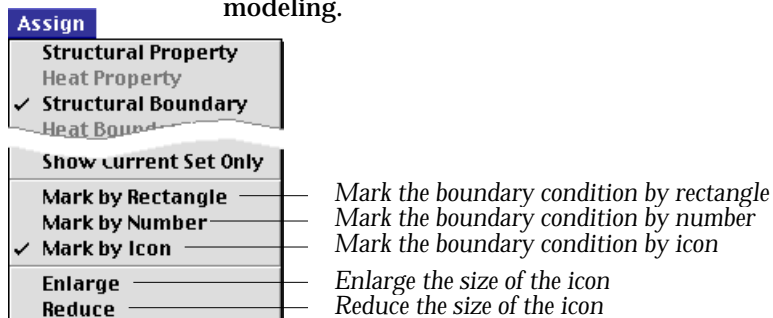
If there exists a boundary condition set identical to the one resulting from the bit ORing, that set is assigned to the node with the overlaid boundary conditions. Otherwise, a new set is created.

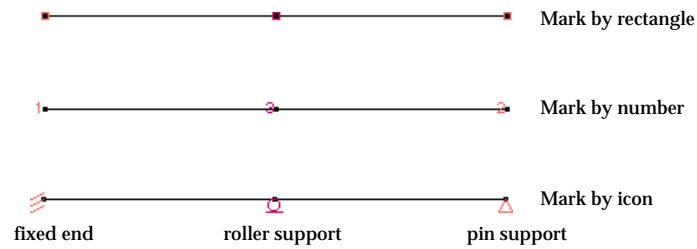


< Example of overlaying boundary conditions >

■ Representation of boundary condition assignment

Nodal assignment of structural boundary conditions is represented by icon, number or rectangle marked at the node. Choose the desired option of marking the boundary conditions from **Assign** menu. The item of the currently applied option is checked as shown below. The icon option is available only for 2-D modeling.





<Options of representing boundary conditions>

If a boundary condition set is assigned to curves or surfaces, the individual nodes on the curves or surfaces are marked. The nodes assigned with the currently active data set are highlighted in dark red, and the nodes with other sets are marked in bright red. The boundary conditions are displayed only when "Struct Boundary" dialog is on the screen. Check "Show Str. Boundary" item of **View** menu to make the boundary conditions displayed even when "Struct Boundary" dialog is not on. The size of the icon representing the boundary conditions can be enlarged or reduced by using "Enlarge" or "Reduce" item of **Assign** menu.

Heat Conduction and Seepage Boundary conditions

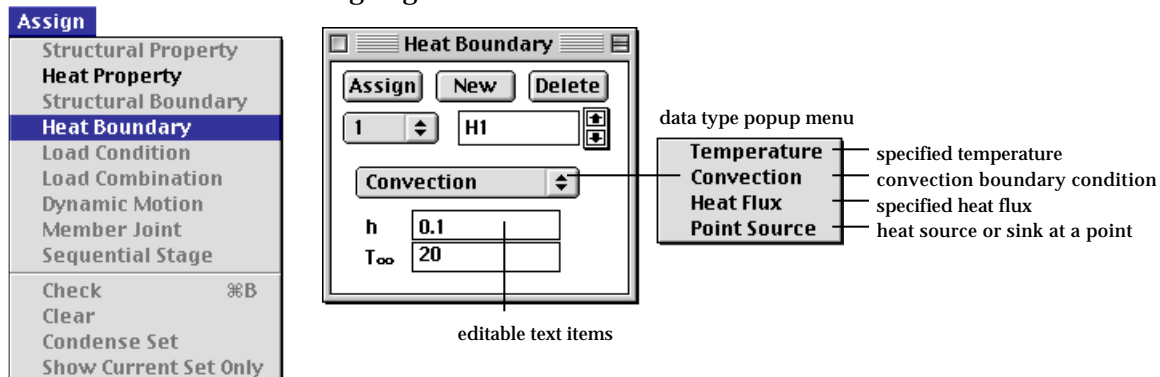
A menu item of **Assign** menu is displayed in one of the two texts "Heat Boundary" and "Seepage Boundary", depending on whether the analysis subject of the project is heat conduction analysis or seepage analysis.

The boundary conditions in heat conduction and seepage analysis can be defined and assigned by choosing the menu item.

Heat conduction boundary conditions are relevant only for heat conduction analysis. Accordingly, "Heat Boundary" item in the menu is enabled only when the analysis subject is set as one of the "Heat Analysis" item in "Problem Setup" dialog. Likewise, seepage boundary conditions are relevant only for seepage analysis. The text of the menu item displayed as "Seepage Boundary" when the analysis subject is set as one of the "Seepage Analysis" item.

Defining heat conduction boundary conditions

You must first create and define sets of heat conduction boundary conditions before assigning them on the heat conduction model.



■ Types of heat conduction boundary condition

There are a few different types of heat conduction boundary conditions as follows:









- temperature: A temperature is specified for various objects such as nodes, curves and surfaces.
- convection: The convection coefficient and the ambient temperature are assigned to an edge curve or a boundary surface.
- heat flux: The heat flux through a curve or a face is specified.
- point source: A node is defined as a heat source. Heat source is more analogous to load condition of structural analysis, and may be treated as such. In VisualFEA, however, heat source is treated as a kind of boundary condition.

The type of heat conduction boundary condition is chosen using the popup menu in “Heat Boundary” dialog. The editable text items are altered subsequent to selection of the popup menu.

■ Editable text items

Each type of heat conduction boundary conditions has different editable text items as shown in the following table. The boundary conditions are defined by entering the numerical values into these editable text items. Centigrade, Fahrenheit, or any other unit may be employed, but should be used consistently for all items.

< Editable text items of heat boundary condition >

Editable Text Items	Assignable Object Types
<p>Temperature ▾</p> <p>T <input type="text" value="28"/> — Temperature in degree centigrade, Fahrenheit, or any other unit</p>	<p> Temperature distribution can be specified at nodes, along curves (of 2-D plane and axisymmetric), or on surface meshes (of 3-D volume).</p> <p> </p>
<p>Convection ▾</p> <p>h <input type="text" value="0.5"/> — Convection coefficient</p> <p>T_∞ <input type="text" value="12.9"/> — Ambient temperature</p>	<p> Convection boundary condition can be assigned to curves (of 2-D plane or axisymmetric) or to surface meshes (of 3-D volume).</p> <p></p>
<p>Heat Flux ▾</p> <p>q <input type="text" value="10"/> — Heat flux per unit length or area</p>	<p> Heat flux can be assigned to curves (of 2-D plane or axisymmetric) or to surface meshes (of 3-D volume).</p> <p></p>
<p>Point Source ▾</p> <p>q <input type="text" value="5.72"/> — Amount of heat generation</p>	<p> Point source can be assigned only to nodes</p>

Assigning heat conduction boundary conditions

After defining a heat conduction boundary condition as a set, you can assign the set to the selected objects by clicking **Assign** button of the dialog. You can scroll the heat conduction boundary condition sets, and make any of them the currently active set, which is always applied for assignment.

■ Selecting objects to assign heat conduction boundary conditions

The assignable object types are determined by the type of boundary conditions, as shown in the previous table. Whenever the popup menu item in the dialog is altered, some selection tools are enabled and others disabled accordingly. You can switch to one of the enabled selection tools, and assign the currently active boundary condition to the objects of the relevant type.

■ Replacing previous assignment

An object can be assigned with only one heat boundary condition set. If you assign a new set to the object which has already been assigned with other set, the old set will be replaced by the new one.

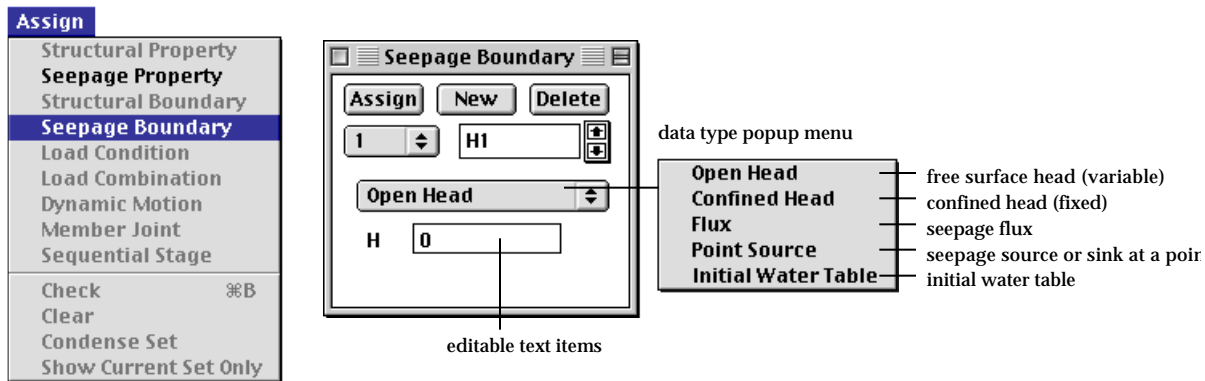
But, it should be noted that one object may encompass other objects of different type, and accordingly multiple sets may actually be assigned to a single object. For example, a temperature may be specified along a curve, while the other temperature is assigned to a node which is on the same curve. Thus, the node is assigned with two condition sets which are in conflict. In this case, the nodal assignment overrides other assignments such that the temperature of the curve is not applied to that node. In general, if one object encompasses the other object, and both objects are assigned with two different heat boundary condition sets, the assignment on the encompassed object always overrides the other assignment.

■ Representation of heat conduction boundary condition assignment

The objects assigned with heat conduction boundary conditions are distinguished from others by small square marks or by color. The nodes assigned with the current set are marked by dark red squares. The nodes assigned with other than the current set are marked by bright red squares. The curves, surface meshes, or volume meshes assigned with the current set are drawn in dark red color, and those assigned with other than the current set are drawn in bright red color.

Seepage boundary conditions

If the analysis subject is set as either "2-D Seepage", "Axisymmetric Seepage", or "3-D Seepage", the text of the menu item "Heat Boundary" changes to "Seepage Boundary". Selection of the menu item pops up the "Seepage Boundary" dialog. Seepage boundary conditions can be defined and assigned, while the dialog is on the screen,



■ Types of seepage boundary condition

There are a few different types of seepage boundary conditions as described below:

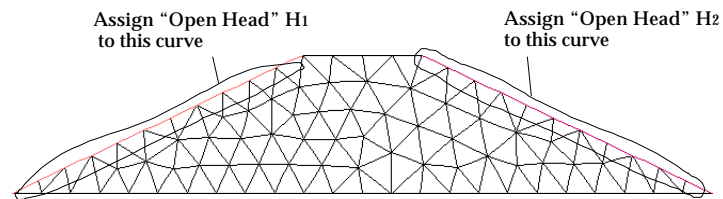
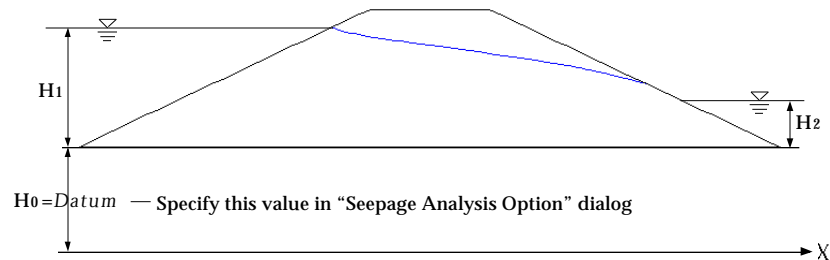
- **open head:** Open head is a type of boundary condition assigned to the domain boundary curves or surfaces such as a upstream or a downstream face of a dam. The boundaries assigned with open head include nodes under and above the specified water head. The nodes under the specified head are prescribed with the water head in the system equations. The nodes above the specified head are so called "review nodes" subject to unknown phreatic water face which are determined through iterative solution process. The open head allocated review nodes automatically to the nodes above water level.
- **confined head:** Confined head is a type of boundary condition prescribing all the assigned nodes with the specified water head .
- **flux:** seepage flow rate through unit area or length.
- **point source:** flow supplied at a point.
- **initial water table:** water table at the initial stage of transient seepage flow, or water table initially assumed in the iterative solution process.

■ Assigning open head boundary condition

Open head boundary conditions can be assigned using "Seepage Boundary" dialog by the following procedures.

- 1) Set the popup menu (Windows: dropdown list) to "Open Head".

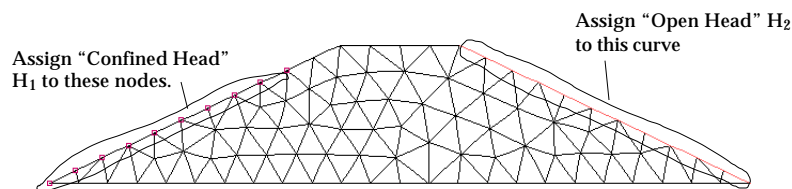
- 2) Insert the value of the water head in the editable text box with label "H".
The upstream water level H_1 and the downstream water level H_2 are the values of the open water heads in the example below.
The water levels are measured from the datum to be specified in "Analysis Option" dialog. Refer to "Setting analysis options for seepage analysis" section of Chapter 6.
- 3) Select the boundary curves or surfaces to be assigned with the open head..
Select curves for 2-D analysis and surfaces for 3-D analysis.
- 4) Assign the boundary condition to the selected objects by clicking **Assign** button of the dialog.



<Example of open head boundary conditions>

■ Assigning confined head boundary condition

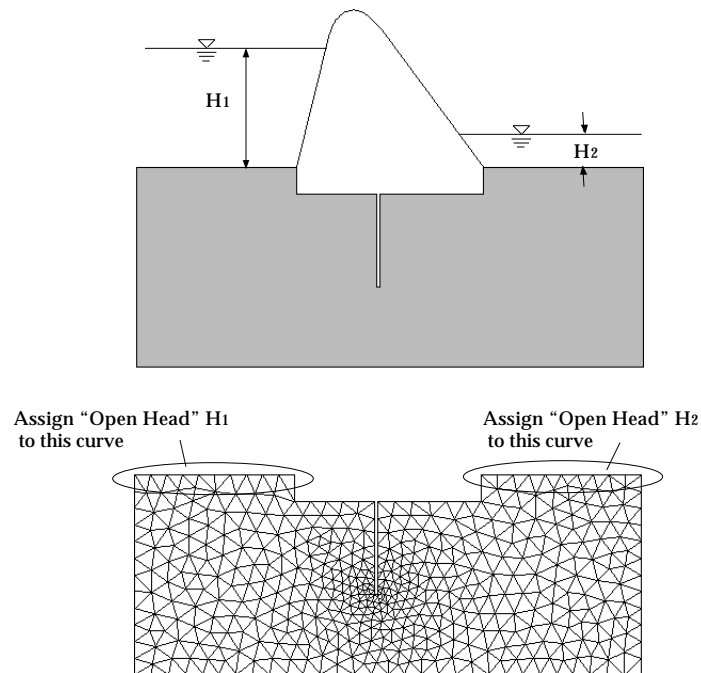
As an alternative to the open head boundary condition in the above example, the nodes of the upstream edge under the water level can be assigned with "Confined Head" H_1 , because the nodes of the upstream edge cannot form a phreatic water face above the upstream water level. However, it should be noted that the confined head should be assigned only to the nodes under the upstream water level.



<Example of a confined head boundary condition in unconfined seepage analysis>

The above example is to show that a confined boundary condition may substitute an open head boundary condition in some cases. But confined boundary conditions are commonly used to prescribe the constant head at the selected nodes or nodes along the selected curve, as shown in the example below. In the case of confined seepage analysis, only confined head conditions are applied.

In order to assign confined head boundary conditions, set the popup menu(Windows: dropdown list) to "Confined Head," and follow the similar procedure as that of open head boundary condition.



<Example of confined head conditions in confined seepage analysis>

■ Assigning flux

Water supply or drain through the boundary surfaces is represented by the boundary condition of flux. In order to assign fluxes, set the popup menu(Windows: dropdown list) to "Flux". The value of the flux should represent volume of water per unit area per unit time. The flux should be assigned to boundary curves in 2-D seepage analysis and to boundary surfaces in 3-D seepage analysis.

■ Assigning point source

Water supply or drain at a point is represented by the boundary condition of point source. In order to assign point sources, set the popup menu(Windows: dropdown list) to "Point Source". The value of the point source should represent volume of water per unit time. The point source should be assigned to nodes.

■ Defining the initial water table

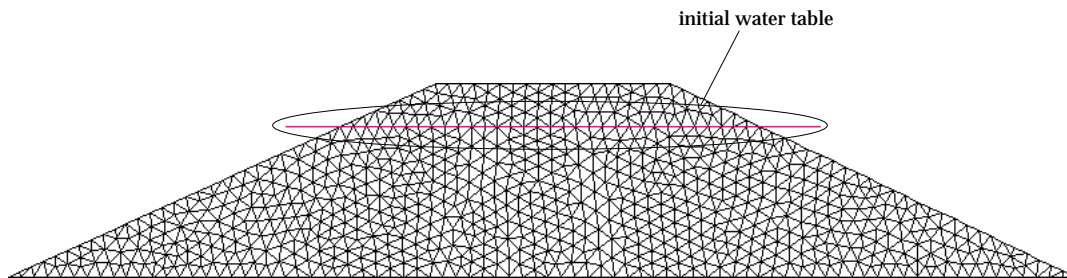
In transient analysis, the initial water table should be defined as the phreatic surface at the initial stage. But in steady state analysis, the initial water table can be considered as the initially assumed phreatic water surface, and therefore, defining the initial water table may not be essential. However, the assumption of the initial water table affects the computational efficiency of the iterative solution process, and is essentially used in obtaining the overall distribution of the negative pore water pressure above phreatic surface. The negative pore water pressure above the initial water table is set to 0.

The initial water table can be defined by the following procedure.

- 1) Create a curve or a surface to be defined as the initial water table.

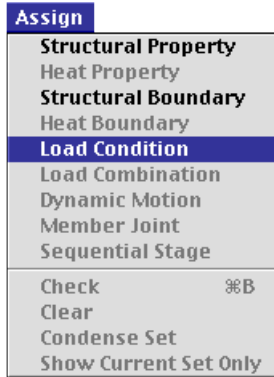
The initial water table is defined by one or more curves in 2-D seepage analysis, and one or more surface meshes in 3-D seepage analysis. The curves or surface meshes can be created as usual. If there already exist curves or surface meshes that can be defined as the initial water table, this step can be ignored.

- 2) Open "Seepage Boundary" dialog if it is not yet opened.
- 3) Set the popup menu (Windows: dropdown list) to "Initial Water Table".
- 4) Select the curves or the surface meshes to be defined as the initial water table.
- 5) Click **Assign** button of the dialog.



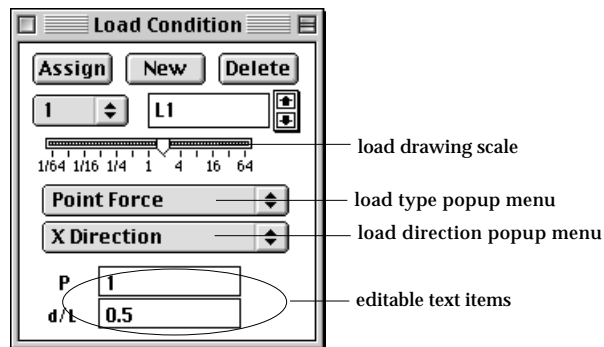
<Example of initial water table>

Load Conditions



The load condition represents the state of various forces applied to the objects in structural analysis.

In order to start assigning load conditions, choose “Load Condition” item in **Assign** menu. “Load Condition” dialog appears, and the current state of their assignment is displayed in the main window.



Defining load condition sets

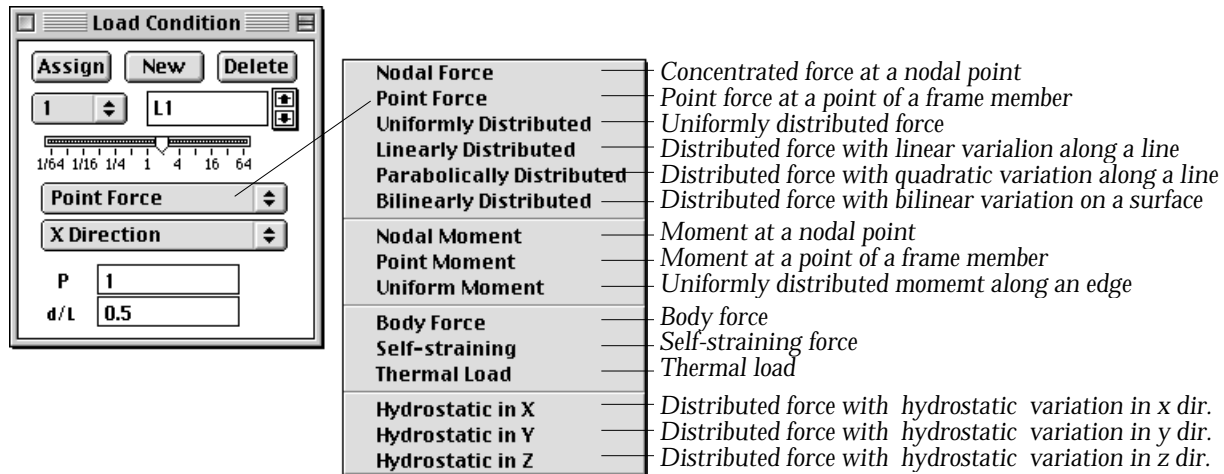
Load conditions are defined and assigned by the data unit called a load condition set. A load condition set is initially created by clicking **NEW** button of “Load Condition” dialog. If a load set is already assigned, additional assignment of the set, either to a single object or multiple objects, creates a new load set automatically by duplicating the set. Loads applied to the assigned object(s) share the new load set.

■ Data items of a load condition









A set consists of many data items including the type, the direction, the magnitude, and the position of a force. The actual data items vary depending on the type of analysis.

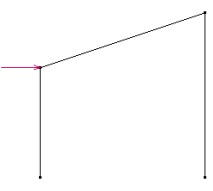
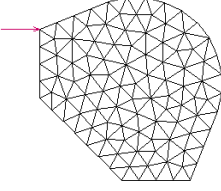
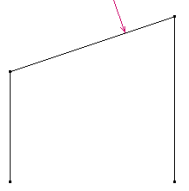
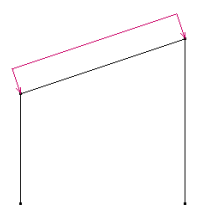
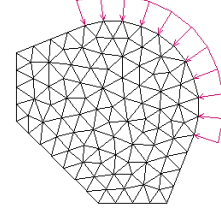
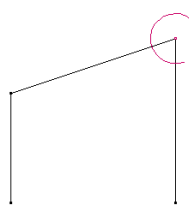
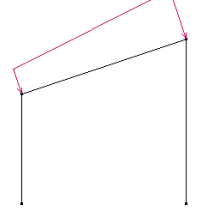
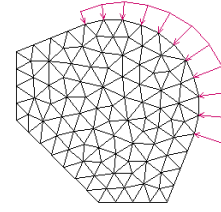
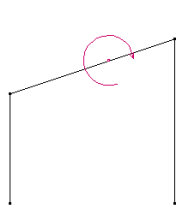
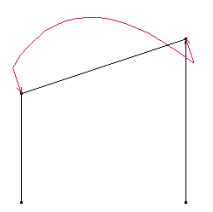
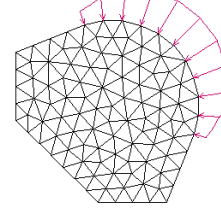
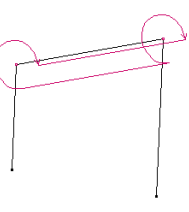
- load type: A number of load types are included in the first popup menu of “Load” dialog.
- load direction: The local directions such as normal and tangential, as well as x, y and z directions in the global coordinates are provided in the second popup menu of the dialog.
- editable text items: Magnitude, position, reference points and others are inputted in the editable text items in the dialog. The editable text items differ depending on the type of the objects.

■ Load types




There are a number of different load types as shown in the the popup menu items. Some of the menu items are enabled, while others are disabled depending on the type of analysis. For example, “Point Force” item is available only for 2-D or 3-D frames.

- nodal force: Nodal forces are applied at nodes. When this item is chosen in the popup menu, the node selection tool  is automatically activated, so that you may select nodes to which the force is applied.
- point force: Point forces are applied at a point between nodes on a frame member. This item is enabled only for 2-D and 3-D frame analysis. When this item is chosen in the popup menu, the element selection tool  is automatically activated.
- uniformly distributed: These are forces distributed uniformly over a curve or a surface. In order to apply uniform forces, you must first activate the curve selection tool , or the surface selection tool , unless one of the two tools is already in action.
- linearly distributed: These are distributed forces with linear variation along a curve. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- parabolically distributed: These are distributed forces with quadratic variation along a curve. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- bilinearly distributed: These distributed forces have bilinear variation on a surface. The force intensity at a point is determined by bilinear interpolation of the intensities at 4 corner points. When this item is chosen in the popup menu, the curve selection tool  is automatically activated.
- nodal moment: This represents a moment or a couple applied at a node. When this item is chosen in the popup menu, the node selection tool  is automatically activated.

Load Type	2-D Frame and 2-D Elasticity Example		Load Type	2-D Frame Example
Nodal Force			Point Force	
Uniform Force			Nodal Moment	
Trapeziform Force			Point Moment	
Parabolic Force			Uniform Moment	

< Examples of load types >

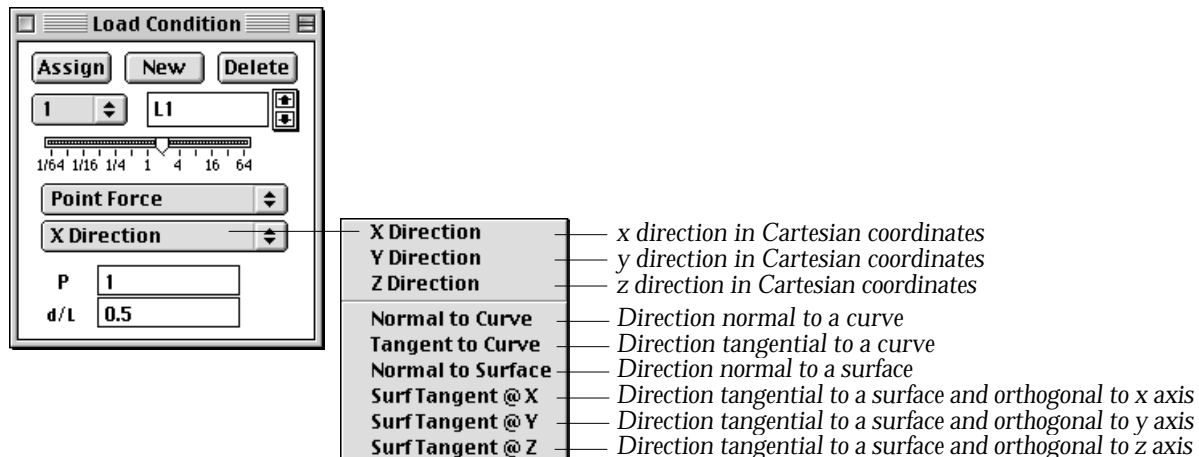
- **point moment:** This represents a moment or a couple applied at a point between nodes on a frame member. This item is enabled only for 2-D and 3-D frame analysis. When this item is chosen in the popup menu, the element selection tool  is automatically activated.
- **uniform moment:** This represents a line moment applied uniformly on a frame member or an edge of a surface.
- **body force:** This is the force due to acceleration of mass. One example of body force is the weight, or gravitational force of an object. Body force is applicable to all types of objects except slip bar and embedded bar elements.
- **self-straining force:** This is internally exerted force due to prestressing, lack of fit, and so on. Self-straining force can be applied to frame members or embedded bar elements.
- **thermal load:** Thermal load implies the temperature variation or temperature gradient which causes strains and stresses. The equivalent

nodal forces are computed from the give temperature distribution. In a coupled analysis of structure and heat conduction, the temperature distribution is not specified by the thermal load but is given by the nodal values computed from heat conduction analysis.

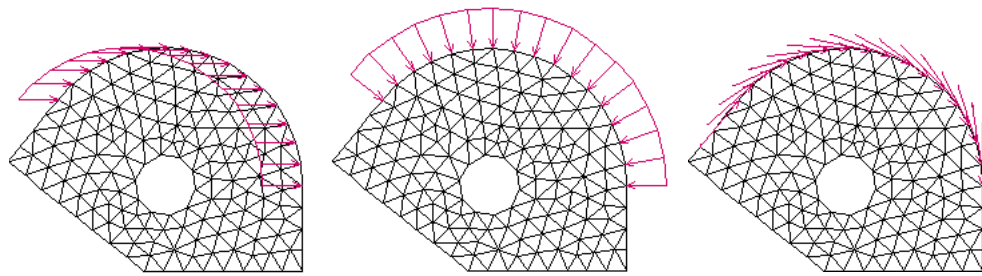
- hydrostatic in X: “Hydrostatic in X” implies linear variation in the X direction. This is to represent forces distributed on an area with intensity varying linearly in X direction and uniform in other directions.
- hydrostatic in Y: “Hydrostatic in Y” implies linear variation in the Y direction. This is to represent forces distributed on an area with intensity varying linearly in Y direction and uniform in other directions.
- hydrostatic in Z: “Hydrostatic in Z” implies linear variation in the Z direction. This is to represent forces distributed on an area with intensity varying linearly in Z direction and uniform in other directions.

■ Load direction

The direction of forces may be specified either in X, Y, and Z axis of Cartesian coordinates system or in local directions normal or tangential to curves or surfaces.



- X direction: X direction in XYZ coordinate system.
- Y direction: Y direction in XYZ coordinate system.
- Z direction: Z direction in XYZ coordinate system.
- normal to curve: Direction normal to the curve on which the force is acting.
- tangent to curve: Direction tangential to the curve on which the force is acting.



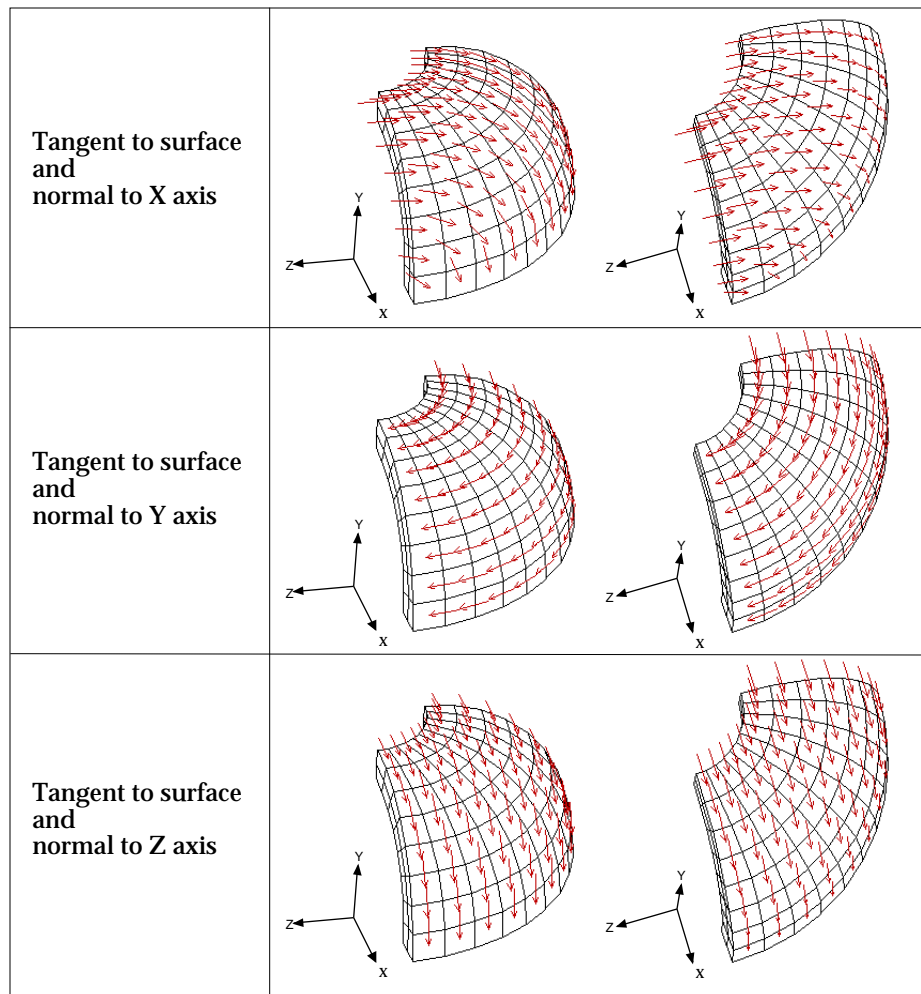
X Direction

Normal to Curve

Tangent to Curve

< Direction of a uniform load applied to a curve >

- **normal to surface:** Direction normal to the surface on which the force is acting.
- **tangent to surface and normal to X:** The direction tangential to the surface on which the force is acting, and normal to X direction. The direction tangential to a surface is not unique. The direction is constrained to be normal to X direction, so that it became unique.
- **tangent to surface and normal to Y:** The direction tangential to the surface on which the force is acting, and normal to Y direction.
- **tangent to surface and normal to Z:** The direction tangential to the surface on which the force is acting, and normal to Y direction.
- **no direction:** This item is provided only for a thermal load which does not have a direction. The load direction popup item automatically turns to this item when the current load type is set as thermal load.


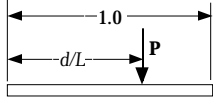
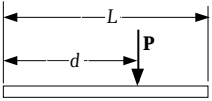




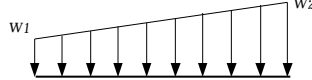

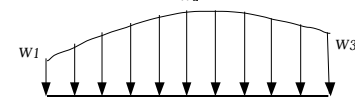

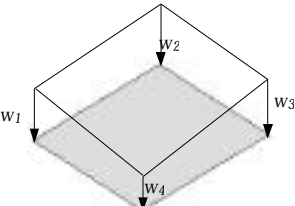



< Directions tangent to surface >



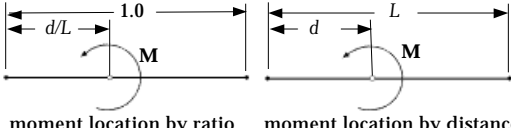


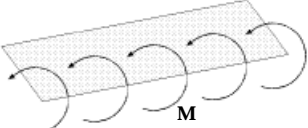






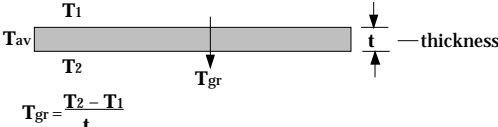


■ Editable text items

The editable text items are different depending on the type of loads as shown in the following figures. When the popup menu item of load type is selected, corresponding editable text items appear on the dialog. The numerical values related to a force are specified by these editable text items.



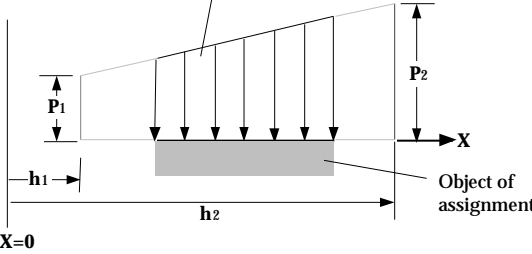



< Editable text items of load conditions >

Editable Text Items	Assignable Object Types
<p>Nodal Force ▾</p> <p>X Direction ▾</p> <p>P <input type="text" value="1"/> force magnitude</p>	<p> Nodal forces can be assigned only to nodal points. If "Nodal Force" is chosen as the type of load to assign, the node selection tool is automatically activated.</p>
<p>Point Force ▾</p> <p>X Direction ▾</p> <p>P <input type="text" value="4.0"/> force magnitude</p> <p>d/L <input type="text" value="0.7"/> force location by ratio</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>force location by ratio</p> </div> <div style="text-align: center;">  <p>force location by distance</p> </div> </div> <p>P <input type="text" value="4"/></p> <p>d <input type="text" value="3.5"/> force location by distance</p> <p>Click here to toggle d/L and d. </p>	<p> Point forces can be assigned only to a frame member. Therefore, this load type option is enabled only for 2-D or 3-D frames. Point forces can be applied either to elements (frame member) or to curves.</p> <p>The force location can be specified either by ratio(d/L) or by distance(d).</p>
<p>Uniformly Distributed ▾</p> <p>X Direction ▾</p> <p>w <input type="text" value="1"/> force intensity per unit length or per unit area</p>	<p> Uniformly distributed forces can be assigned to elements (except solid elements), to curves or to surface meshes.</p> <p> This type of force cannot be applied to solid elements.</p>
<p>Linearly Distributed ▾</p> <p>X Direction ▾</p> <p>w₁ <input type="text" value="0.8"/> force intensity at one end</p> <p>w₂ <input type="text" value="2.3"/> force intensity at the other end</p> 	<p> Linearly varying trapesiform forces can be applied only to curves. In order to assign linear varying forces to surface meshes, use "Hydrostatic in X", "Hydrostatic in Y" or "Hydrostatic in Z"</p>
<p>Parabolically Distributed ▾</p> <p>X Direction ▾</p> <p>w₁ <input type="text" value="1.0"/> force intensity at one end</p> <p>w₂ <input type="text" value="2.5"/> force intensity at mid point</p> <p>w₃ <input type="text" value="1.6"/> force intensity at the other end</p> 	<p> Quadratically varying parabolic forces can be applied only to curves or frame elements.</p>
<p>Bilinearly Distributed ▾</p> <p>X Direction ▾</p> <p>w₁ <input type="text" value="2"/> <input checked="" type="checkbox"/> w₁ at corner 1</p> <p>w₂ <input type="text" value="3"/> <input checked="" type="checkbox"/> w₂ at corner 2</p> <p>w₃ <input type="text" value="2"/> <input checked="" type="checkbox"/> w₃ at corner 3</p> <p>w₄ <input type="text" value="1"/> <input checked="" type="checkbox"/> w₄ at corner 4</p> 	<p> Bilinearly varying distributed forces can be applied only to surface meshes.</p>

< Editable text items of load conditions (continued)>

Editable Text Items	Assignable Object Types
<p>Nodal Moment ▾</p> <p>Z Axis ▾</p> <p>M <input type="text" value="1.3"/> — magnitude of moment</p> 	<p> Nodal moment can be applied to nodes of frame members. Therefore this popup menu item is enabled only for frame analysis.</p>
<p>Point Moment ▾</p> <p>Z Axis ▾</p> <p>M <input type="text" value="3.21"/> — magnitude of moment</p> <p>d/L <input type="text" value="0.7"/> — moment location by ratio</p>  <p>moment location by ratio moment location by distance</p> <p>M <input type="text" value="3.21"/></p> <p>d <input type="text" value="2.8"/> — force location by distance</p> <p>Click here to toggle d/L and d.</p>	<p> Point moments can be assigned only to a frame member. Therefore, this load type option is enabled only for 2-D or 3-D frames. Point moments can be applied either to elements (frame member) or to curves. The moment location can be specified either by ratio(d/L) or by distance(d).</p> <p></p>
<p>Uniform Moment ▾</p> <p>Z Axis ▾</p> <p>M <input type="text" value="3.21"/> — moment intensity per unit length</p> 	<p> Uniformly distributed moments can be applied to frame elements or to edges of a shell or a plate. Therefore, this type of load is valid only for 2-D frame, 3-D frame, plate bending and shell models.</p> <p></p>
<p>Body Force ▾</p> <p>X Direction ▾</p> <p>a <input type="text" value="1"/> — acceleration</p> <p>The magnitude of body force is determined by the acceleration and the unit mass which is specified as an item of the element property</p> <p>body force $F_b = W_u a$ — acceleration</p> <p>unit mass</p>	<p> Body forces can be assigned to elements, curves(for frames only), surface meshes(for planar elasticity, plate and shell), and volume meshes(for 3-D solid)</p> <p></p> <p></p> <p></p>
<p>Thermal Load ▾</p> <p>Tangent to Curve ▾</p> <p>T_{av} <input type="text" value="24.5"/> — average temperature</p> <p>T_{gr} <input type="text" value="2.3"/> — temperature gradient</p>  <p>T_{av} $T_{gr} = \frac{T_2 - T_1}{t}$ — thickness</p>	<p> Thermal loads can be applied to frame elements and curves. This type of load is valid only for 2-D and 3-D frame.</p> <p></p>

< Editable text items of load conditions (continued)>

Editable Text Items	Assignable Object Types
<p>Self-straining ▾</p> <p>Tangent to Curve ▾</p> <p>P <input type="text" value="1"/> — magnitude of straining force</p> <p style="text-align: center;">P ←————→ P</p>	<p> Self-straining forces can be applied to elements and curves, which should be members of truss or frame structures, or embedded bars within solids.</p> <p></p>
<p>Hydrostatic in X ▾</p> <p>Normal to Curve ▾</p> <p>P₁ <input type="text" value="1.2"/> — force intensity w at X=h₁</p> <p>P₂ <input type="text" value="2.5"/> — force intensity w at X=h₂</p> <p>h₁ <input type="text" value="-2"/> — X coordinate at which w=P₁</p> <p>h₂ <input type="text" value="4.1"/> — X coordinate at which w=P₂</p> <p>Hydrostatic in Y ▾</p> <p>X Direction ▾</p> <p>P₁ <input type="text" value="-3.8"/> — force intensity w at Y=h₁</p> <p>P₂ <input type="text" value="3.4"/> — force intensity w at Y=h₂</p> <p>h₁ <input type="text" value="2.1"/> — Y coordinate at which w=P₁</p> <p>h₂ <input type="text" value="9.84"/> — Y coordinate at which w=P₂</p> <p>Hydrostatic in Z ▾</p> <p>SurfTangent @X ▾</p> <p>P₁ <input type="text" value="5.3"/> — force intensity w at Z=h₁</p> <p>P₂ <input type="text" value="-1.5"/> — force intensity w at Z=h₂</p> <p>h₁ <input type="text" value="1.234"/> — Z coordinate at which w=P₁</p> <p>h₂ <input type="text" value="0"/> — Z coordinate at which w=P₂</p> <div style="text-align: center;"> <p>“Hydrostatic in X”</p>  <p>Object of assignment</p> </div>	<p> “Hydrostatic in X”, “Hydrostatic in Y”, and “Hydrostatic in Z” can be applied to elements (surface elements only), curves and surface meshes.</p> <p></p> <p></p>

Defining load condition sets for dynamic analysis

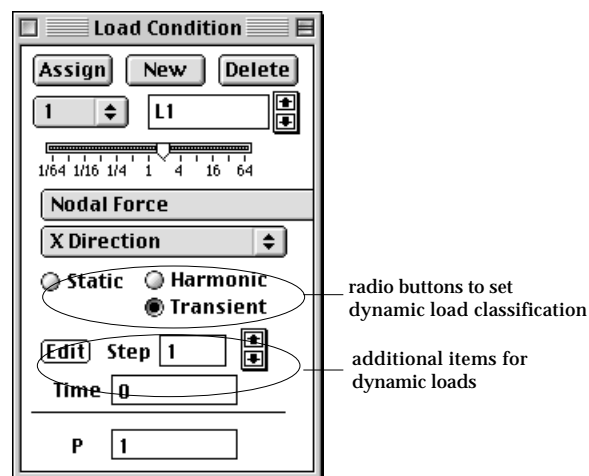
For dynamic analysis, load conditions may or may not have time dependent characteristics. Therefore, assignment of dynamic load conditions involves more steps than that of static load conditions, as described below.

■ Setting the time dependency

The dynamic load condition is classified further its time dependency as follows.

- static: static load condition independent of time.
- harmonic: time-dependent load condition with periodically changing magnitude expressed by a sinusoidal equation.
- transient: time-dependent load condition represented in multiple time steps.

Thus, "Load Condition" dialog for dynamic analysis has radio buttons, "Static", "Harmonic" and "Transient." Time dependency of a dynamic load is set by turning on the corresponding radio button on "Load Condition" dialog.



As you alter the time dependency, you will notice the changing items below the radio buttons. Time dependent attributes of dynamic loads are defined and modified using these items.

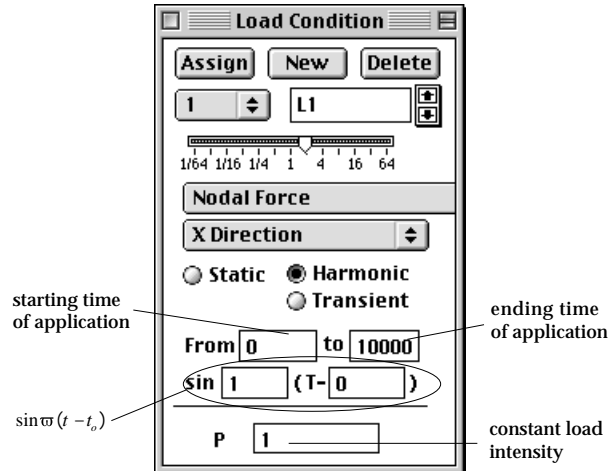
■ Defining static load for dynamic analysis

Static load condition is assumed to have constant magnitude, direction and position throughout the whole duration of analysis, and can be defined and assigned in the same way as for static analysis. "Static" radio button should be turned on to start defining static loads.

■ Defining harmonic load for dynamic analysis

A harmonic load has periodic characteristics with changing magnitude expressed by the sinusoidal function $\sin \omega(t - t_o)$. The intensity of the load acting at a

specific time is obtained by multiplying the function by the constant load intensity given at the bottom of the dialog.

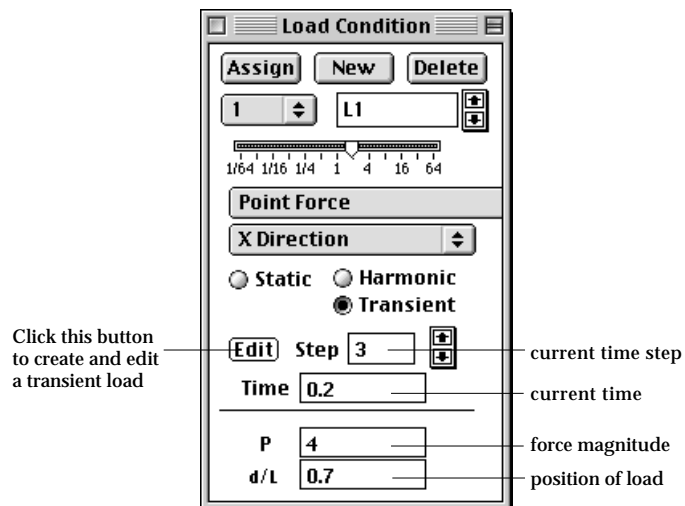


■ Defining transient load for dynamic analysis

A transient load is defined by a series of values at a number of time steps, which can be created and edited by the following steps.

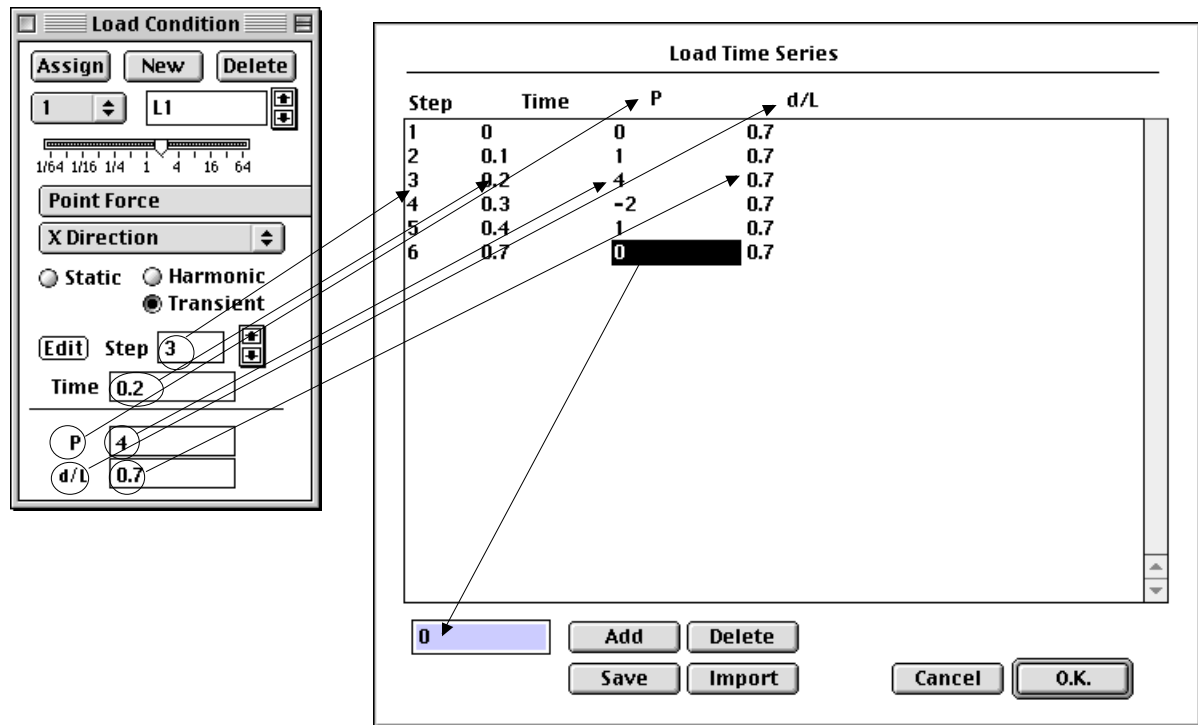
- 1) Turn on "Transient" radio button in "Load Condition" dialog.

Then, items below the radio buttons take on the appearance shown below. They include the current step number and the current time. The current time step can be edited by directly changing the number in the box, or by scrolling the number.



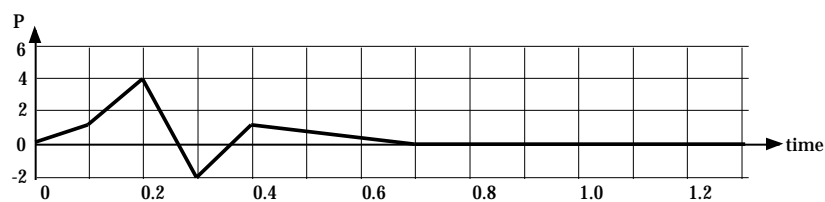
- 2) Click **Edit** button.

Then, "Load Time Series" dialog pops up. The dialog produces editable data items for each time step you create. The data items are equivalent to the data items in "Load Condition" dialog.



- 3) Click **Add** button to add a time step.
A line of data items will be created for the new time step. In order to delete a time step, click **Delete** button after selecting the corresponding data line.
- 4) Select a line or a cell to edit.
Then, the data on the line or in the cell will be reflected in the editable text boxes at the bottom of the dialog
- 5) Edit the text in the editable text box.
As you edit the text in a text box, it will immediately be echoed in the actual data line.
- 6) Click **O.K.** button after completing creation of time steps.
"Load Time Series" closes and the data from the dialog are automatically reflected in "Load Condition" dialog.

The data on the last line are assumed to be maintained up to the last time step in the analysis. Thus the data shown in the above example dialog will create the following time series data.



< An example of transient load >

Assigning load condition sets

A load condition set is defined as explained in the previous section. The next step is to assign the set to a certain object. You can assign the currently active set to selected objects by clicking **Assign** button of the dialog.

■ Selecting objects to assign load conditions

The load condition sets may be applied to various objects. However, assignable object types are limited and determined by the type of load conditions, as indicated in the previous figures of “Editable text items”. Only the selection tools of assignable objects are enabled and others are disabled, whenever the type of the current load set is altered.

■ Assigning a load condition set to multiple objects

You can apply the current load condition set to more than one object. However, once the set is assigned, extra new load condition sets are automatically generated so that each of the selected objects is assigned with an independent set which has the same characteristics as that of the current set. So, you will find that the number of load condition sets is increased as much as the number of assigned objects minus one.

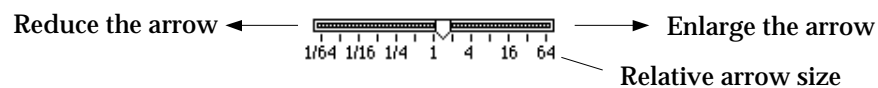
■ Assigning multiple load condition sets to a single object

One object may be assigned with more than one load condition sets. But there is a possibility that you assign, by mistake, an identical load conditions to the same object more than once. In order to prevent such a trouble, VisualFEA does not allow assigning the same load condition to one object twice or more.

■ Representation of load assignment

The assigned forces, except body forces, are represented by arrows indicating the point and the direction of their application. Body forces are indicated by x mark, instead of arrow, at the centroid of each element.

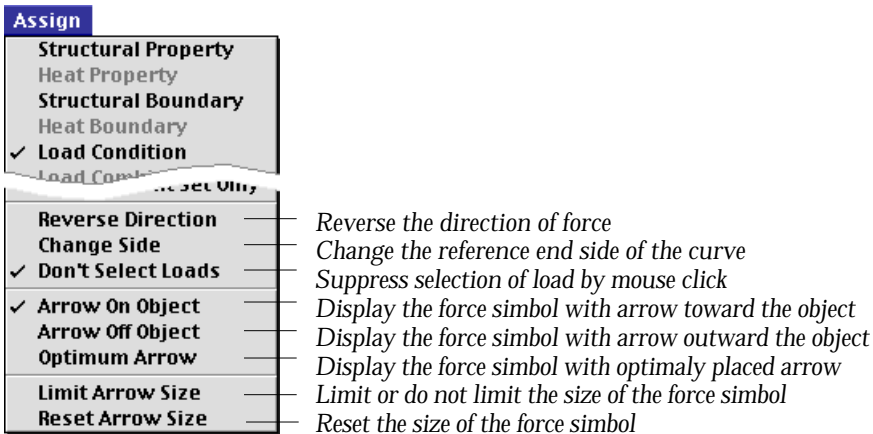
The size of arrow can be adjusted using the slide bar of load drawing scale. The slide bar has relative scale of arrow size. The arrow size can be lengthened by moving the thumb of the bar to the right, and shortened by moving the thumb to the left.



< Load drawing scale >

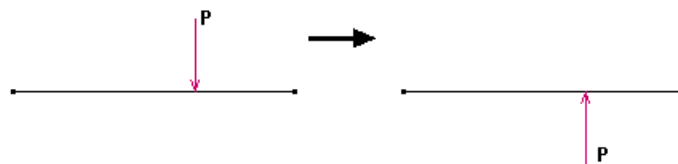
Other functions related to assigning load conditions

Load conditions can be defined and assigned while the load condition dialog appears on the screen. At this time, the few menu items related with load conditions are attached to **Assign** menu as shown below.



■ Reversing the force direction

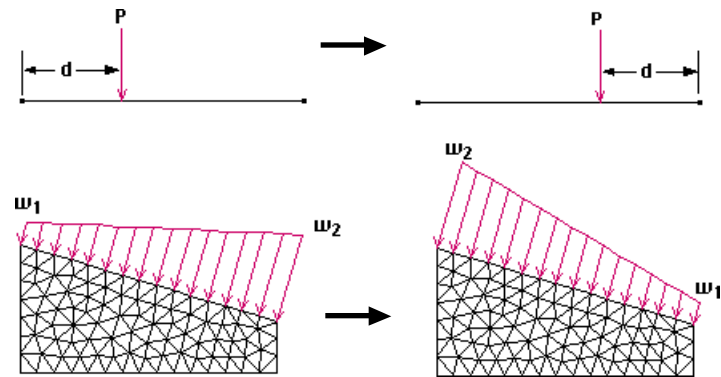
The direction of the force can be reversed by simply choosing the item “Reverse Direction” from **Assign** menu. The force direction of the current load condition set is reversed, and accordingly, the corresponding editable text items of “Load Condition” dialog is altered with reversed sign. If a load condition set is applied to multiple objects, and “Reverse Direction” command is issued immediately, then the direction of all the lastly assigned loads will be reversed.



< Result of “Reverse Direction” command >

■ Exchanging the reference end of the curve

The location of point force is represented by the distance from one end of a curve or a frame element. In this case, the starting point of distance is the reference end of the curve or the element. The magnitude of trapezoidal force is represented by two load intensity W_1 and W_2 . The end point with W_1 may be the reference point. It is sometimes necessary to switch the reference point. This can be done simply choosing the menu item “Change Side” from **Assign** menu.



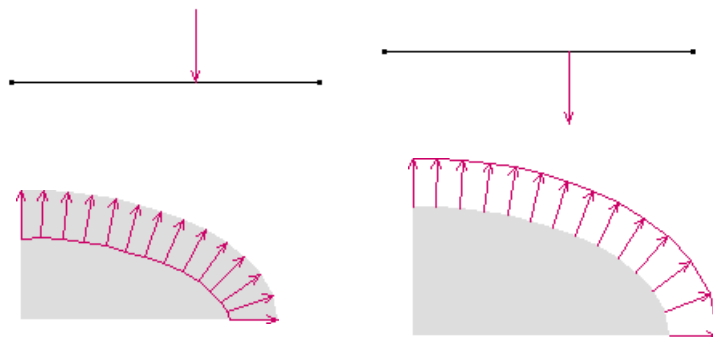
< Result of “Change Side” command>

■ Suppressing load selection

Loads can be selected simply by clicking their symbols. You may sometimes be annoyed by unintentionally selecting loads instead of other objects which you actually want to select. This may happen especially when many load symbols are densely located at the point of mouse click. Under such situation, it is convenient to suppress the selection of loads temporarily. Suppressing and desuppressing can be toggled by choosing “Don’t Select Loads” item from menu. The current state is indicated by the check mark in front of the menu item.

■ Changing the placement of force symbol

Forces are represented by arrows. These arrows may be placed in two different ways: one with arrows on the object, and the other with arrows off the object. The placement of the arrow can be switched by selecting either “Arrow on Object” or “Arrow off Object” from **Assign** menu.



“Arrow on Object”

“Arrow off Object”

<Placement of force symbol>

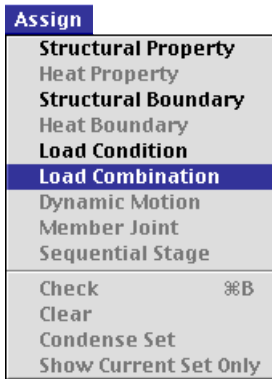
■ Limiting the size of force symbol

The size of the arrow representing a force is determined to be approximately proportional to the magnitude of the force. When there are big differences in the magnitudes of the forces, some of arrows become too large to be appropriately drawn on the screen. For such circumstances, it is necessary to limit the size of arrows. This can be done by selecting “Limit Arrow Size” item from **Assign** menu.

■ Resetting the size of force symbol

As you continue assigning load conditions, the size of arrow may become inappropriate, even though the load drawing scale can be adjusted using the slide bar of "Load Condition" dialog. In such case, it is desirable to reset the default size of the arrow by selecting “Reset Arrow Size” item from **Assign** menu.

Load Combination

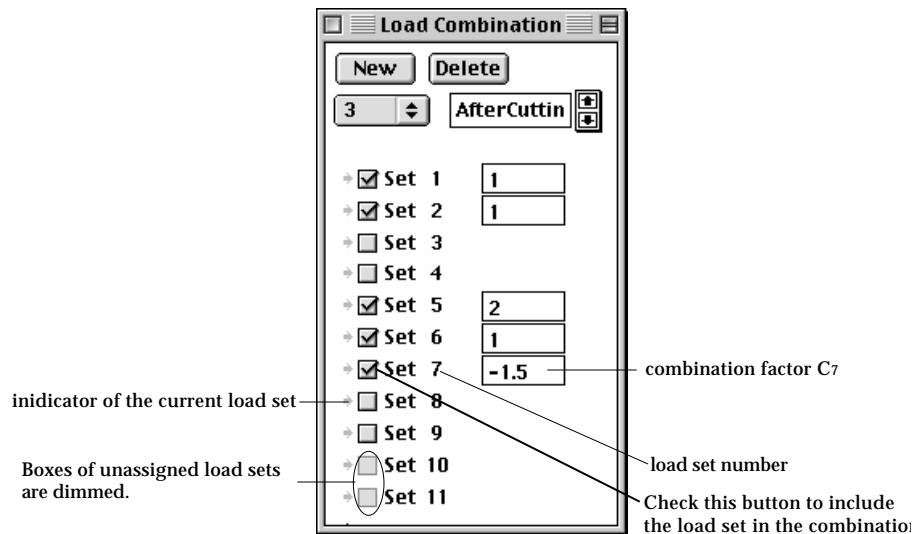


Multiple sets of analysis results can be obtained at once by solving a model with multiple load cases. Each load case is defined by linear combination of the load sets. A force vector \mathbf{F}_r for the system equations is obtained by linear combination of m force vectors, $\mathbf{F}_1, \dots, \mathbf{F}_m$, which may be expressed by the following equation,

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

As many load cases as desired can be formed by various combinations of the load sets, and can be used for independent analyses. Instead of solving the equations with each one of the load cases, the program solves all the cases at once, and thus saves the computing time significantly. The results from various load cases can be combined further with desired factors to form a new sets of analysis results.

The function of load combination is valid only for linear static analysis.



■ Defining load combinations

Load combination is meaningful only when there are 2 or more load condition sets applied to the model. A load case is defined by combining all or parts of the load sets with certain combination factors by the following factors.

- 1) Choose "Load Combination" item from **Assign** menu.
Then, "Load Combination" dialog appears on the screen.
- 2) Create or delete a combination set using **New** or **Delete** button in the dialog.

The usage of the dialog is similar to that of assignment dialogs except that there is no **Assign** button.

- 3) Check the load sets to be include in the combination.

Click the check boxes to turn on or off the inclusion of load sets in the combination. Only the load sets applied to the model can be included in a combination.

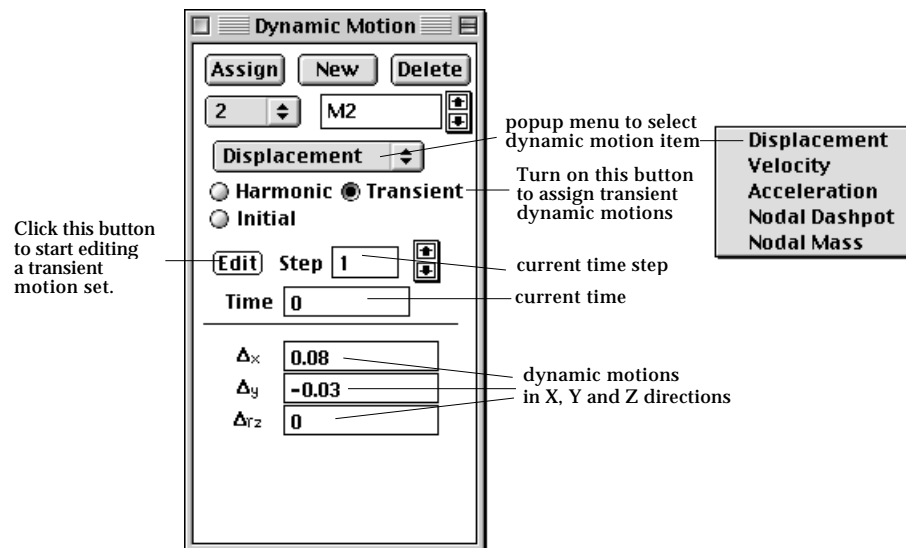
- 4) Insert the combination factors c_i in the editable text boxes.

There appears an editable text box to the right of the included load set text. Insert the text of the combination factor in the box. The factor can be either positive or negative.

You may define as many load cases as necessary. However, it should be considered that further combination of analysis results is also possible after solving the model with multiple load combinations.

■ Handling load condition sets in "Load Combination" dialog

The "Load Combination" dialog may be displayed along with the "Load Condition" dialog as shown in the figure below. You may change the current load set by clicking the corresponding indicator in the "Load Combination" dialog. Alternatively, you may also change the highlighted indicator by selecting a popup menu(Windows:dropdown list) item in the "Load Condition" dialog.

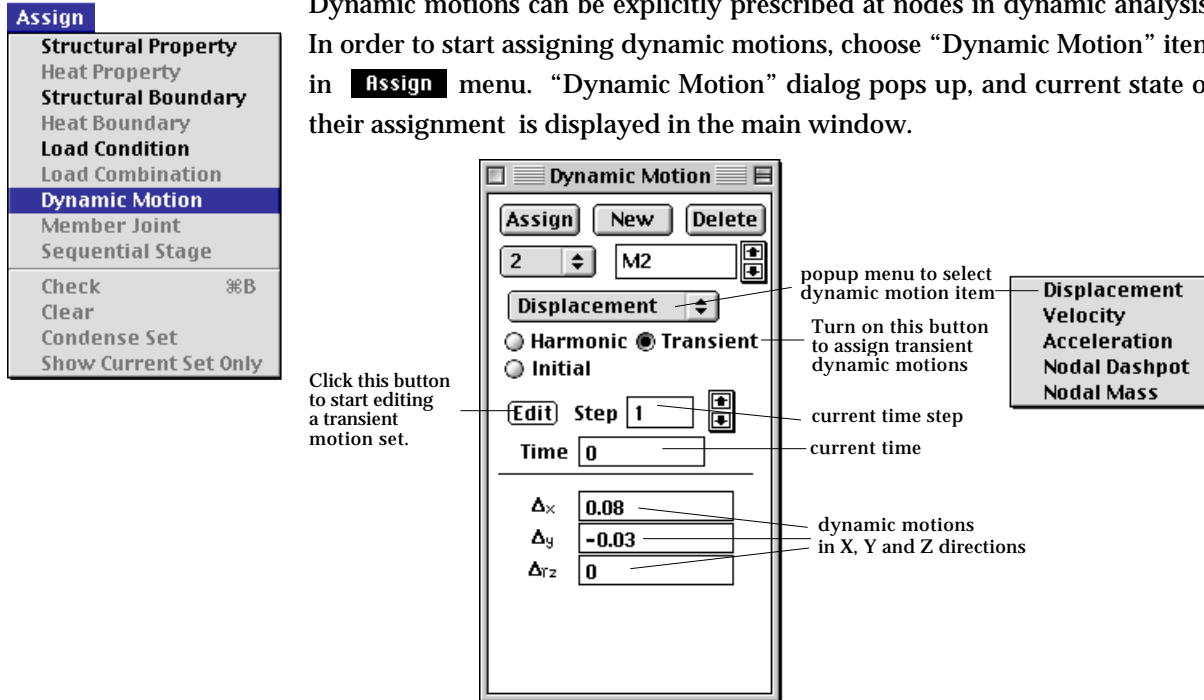


<"Load Combination" dialog displayed along with "Load Condition" dialog>

■ Combining analysis results of multiple load conditions

The analysis results with various load combinations can be combined further as described in Chapter 7.

Dynamic Motions



Dynamic motions can be applied to fixed degrees of freedom as well as to free d.o.f. However, application of motion to fixed d.o.f. tends to increase the solution time.

Defining and assigning dynamic motion sets

Dynamic motions can be defined and assigned by the unit of data set similar to load condition set. A dynamic motion set is initially created by clicking **New** button of "Dynamic Motion" dialog. If a dynamic motion set is already assigned, additional assignment of the set, either to a single object or multiple objects, creates a new set automatically by duplicating the original one.

The popup menu of "Dynamic Motion" dialog has the following items:

- "Displacement": transnational displacements assigned at nodes.
- "Velocity": transnational velocity assigned at nodes.
- "Acceleration": transnational acceleration assigned at nodes.
- "Nodal Dashpot": damping dashpot attached at nodes.
- "Nodal Mass": mass assigned to nodes.

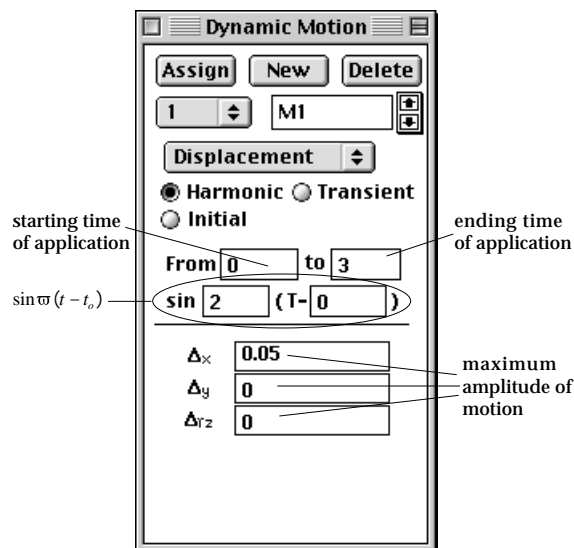
The top 3 items correspond to types of dynamic motion and the other two to dynamic resistance, which is described in the next section.

■ Setting the time dependency of dynamic motion

The time dependency of dynamic motion can be set as one of the 3 types, harmonic transient and initial types. In order to set the time dependency, click the corresponding radio button, "Harmonic", "Transient". or "Initial" in "Dynamic Motion" dialog. As you alter the option of time dependency, you will notice the changing items below the radio buttons. Time dependent variation of dynamic motions is defined and modified using these items.

■ Defining harmonic motions

A harmonic motion has periodic characteristics with changing magnitude expressed by a sinusoidal function $\sin \omega(t - t_o)$. The dynamic motion acting at a specific time is obtained by multiplying the function by the amplitude of motion given at the bottom of the dialog.



Click "Harmonic" radio button to define a set of harmonic motions. The amplitudes of the motion components are defined by inserting the values in the editable text boxes of motion components.

■ Defining transient motions

A transient motion is defined by a series of values at a number of time steps, which can be created and edited by the following steps.

- 1) Select an item of dynamic motion from popup menu in "Dynamic Motion" dialog.
One of "Displacement", "Velocity" and "Acceleration" items should be selected.
- 2) Turn on "Transient" radio button in the dialog.
Then, the items below the radio buttons take on the appearance shown at the

beginning of this section ("Dynamic Motion"). They include the current step number and the current time. The current time step can be edited by directly changing the number in the box, or by scrolling the number.

- 3) Click **Edit** button.

Then, "Dynamic Motion Series" dialog pops up. The dialog produces editable data items for each time step you create. The data items are equivalent to the data items in "Dynamic Motion" dialog.

Dynamic Motion Series

Step	Time	Dx	Dy	Dz
6	1.2	0.02	-0.03	0
7	1.3	0.03	-0.02	0
8	1.4	0.05	0	0
9	1.5	0.03	0.02	0
10	1.6	0.01	0.03	0
11	1.8	-0.01	0.05	0
12	2	-0.03	0.06	0
13	2.1	-0.06	0.05	0
14	2.2	-0.02	0.02	0
15	2.3	0.01	0	0
16	2.5	0.04	-0.02	0
17	2.6	0.07	-0.03	0
18	2.7	0.05	-0.01	0
19	2.8	0.03	-0.02	0
20	2.9	0.06	-0.04	0
21	3	0.08	-0.03	0
22	3.2	0.05	-0.01	0
23	3.3	0.02	0.01	0
24	3.4	-0.02	0.03	0
25	3.5	-0.05	0.05	0

- 4) Click **Add** button to add a time step.

A line of data items will be created for the new time step. In order to delete a time step, click **Delete** button after selecting the corresponding data line.

- 5) Select a line or a cell to edit.

Then, the data on the line or in the cell will be reflected in the editable text boxes at the bottom of the dialog

- 6) Edit the text in the editable text box.

As you edit the text in a text box, it will immediately be echoed in the actual data line.

- 7) Click **O.K.** button after completing creation of time steps.





"Dynamic Motion Series" closes and the data from the dialog are automatically reflected in "Dynamic Motion" dialog.

The data on the last line are assumed to be maintained up to the last time step in the analysis.

■ Defining initial motions

Initial motions are used to define the initial state of the analysis at time 0. The type of initial motion is either displacement, velocity or acceleration. Click "Initial" radio button of the "Dynamic Motion" dialog to define and assign initial motions.

■ Assigning dynamic motions

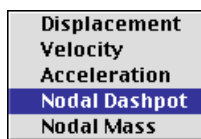
Dynamic motions can be assigned to various types of objects including nodes, curves, surface meshes or volume meshes. However, they are eventually assigned to nodes contained in the assigned objects. You should first choose the selection tool, , ,  or  in order to assign the dynamic motion sets to objects of desired type.

Defining and assigning nodal dynamic properties

As described in the previous section, the popup menu of "Dynamic Motion" dialog has items other than dynamic motion itself. They are concerned with dynamic properties assigned at nodal points. "Nodal Dashpot" is to assign damping characteristics to nodes, and "Nodal Mass" is to assign concentrated mass to nodes. A nodal dashpot is a damper decelerating nodal movement, and nodal mass carries inertia force conceptually concentrated at nodal points.

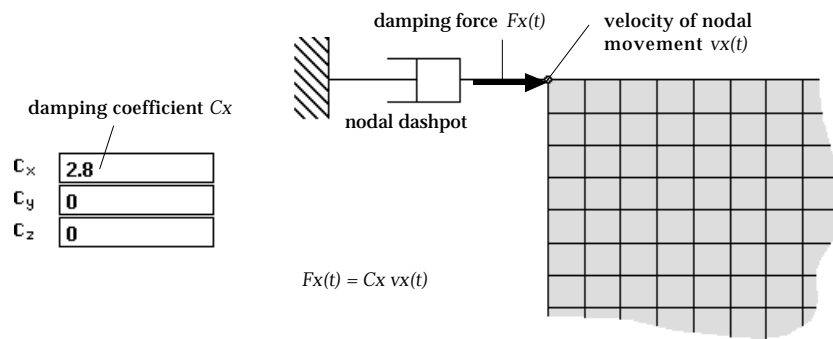
■ Nodal dashpot

Nodal dashpot represents a damper attached to a node. A nodal dashpot can be defined and assigned by the following steps:



- 1) Select "Nodal Dashpot" item from popup menu in "Dynamic Motion" dialog.
The editable text items on the dialog turn into states for defining a damping coefficient for each nodal d.o.f.
- 2) Insert values into the editable text items.
A damping coefficient is specified independently for each nodal d.o.f. For example, C_x , C_y , and C_z are damping coefficients for 3 nodal d.o.f in 3-D solid case.
- 3) Select nodes or objects containing the nodes to assign the nodal dashpot..
- 4) Click **Assign** button in the dialog.

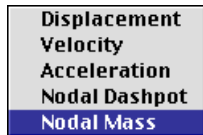
The nodes assigned with the dashpot are marked by red rectangle.



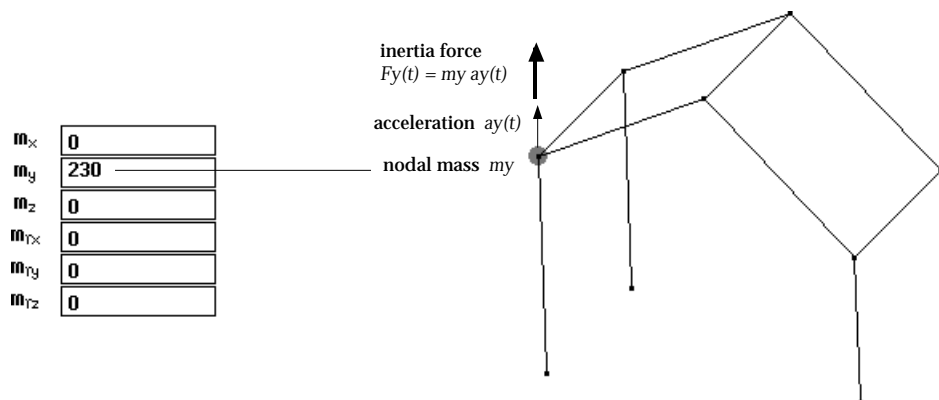
<Conceptual diagram of nodal dashpot>

■ Nodal mass

A nodal mass is a mass conceptually concentrated at a node, which makes the nodal movement produce inertia force or moment. Nodal mass can be defined and assigned by the following steps:

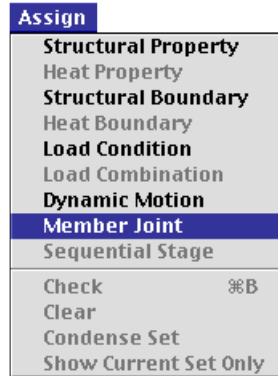


- 1) Select "Nodal Mass" item from popup menu in "Dynamic Motion" dialog.
The editable text items on the dialog turn into the state to get nodal mass components defined for each nodal d.o.f.
- 2) Insert values into the editable text items.
The conceptual nodal mass is specified independently for each nodal d.o.f. For example, m_x , m_y , m_z , m_{rx} , m_{ry} , m_{rz} are mass components for 6 nodal d.o.f in 3-D frame model. The latter 3 are rotary inertias.
- 3) Select nodes or objects containing the nodes to assign the nodal mass..
- 4) Click **Assign** button in the dialog.
The nodes assigned with the nodal mass are marked by red rectangle.



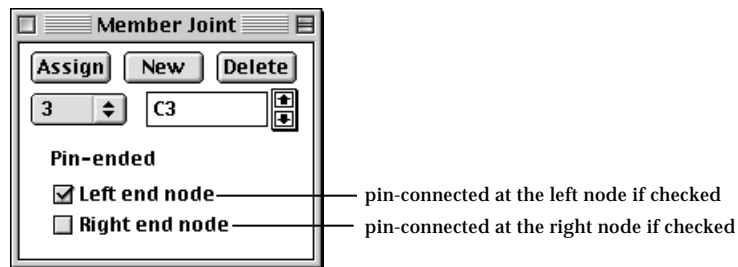
<Conceptual diagram of nodal mass>

Frame Member Joint Conditions



Frame member joint conditions are meaningful only for 2-D and 3-D frame analysis. Frame members are basically connected rigidly with other members at their joints. But some of them may be pin-jointed. The state of member connection can be defined as member joint condition sets and assigned to corresponding members.

In order to start assigning member joint conditions, choose “Element Joint” item from **Assign** menu. Then, “Member Joint” dialog box appears on the screen.

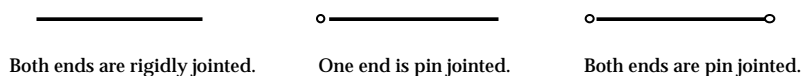


Defining frame member joint conditions

The frame member joint conditions are defined as a data set using “Member Joint” dialog. There are only two data items, each of which represents the joint condition at one end of a frame member. It is assumed that each frame member has only two end nodes: the left and the right end node. The check mark of the check box in front of “Left end node” or “Right end node” indicates that the corresponding node is pin-jointed. The check mark is turned on and off by clicking the check box. In order to define a truss member joint, turn on both check boxes. Unchecking both of the check box makes the member joint condition with both ends rigidly jointed. Defining such a joint condition is redundant, because all the frame members are originally assumed to be rigidly jointed at both of their ends.

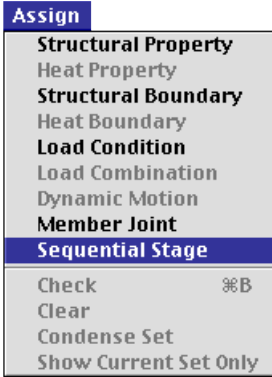
Assigning frame member joint conditions

In order to assign the currently active member joint condition set, first select the member, and click **Assign** button of the dialog. The member will get the joint condition defined by the set, and will be marked in red colors. The pinned joint is also marked in the drawing of the member as shown below.



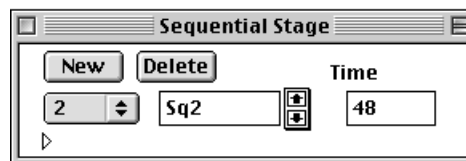
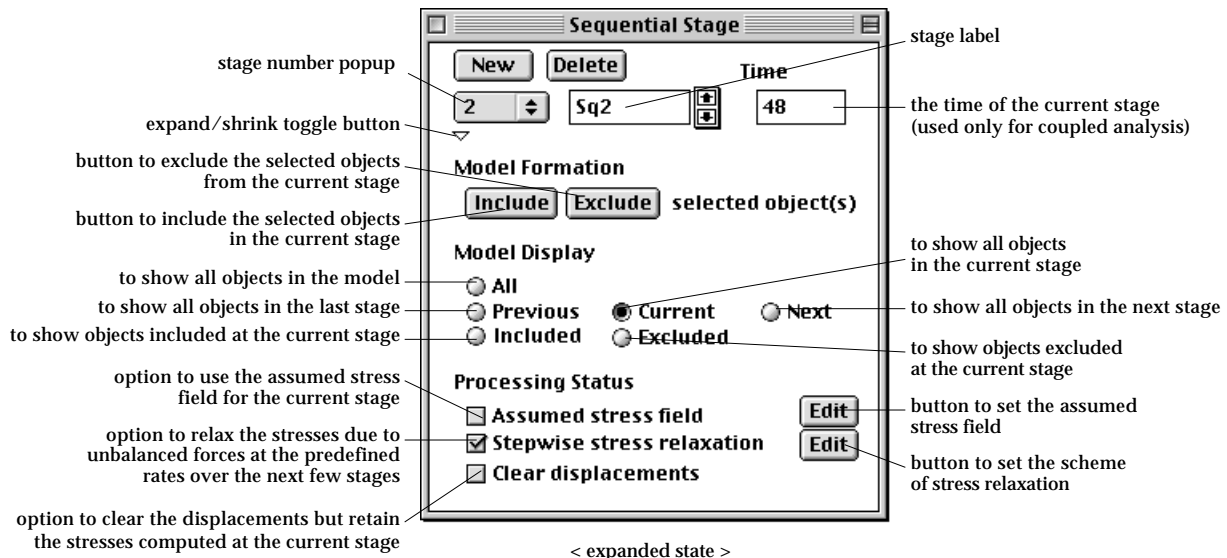
<Representation of member connectivity>

Sequentially Staged Modeling













Sequentially staged modeling is devised for analysis of objects with changing geometry, material properties and load conditions over a given period of time. Such a modeling is useful in analyzing structures like dams, tunnels and bridges under construction. The sequentially staged modeling option is turned on by checking the "Sequential" button of the "Project Setup" dialog. (Refer to "Project and File" section of Chapter 1.) The menu item, "Sequential Stage" of **Assign** menu is enabled when this modeling option is turned on.

The "Sequential Stage" dialog can be toggled between the expanded and shrunk states using the toggle button. The dialog is in the expanded state, and turns to the shrunk state automatically when an assignment dialog (such as "Property" or "Load Condition" dialog) is opened.



< shrunk state >

In order to start defining and assigning the sequential stages, choose "Sequential Stage" item from **Assign** menu. Then, "Sequential Stage" dialog opens as shown below. The model of each stage is created by using this dialog. Usage of the dialog items is explained below.

- | | |
|---|--|
|  | Click this button to create a new stage and display the state of the new stage. The new stage model initially inherit its contents from the last stage model or from the base model. |
|  | The current stage is deleted by clicking this button. The next or the previous stage becomes the current stage. |
|  | The current stage number is displayed as a popup menu(Windows: drop-down list) item. You may move to the desired stage using this popup menu (Windows: drop-down list). |
|  | The label of the current data set. You may label a data set by entering a character string in the text box. |
|  | This button is used to scroll up the stage. The current stage is scrolled up by this button. Clicking this button once reduces the current stage number by one. |
|  | This button is used to scroll down the stage. The current stage is scrolled down by this button. Clicking this button once increases the current stage number by one. |
| Time
 | The point of time for the current stage. This item is used only for structural analysis coupled with transient seepage -analysis. |
|  | Expand/shrink toggle button. Click this button to expand or to shrink the dialog. |
|  | Click this button to include the selected objects into the current stage model. If the selected objects are already included in the current stage model, this button is dimmed. |
|  | Click this button to exclude the selected objects from the current stage model. If the selected objects are not included in the current stage model, this button is dimmed. |
| <input checked="" type="radio"/> All | If this radio button is on, all the objects, i.e. the objects of the base model are displayed. Click this button to show all objects of the base model independent of the stages. |
| <input type="radio"/> Previous | Click this button to display the previous stage model. |
| <input type="radio"/> Current | Click this button to display the current stage model. |
| <input type="radio"/> Next | Click this button to display the next stage model. |
| <input type="radio"/> Included | Click this button to display the objects included at the current stage, i.e., the objects which are not in the last, but in the current stage model. |
| <input type="radio"/> Excluded | Click this button to display the objects excluded at the current stage, i.e., the objects which are in the last, but not in the current stage model. |

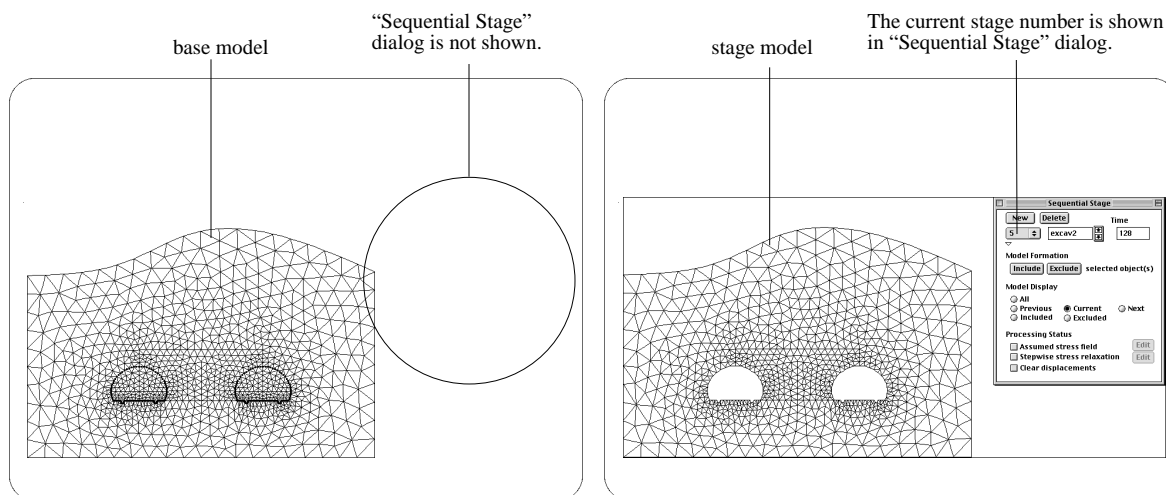
Concept of sequentially staged modeling

Stage is a core concept of the sequentially staged modeling. This section describes the finite element procedures based on the stages, and the inter-relation of analysis results and modeling data between stages.

■ The base model and the stage models

A sequentially staged model consists of 2 or more stages. Each stage has all ingredients of finite element analysis by itself and is modeled to represent the actual situation with changing geometry, properties, and load conditions. The finite element analysis model for a stage is termed here as "stage model", while the model with all data shared by all stages are termed as "base model." A sequentially stage modeling always involves one base model and 2 or more stage models.

The base model is displayed on the screen when the "Sequential Stage" dialog is not shown. The stage model of the current stage is displayed when "Sequential Stage" dialog is opened. You can move to the desired stage either by using the stage number popup menu(Windows: dropdown list), or by using the scroll buttons. The current stage number appears as the popup text. The current stage can also be identified by the label displayed in the label text box. The geometry, properties, boundary conditions and load conditions of the current stage model is displayed in the main window if the dialog is opened and the display mode is set to "Current". At the moment you close the dialog, the display on the main window returns to the base model.



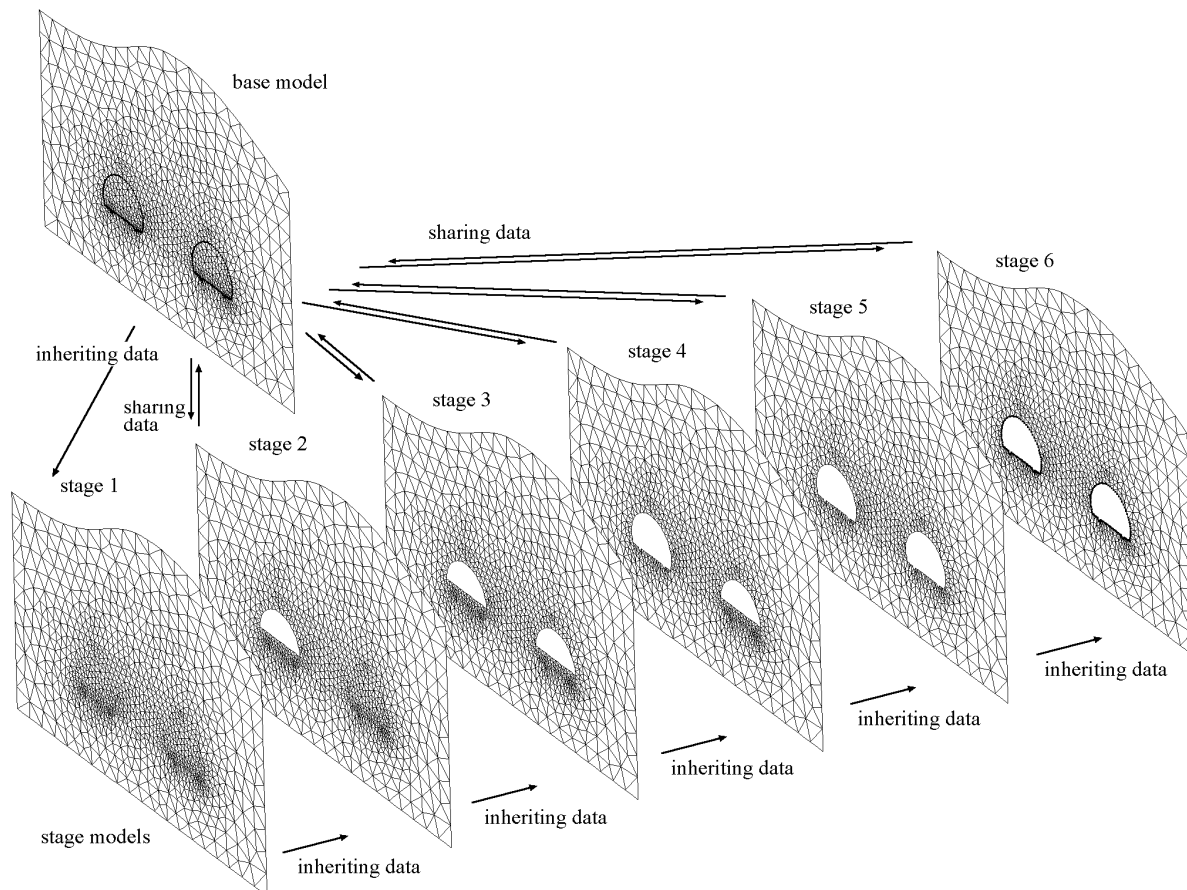
<Base model and stage model>

■ Data sharing and inheriting

All the stage models share the data of the base model. In other words, the base model comprises all the data included in the stage models. The first created stage model (the stage model of stage 1) initially inherits its data from the base model, and is later modified to keep only necessary data by using exclusion, assignment or clearing actions. Likewise, a subsequent stage model initially inherits its data from the preceding stage model, and is modified later by various actions.

Creation or modification of geometric objects, i.e., curves and meshes are allowed only for the base model. Thus, new objects cannot be attached to a stage model directly. The geometric objects created for the base model are automatically inherited to all the currently existing stages.

On the other hand, properties and load conditions can be assigned directly to a stage model. Any assignment performed at a stage level is applied to the corresponding stage model, and also included in the base model. But the assignment is not applied to the other stage models. Nevertheless, any assignment to the base model is automatically inherited to all the currently existing stages.



<Concept of data sharing and inheriting>

■ Solution process of stage models

A complete processing of finite element solver is performed for each of the stage model. The processes are repeated, starting from the first stage and proceeding sequentially to the last. Each process of a stage model is not independent from the others. A stage model is processed in association with its preceding or subsequent stage. The solution results of a stage model are reflected in the processing of the next stage model. For example, unbalanced forces resulting from excavation at a stage are relaxed over the next few stages with prescribed rates. The solution results such as displacements, stresses and so on are accumulated stage after stage. The data accumulation can be artificially controlled if necessary. As an example, the nodal displacements are cleared at the first stage while the computed stresses are retained, which is usually applied to modeling construction stages in consolidated soil.

■ Procedure of sequentially staged modeling

A stage model is constructed in the form of extracting necessary components from the base model. The procedure of sequentially staged modeling can be summarized as follows:

- 1) Create the base model.

It is recommended to complete creating the base model before creating the stages, although the base model may be added or modified in the middle of working with stage models.

- 2) Create stage models and build the geometry of the stage models.

The geometry of a stage is constructed initially by inheriting the geometry from the base model or from the previous stage, and then modified by "Include" and "Exclude" actions.

- 3) Build the data assignment of each stage model.

Property and load condition data of a stage is constructed initially by inheriting the data from the base model or from the previous stage, and then modified by assigning or clearing data.

- 4) Execute the solution process.

The finite element solution process is launched in the same way as non-staged models. Select "Solve" item of **Solve** menu to go through the solution process. (Refer to "Processing of structural analysis" section of Chapter 6.) However, a complete solution process is repeated for each of the stages. The analysis results of a stage are accumulated and applied as a part of input data for the analysis of the subsequent stages.

- 5) Postprocess the analysis results.

The analysis results of the staged model are arranged and saved as a sequential process. The visualization of the analysis results can also be controlled sequentially in accordance with the staged progress.

Creating stage models

Sequentially staged modeling is initiated by choosing "Sequential Stage" button from **Assign** menu. Stage 1 exists at the beginning of staged modeling. Stage 1 initially includes all the objects created so far, and the current assignment of properties, boundary conditions and load conditions.

■ Creating a new stage

A new stage is added to the staged model by clicking **New** button. If the current stage is the last one, the new stage is attached to the last. If the current stage is not the last one, the new stage is inserted between the current and the next stage. Accordingly, the next and the following stages are reassigned with subsequent numbers.

The new stage model initially inherits the geometry and the attributes from the current stage which is now stated as previous stage. The newly created stage takes the status of current stage.

■ Deleting a stage

The current stage is deleted by clicking **Delete** button. If the current stage is the last one, the previous stage takes the status of current stage. Otherwise, the next stage takes the status of current stage, and the subsequent stages are renumbered accordingly. There should be at least one stage, and therefore the current stage cannot be deleted if there is only one stage.

■ Moving to the desired stage

You can move to the desired stage either by using the stage number popup menu(Windows: dropdown list), or by using the scroll buttons. The current stage number appears as the popup text. The current stage can also be identified by the label displayed in the label text box. the contents of the main window, i.e., geometry, properties, boundary conditions and load conditions are displayed on the basis of the current stage.

Building the geometry of stage models

In a sequentially staged modeling, each of the stages represents a complete model with all ingredient of finite element analysis, composed of geometry, properties, boundary conditions and load conditions. However, it is not necessary to create independent models for the stages. Instead, a stage can be constructed simply by including and excluding its components from the predefined base model.

■ Including the selected objects to the current stage

It is the case in which the objects not in the previous stage is to be included into the

current stage. Before including the objects into the current stage, they may not be displayed at the current stage depending on how the display option is set. In order to make the desired object visible, check "All" or "Last" radio button. Then, select the object, and click **Include** button. If the selected objects are already within the current model, the button is dimmed.

■ Excluding the selected objects from the current stage

It is the case in which the objects in the previous stage is to be exclude from the current stage. Select the objects to exclude, and click **Exclude** button. If the selected objects are not within the current model, the button is dimmed.

■ Handling objects created after creation of some stages

As mentioned previously, it is desirable to complete the creation of the base model and then to start creating the stages. Owing to some reason, however, a part of the base model might have been created after creation of some stages, all or in part. In such situations, the newly created objects are automatically included into all the stages already created. If unwanted objects are included in this way, you should move to the relevant stage, and exclude the objects manually.

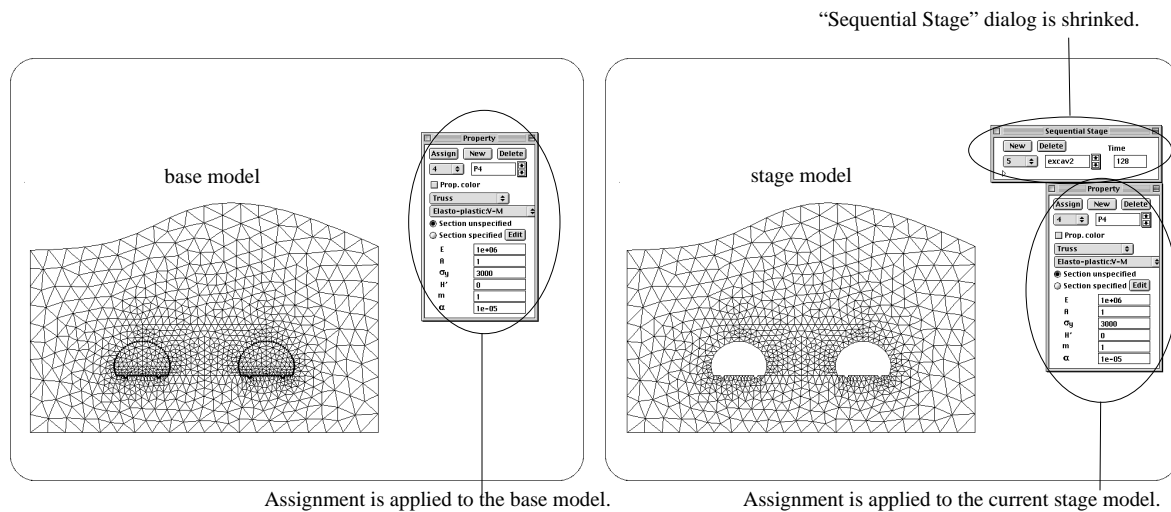
■ "Model Display" options

Only displayed objects are selectable, and thus can be included or excluded as desired. Therefore, it is necessary to control the state of visibility of the desired objects for building the geometry of a stage. The "Sequential Stage" dialog has radio buttons, "All", "Previous", "Current", "Next", "Included" and "Excluded" for the option of object display. The usage of the buttons is described at the beginning of this section.

Property assignment in sequentially staged modeling

In a sequentially staged modeling, the property assignment also varies with stages. For example, a surface mesh region is assigned with property set A at stage 1, and the property of the same region is altered to set B at stage 2. This is the case of a construction modeling with changing material property in some part. Property assignment can be handled either collectively by working with the base model or individually by working with a stage model.

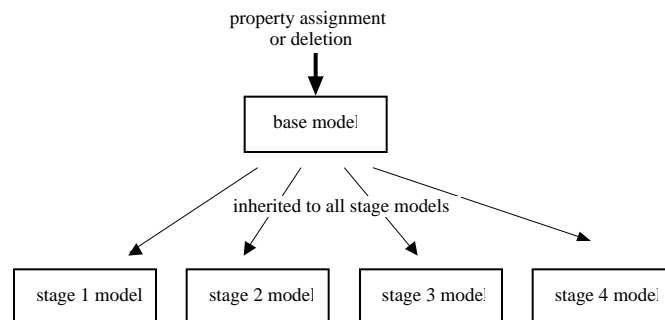
In order to start property assignment, select "Structural Property" item from **Assign** menu. If the "Sequential Stage" dialog is not currently on, the property assignment is performed for the base model. If the dialog is already opened, the dialog shrinks, and the "Structural Property" dialog is positioned at the bottom of the "Sequential Stage" dialog as shown in the figure below. In this case, the assignment is applied to the current stage model.



< Property assignment to the base model and to a stage model >

■ Assigning properties to the base model

The base model takes the property assignment and deletion, when "Sequential Stage" dialog is not on. And the assignment to the base model is automatically inherited to all stage models which have the assigned objects. Deletion of a property set in the base model is also applied to all stage models. Thus, property assignment or deletion in the base model overrides the previous assignments in all stage models at once.

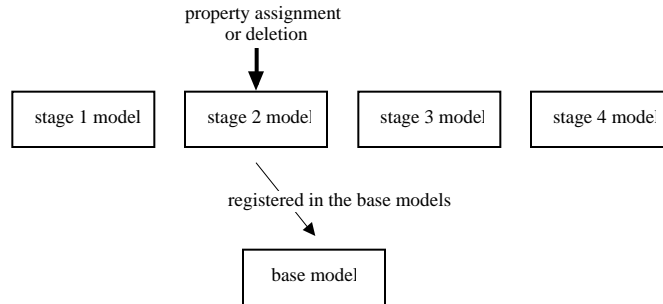


<Property assignment of the base model>

■ Assigning properties to stage models

Only one property set can be applied to an object in the base model, although the property of an object can be altered from one stage to the other. Assignment of the changing properties can be achieved by assigning different property sets to the stage models. The current stage model takes the property assignment and deletion, when "Sequential Stage" dialog is on. The current stage is indicated by the popup menu item in the dialog. Property assignment to a stage model does

not affect the assignment of other stage models. However, the assignment is registered in the base model.



<Property assignment of a stage model>

Load assignment in sequentially staged modeling

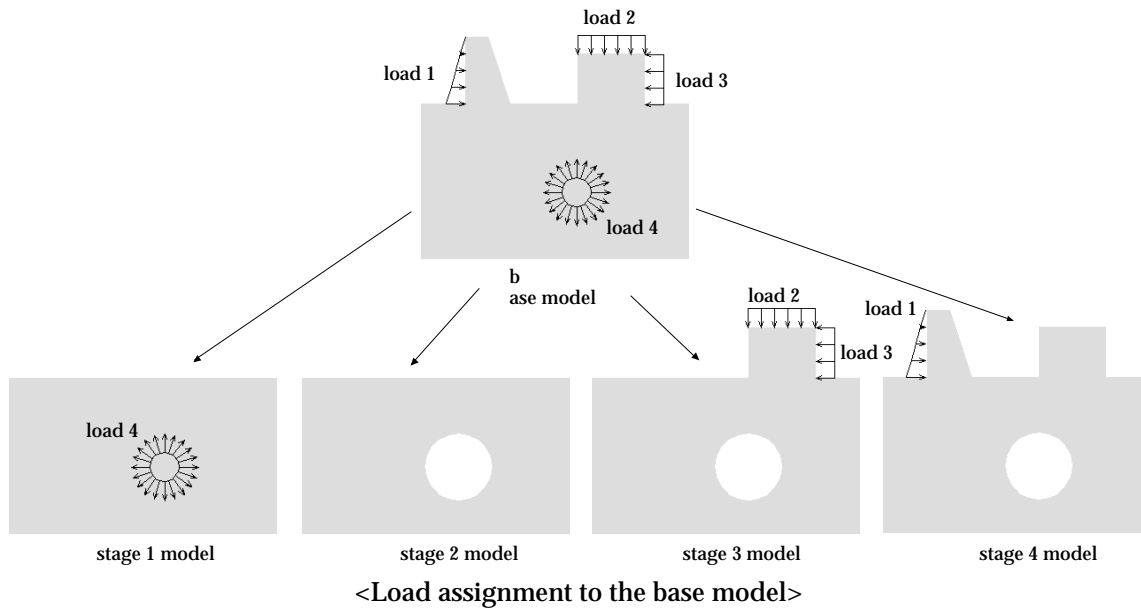
Load assignment can be handled either collectively by working with the base model or individually by working with a stage model. It is similar to the case of property assignment. The load assigned to the base model is inherited to the stage model. However, in solving the whole staged model, a load assignment to an object is effective only for one stage model. In the case of assignment with the base model, the load set is effective at the stage in which the assigned object first appears. In the case of assignment with a stage model, the load set is effective at that stage only.

Assignment of load conditions is initiated by selecting "Load Condition" item from **Assign** menu. If the "Sequential Stage" dialog is not currently on, the load assignment is performed for the base model. If the dialog is already opened, the dialog shrinks, and the "Load Condition" dialog is positioned at the bottom of the "Sequential Stage" dialog as in the case of property assignment.

■ Assigning loads to the base model

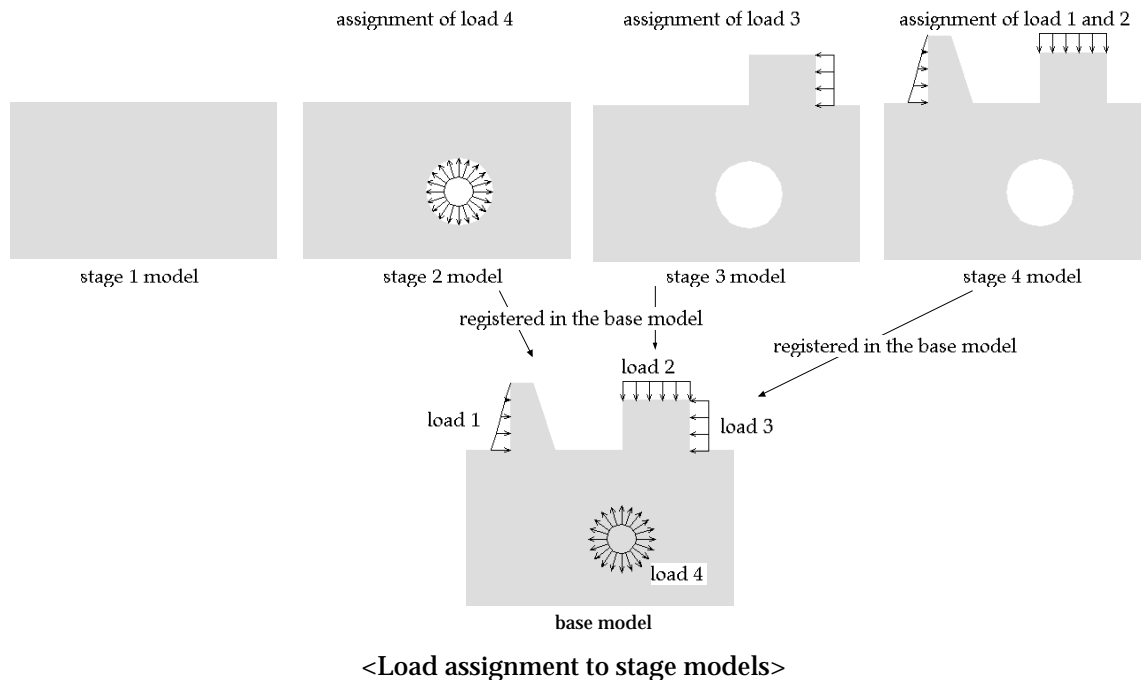
A load condition applied to the base model is inherited to stage models. This does not mean that the load condition is applied to all stage models, but means that the load takes effect at one of the inherited stage models in which the assigned object appears for the first time. Each of the load conditions is involved only once in the system equations, and it is the system equations at the stage the load takes effect. Thus, actual application of the load conditions in the base model is determined automatically by the program, and cannot be controlled by the user.

The following example illustrates how the load assignment of the base model takes effect in stage models. Load 1 to 4 are applied to the base model. Load 4 becomes effective at stage 1, because the object assigned with the load is already in the stage model. Load 2 and 3 take effect at stage 3 in which the assigned object appears. Finally, load 1 appears at stage 4.



■ Assigning loads to stage models

A load condition applied to the base model takes effect at the stage the assigned object appears first. Thus, the stage of actual application may not be realized as desired. Such a problem can be solved by assigning the load condition directly to the desired stage. The following figure shows an example illustrating the difference between the load assignment to the base model and the assignment to the stage models.



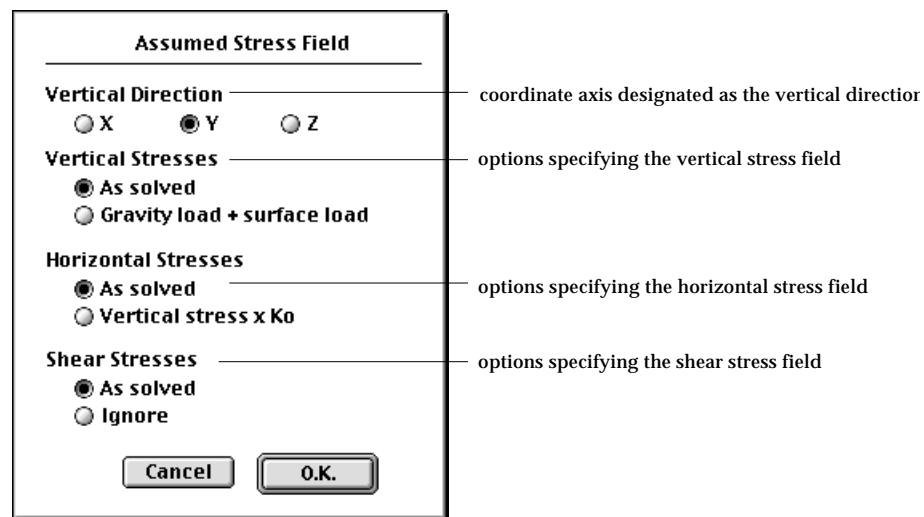
Control in processing of a sequentially staged model

The solution results of a stage model and their reflection on subsequent stage models can be artificially controlled to embody the actual situation more realistically.

■ Assumed stress field of a stage model

The solution results of a stage model can be replaced by assumed stress field as a whole or in part. Such a fabrication of the analysis results is often used to represent the initial state in soil mechanical analyses.

If you check the "Assumed stress field" box, **Edit** button to the right turns to enabled state from disabled state. Click the button to designate how the stress field should be assumed. Then, "Assumed Stress Field" dialog appears.



Using the dialog, the stresses can be controlled as follows:

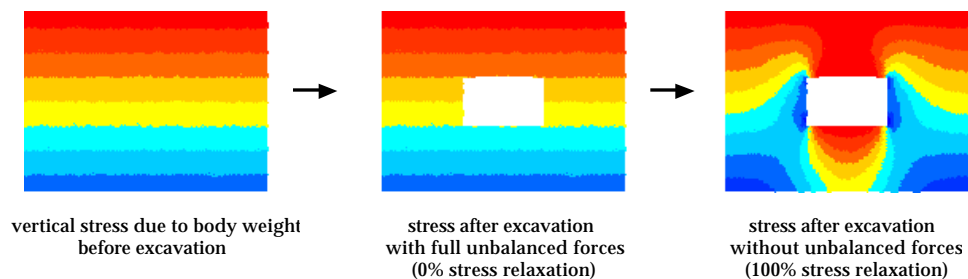
- "Vertical Direction": Select the vertical direction from the coordinate axis, X, Y and Z. This option is enabled only for 3-D problems. The Y direction is assumed to be the vertical direction for 2-D modeling.
- "Vertical Stresses": Select the way how the stresses in the vertical direction should be determined. If "As solved" ratio button is on, the vertical stresses are determined by the finite element solution. Otherwise, the vertical stresses are directly calculated from the vertical loads.
- "Horizontal Stresses": Select the way how the stresses in the horizontal plane should be determined. If "Vertical Stress $\times K_0$ " option is on, the horizontal stresses are directly calculated from the vertical stresses using the neutral soil pressure coefficient K_0 .

- "Shear Stresses": Select the way how the shear stresses should be determined. If "Ignore" option is on, the shear stresses are set to 0, regardless of the actually computed values.

In order to nullify the rule of stress assumption and use the solved values, uncheck the "Assumed stress field" box.

■ Prescribing the staged stress relaxation rate

Excavation during construction can be modeled simply by excluding the elements of the excavation part from the corresponding stage model. Removal of elements results in change of stresses. In sequentially staged analysis, it is sometimes assumed that the stress change does not occur at once, but gradually over a few stages. Such an assumption is often adopted in 2 dimensional modeling of excavation progressing in the direction vertical to the plane. The assumption is based on the concept of unbalanced forces and stress relaxation. At the beginning, there exist unbalanced forces resisting the stress change. Stress relaxation, i.e., transition of stresses to the final state progresses gradually in accordance with the reduction of the unbalanced forces.



< Concept of unbalanced forces and stress relaxation >

The gradual stress transition is represented by the stress relaxation rates prescribed for the stages subsequent to the excavation. The procedure of prescribing the stress relaxation rates is as follows:

- 1) Check "Stepwise stress relaxation" box in the "Sequential Stage" dialog. Only when the box is checked, **Edit** button to the right is enabled. Otherwise, it is disabled.
- 2) Click **Edit** button to open "Stress Relaxation Rate" dialog. The stress relaxation rate is prescribed using this dialog.
- 3) Create the lines of stages by clicking **Add** button. The stage offset indicate the number of stages from the current stage. The offset of the current stage is 0. Create as many stages as the relaxation rate is prescribed.
- 4) Set the relaxation rates.

Set the relaxation rate for each line of stages. There are 2 columns of relaxation rates. "No Refilling" column prescribes the relaxation rates of excavation without refilling. "w/ Refilling" column applies to the relaxation rate of excavation with later refilling.

- 4) Click button to complete prescribing the stress relaxation rate.

The relaxation rate data is saved to a text file by clicking button, and imported from a text file by clicking button.

■ Clearing displacements

There is an option to clear all the displacements accumulated up to this stage. Check "Clear displacement" box to remove the displacements but retain the stresses. This option is often used at the initial stage of soil mechanical analysis to represent the initial state of stresses due to body weight.

The dialog box titled "Stress Relaxation Rate" contains a table with the following columns: "number of stages from the current stage", "time from the current stage", "stress relaxation rate for excavation without refilling", and "stress relaxation rate for excavation with refilling". The table has three rows: "Stage Elapse", "Time", and "No Refilling w/ Refilling". The "No Refilling" column has values 0.4, 0.3, and 0.3 for stages 0, 1, and 2 respectively. The "w/ Refilling" column has values 1, 0, and 0 for stages 0, 1, and 2 respectively. The "current stage" is indicated as 0. Below the table are buttons for "Add", "Delete", "Save", "Import", "Cancel", and "O.K.". There are also radio buttons for "Apply by stage" (selected) and "Apply by time".

Annotations on the left side of the dialog box include:

- time from the current stage
- stress relaxation rate for excavation with refilling
- number of stages from the current stage
- stress relaxation rate for excavation without refilling
- current stage
- delete the selected line
- add a new line
- save the data in a text file
- read the data from a text file

Annotations on the right side of the dialog box include:

- pre-excitation stage
- stage elapse = 0 (excavation stage)
- 40% stress relaxation (60 % unbalanced forces)
- stage elapse = 1
- 70% stress relaxation (30 % unbalanced forces)
- stage elapse = 2
- 100% stress relaxation (0 % unbalanced forces)

Four stress distribution plots are shown on the right, corresponding to the stages: pre-excitation stage, stage elapse = 0, stage elapse = 1, and stage elapse = 2. The plots show the stress distribution in a cross-section of the excavation, with colors representing different stress levels (red for high, blue for low).

< Prescribing the stress relaxation rate >

