

Chapter 3

Curves and Surface Primitives

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

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

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

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

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

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

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

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

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

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

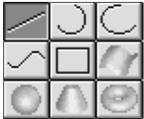
Creating Curves and Surface Primitives

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

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

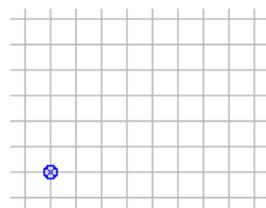
To create a type of curve or surface primitive, the corresponding tool button should be pressed. As long as the button is pressed, inputting of the control points can proceed without interruption. For example, a view angle or rendering mode can be changed without terminating the input procedure.

Creating straight lines



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

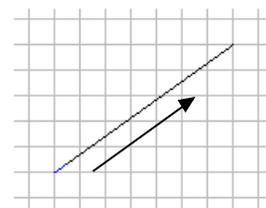
A straight line has a direction. This direction is determined by which one of the two end points is first defined. That is, a straight line is directed from the first end point to the other. The direction of a line makes sense in such cases as dividing a line and assigning load conditions. However, these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



Inputting the first point



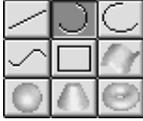
Inputting the 2nd point



Inputting the 3rd point

< Procedure of inputting a circular arc >

Creating circles or circular arcs



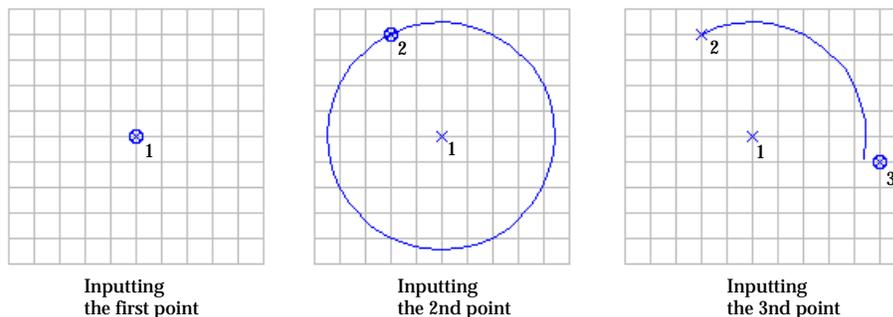
Press the circle tool button  in order to create circles and circular arcs. The **Circle** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to set whether to make circle or to make arc, and how to input. The first 5 items are used to create circular arcs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

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

■ Arc

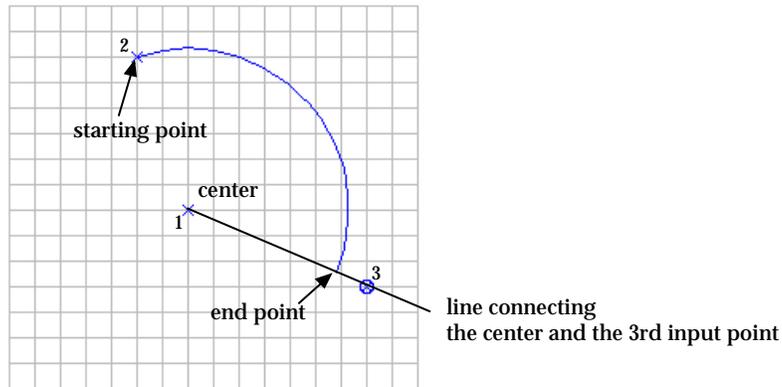
While this option is active, an arc is created by entering its center and two end points. The direction of the arc is determined so that its central angle become less than 180° .

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



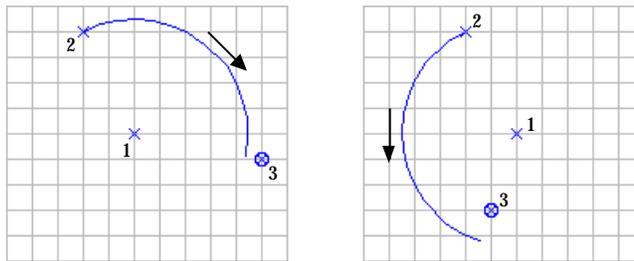
< Procedure of inputting a circular arc >

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



< Construction of an arc >

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

< The arcs are made so that their central angles are less than 180° >

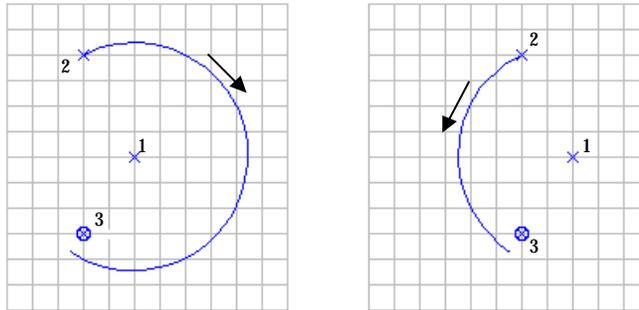
■ Clockwise Arc

While this option is active, a circular arc is created by entering its center and two end points. The arc is directed clockwise from the first end point to the second.

The meaning of "clockwise" or "counter clockwise" is relative to the view direction in 3-dimensional space. That is, the direction is based on the current view of the screen. Therefore, a clockwise arc may look counter clockwise, when the screen view is reversed by rotating the view direction. However, the absolute direction of the arc in 3-dimensional space remains the same as created, regardless of the view transformation.

■ Counter-CW Arc

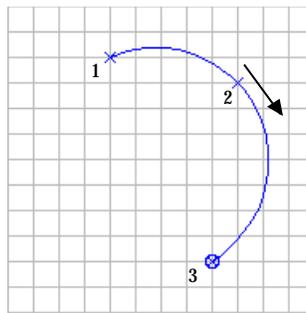
While this option is active, a circular arc is created by entering its center and two end points. The arc is directed counter-clockwise from the first end point to the second. Here, the direction of the arc is based on the current view of the screen, as in the case of "Clockwise arc".



< Clockwise arc and counter-CW arc >

■ Three Point Arc

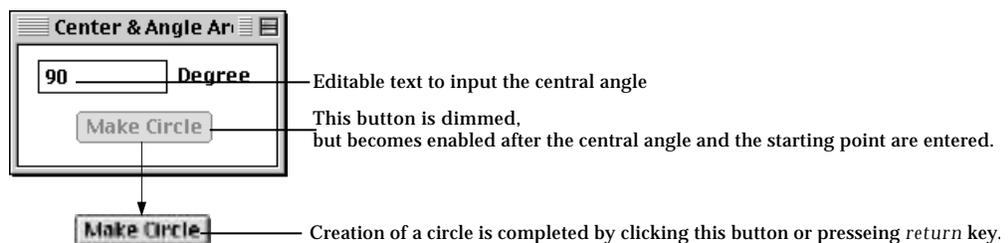
While this option is active, a circular arc is created by entering three points on the arc consecutively, i.e. the starting point, a mid-point, and the terminal point. The arc is directed from the starting point through the mid-point to the terminal point.



< Three point arc >

■ Center & Angle Arc

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



■ Clockwise Circle

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

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

■ Counter-CW Circle

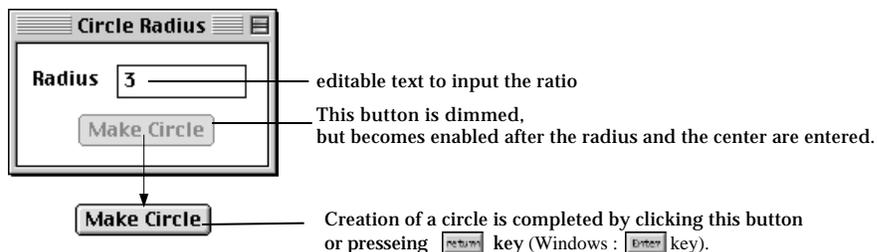
While this option is checked, a circle is created by entering three points in the same manner as in the case of "Clockwise Circle." The circle is directed counter-clockwise

■ Three Point Circle

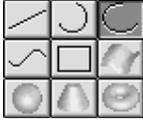
While this option is checked, a circle is created by entering three points on the circle consecutively, i.e. the starting point and two other points on the circumference. The circle is directed from the starting point toward the second.

■ Center & Radius Circle

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

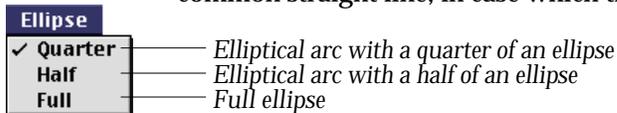


Creating ellipses or elliptical arcs



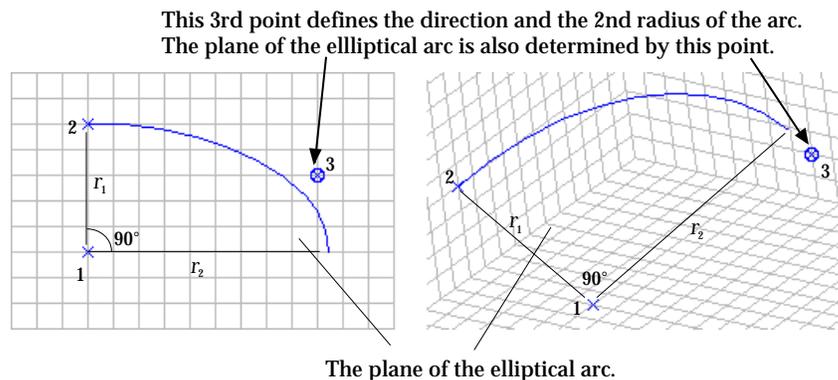
Press the ellipse tool button  in order to create ellipses or elliptical arcs. The **Ellipse** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to select the type of the ellipse, i.e., quarter, half or full ellipse. The  marked option is used for creation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

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



■ Quarter

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

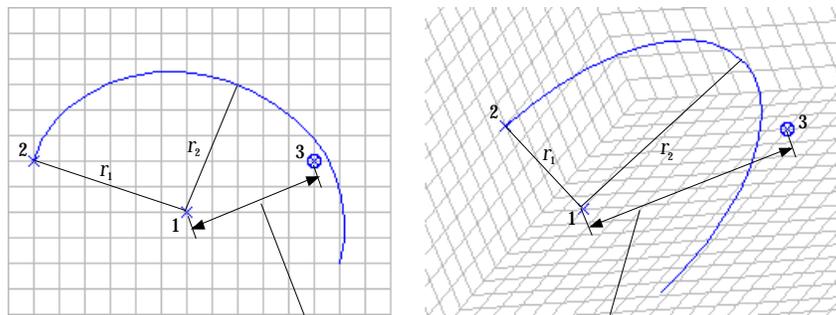


< Quarter ellipse >

■ Half

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

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

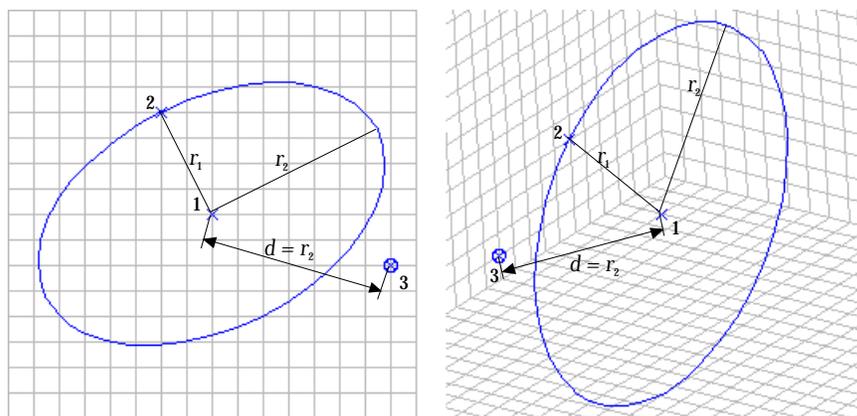


The length of the 2nd radius, r_2 is equal to this distance.

< Half ellipse >

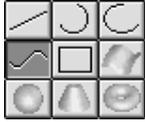
■ Full

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

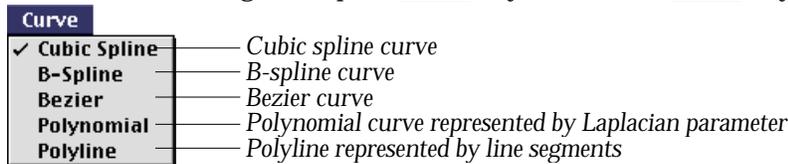


< Full ellipse >

Creating parametric curves

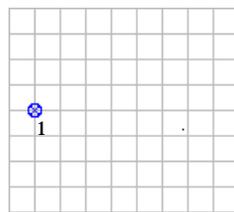


Press the curve tool button  in order to create parametric curves, which are defined by three or more control points. The **Curve** menu appears at the right end of the menu bar. The menu items, as shown below, consist of options to select the type of the curve, i.e., cubic spline, B-spline, Bezier,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

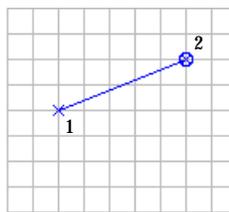


■ Cubic Spline

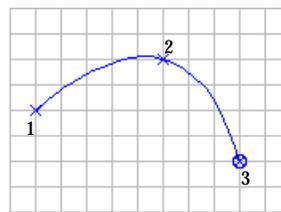
While this option is checked, a cubic spline curve is created by entering four or more points. When the first and the second point are entered, a straight line is drawn, connecting these two points. The third point makes a parabola. A cubic spline curve is formed after the fourth and the consecutive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



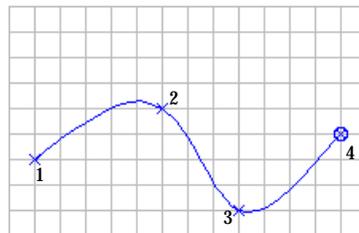
The first point is entered.



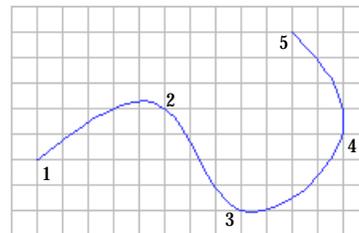
Straight line is formed by the 2nd point.



A parabola is formed by the 3rd point.



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



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

< Progress of creating cubic spline >

■ B-Spline

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

■ Bezier

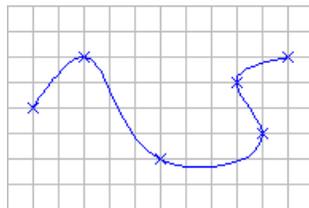
While this option is checked, a Bezier curve is created by entering three or more control points. A Bezier curve is formed when the third and the consecutive points are entered. The process of creating Bezier curves is similar to that of creating cubic spline curves.

■ Polynomial

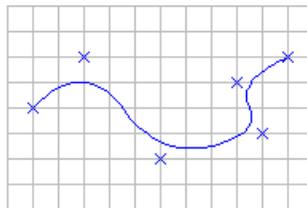
While this option is checked, a polynomial curve is created by entering three or more control points. A polynomial curve represented by Laplacian parameters is formed when the third and the consecutive points are entered.

■ Polyline

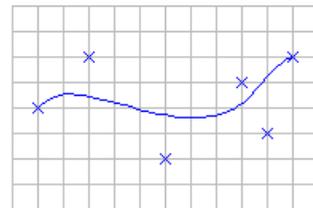
While this option is checked, a polyline is created by entering three or more control points. A polyline is represented by line segments connecting consecutive two points. A polygon is obtained by overlapping the last control point over the first one.



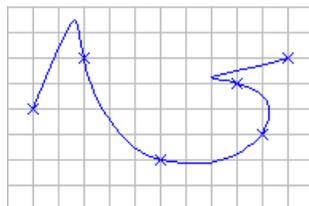
Cubic spline curve



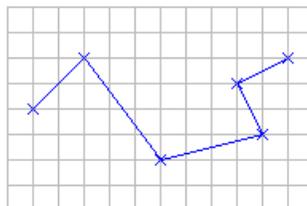
B-spline curve



Bezier curve



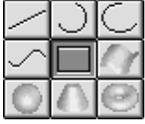
Polynomial curve



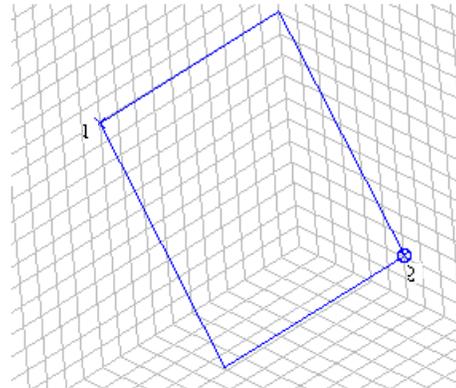
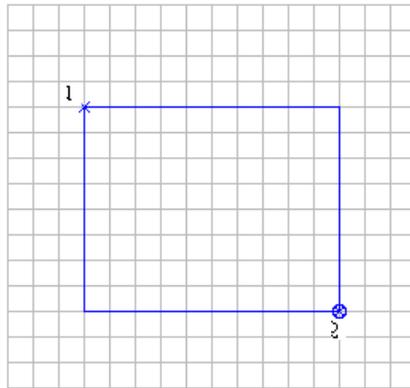
Polyline

< Parametric curves >

Creating rectangles



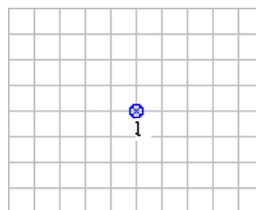
Press the rectangle tool button  in order to create rectangles, which are defined by entering two diagonal corner points of the rectangle. If these two points are on a grid plane, the rectangle is created on the same plane. For example, the first and the second points have the equal z coordinate value,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



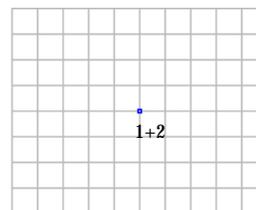
< Rectangle >

Creating single points

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



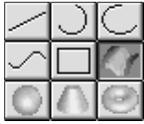
Inputting the first point.



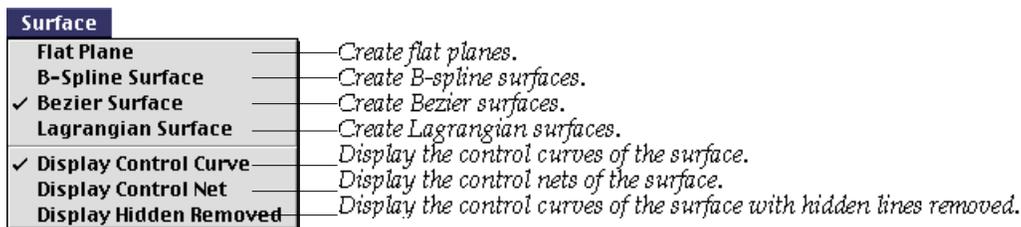
Creating a single point by clicking the first point again.

< Single point >

Creating parametric surfaces

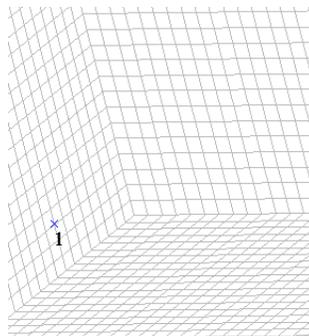


Press the surface tool button  in order to create parametric surfaces. The types of parametric surface include flat plane, B-spline surface, Bezier surface and Lagrangian surface. In order to set the type for creation, select one of the items from the **Surface** menu shown below. Currently effective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

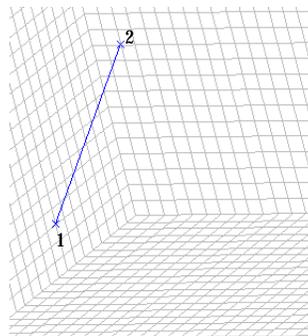


■ Flat Plane

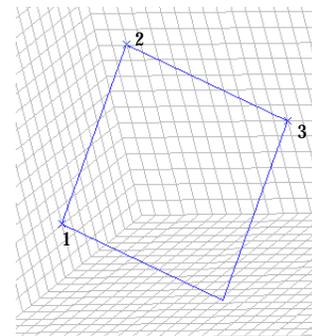
While this type is marked, a flat plane is created by entering 3 points. As exemplified in the following figure, the flat plate is represented by a diamond shape. The first and the second input point form an edge of the diamond, and the second and the third point form another one.



Inputting 1st point



Inputting 2nd point



Inputting 3rd point

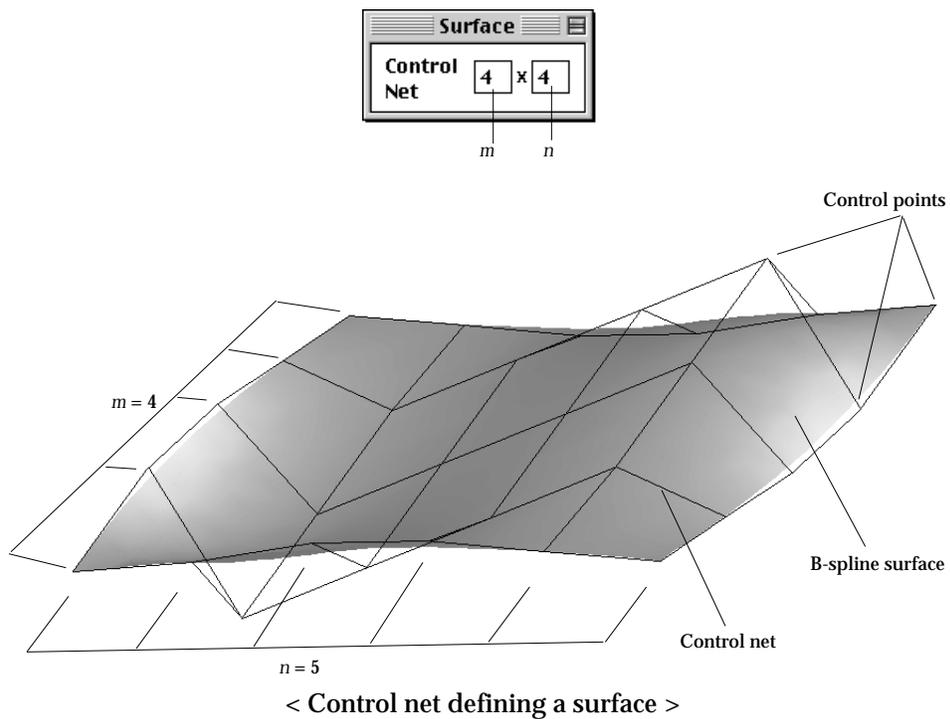
< Creating a flat plane >

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

coordinates.

■ B-spline Surface

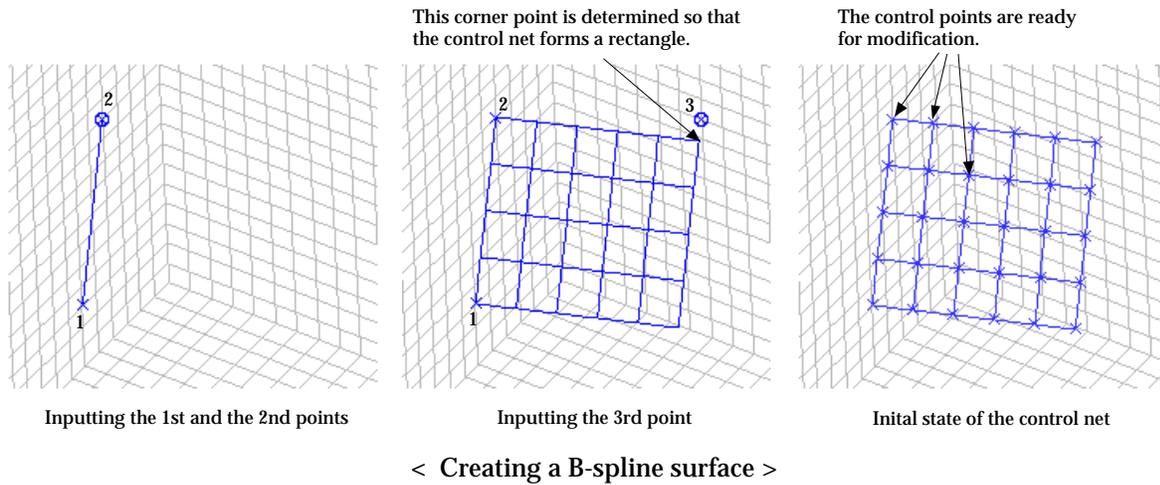
While this type is marked, a B-spline surface is created by specifying the size of the control net, entering 3 corner points and consecutively editing the coordinates of the control points. The size of the control net is defined in the form of $m \times n$ and entered using "Surface" dialog. Here, the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



To create a B-spline surface,

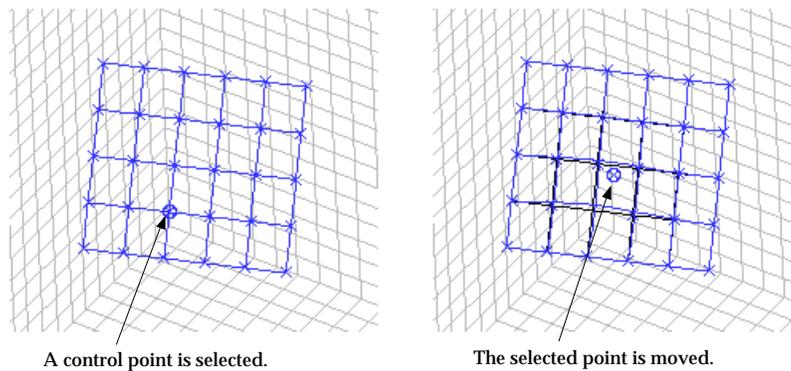
1. Enter $m \times n$, the size of the control net in the "Surface" dialog.
2. Input the first and the second point.
They form a line connecting two corners of the control net. The line is also an edge of the control net, consisting of $(m+1)$ control points.
3. Input the the third point.
The initial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

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



desired positions as follows.

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



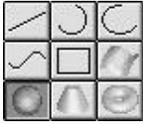
< Modifying the control net >

■ Bezier Surface

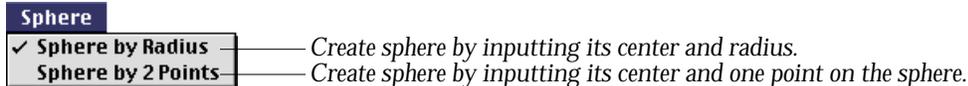
■ Lagrangian Surface

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

Creating spheres



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



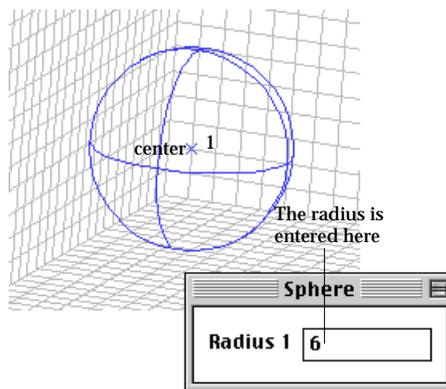
■ Sphere by Radius

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

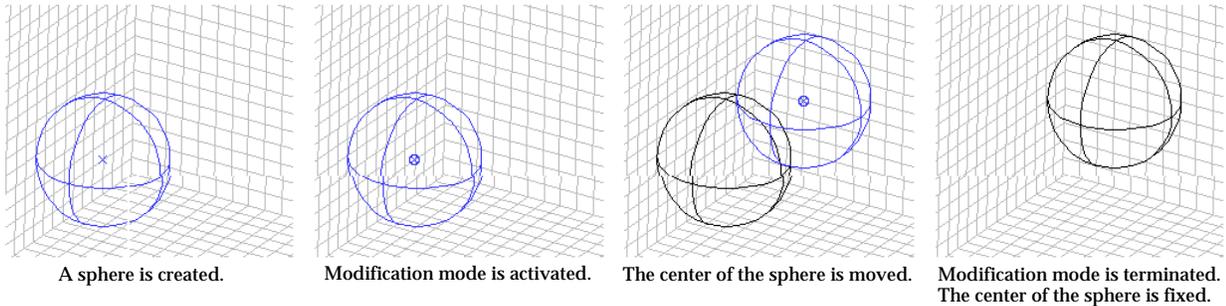
Modification mode is automatically activated right after a sphere is created. Its center and radius can be modified consecutively. Place the screen cursor on the center marked with \times , and press the mouse button. Then, the mark will be changed into \otimes . Keep the mouse button pressed while moving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

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

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



< Creating a sphere by its center and radius >



< Moving the center of the newly created sphere >

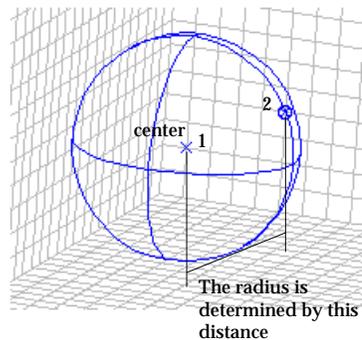
■ Sphere by 2 Points

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

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

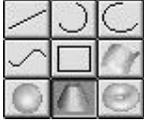
Modification mode is automatically activated right after a sphere is created. And, its center and the second point can be modified consecutively in the same way as explained for "Sphere by radius". While moving one of the two points, the other point stays unchanged, and accordingly the radius of the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

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



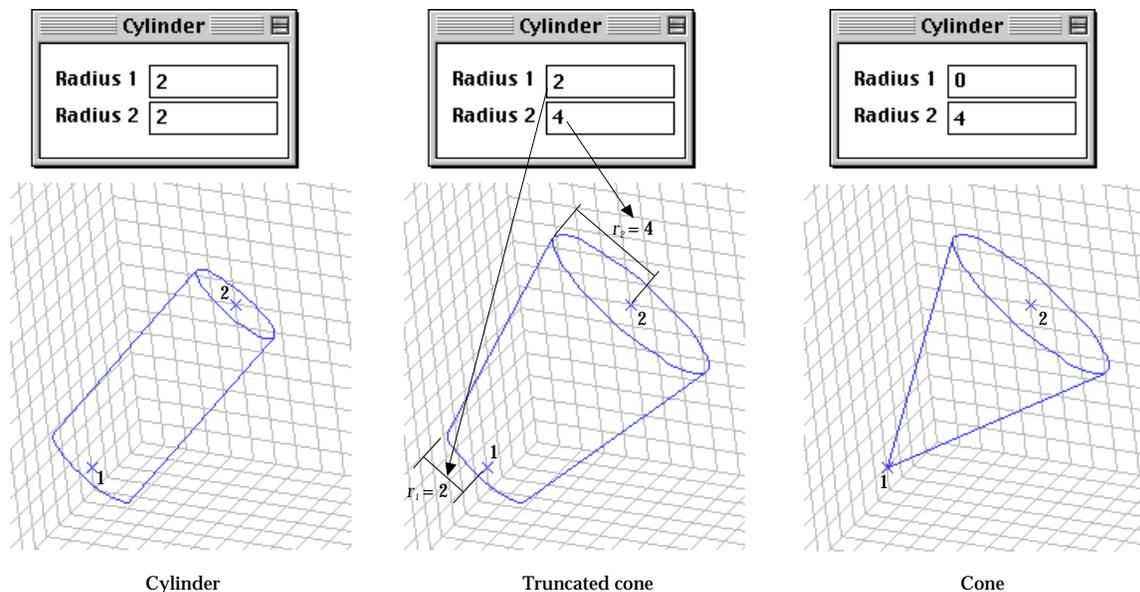
< Inputting a sphere by its center and one point on the sphere >

Creating cylinders, cones or truncated cones



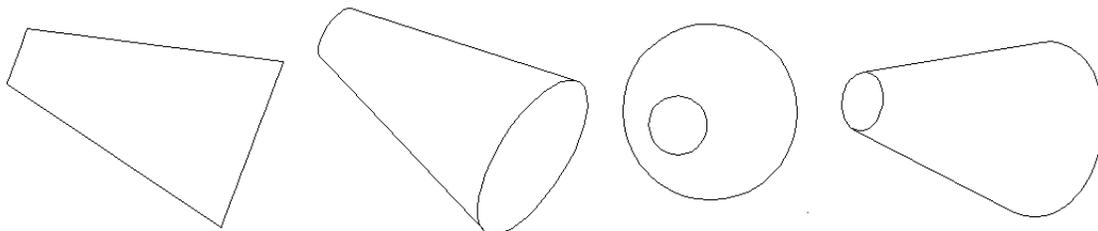
Press the cylinder tool button  in order to create cylinders, cones or truncated cones, which are treated as the same kind of primitive surfaces in VisualFEA. These primitive surfaces are defined by two end points which form an axis, and two radii. A cylinder is created when the two radii are the same.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

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



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

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



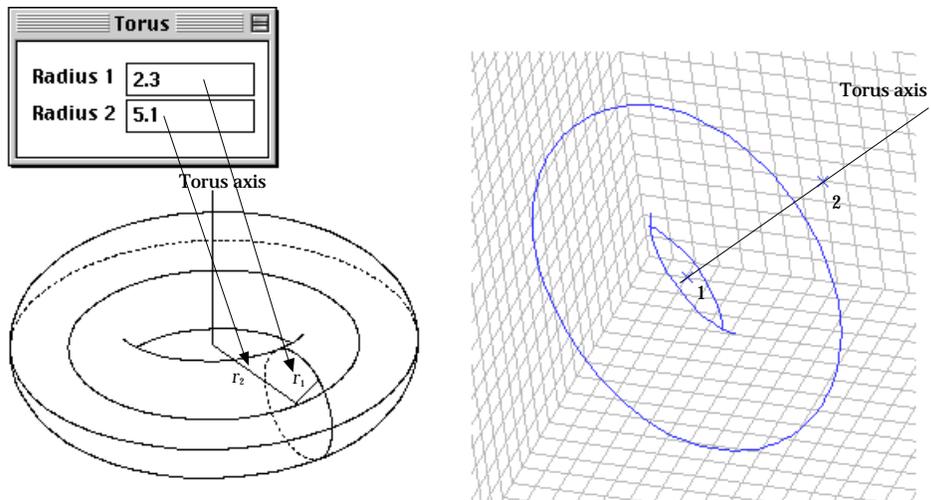
< Different display of a truncated cone depending on the view direction >

Creating tori



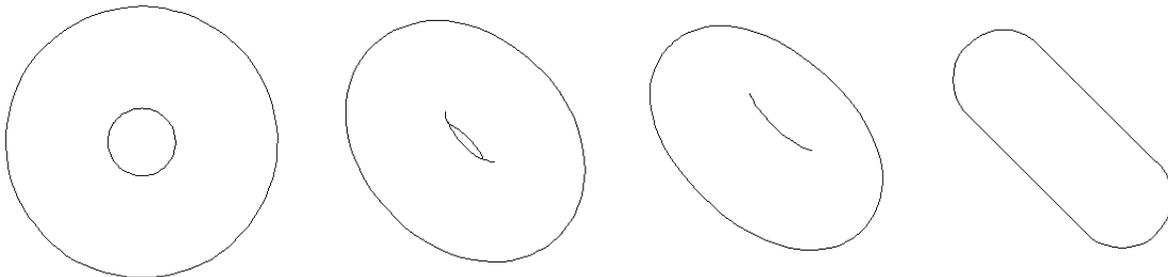
Press the torus tool button  in order to create tori, each of which has an axis and two radii. The axis is defined by two points. The texts entered for "Radius 1" and "Radius 2" define respectively the radius r_1 of the small circle and r_2 of the large one as indicated in the figure below. The texts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

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



< The axis and two radii of a torus >

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



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

Handling Curves and Surface Primitives

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

General editing commands

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

■ Deleting curves and surface primitives

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

■ Copying curves and surface primitives

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

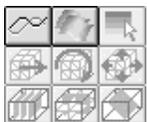
■ Cutting curves and surface primitives

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

■ Pasting curves and surface primitives

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

Reshaping and moving



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

deleting control points are not allowed.

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

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

■ Activating modification mode

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

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

■ Moving control points by mouse

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

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

■ Moving control points by keyboard input

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

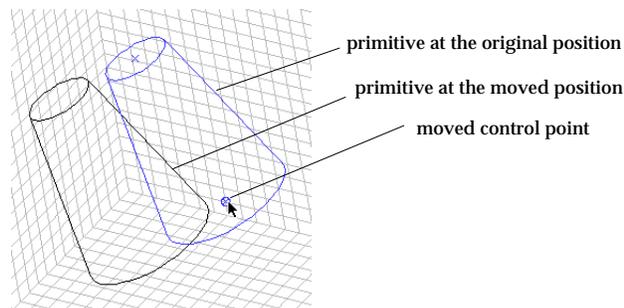
- 1) Select a curve or a surface primitive to modify.
- 2) Click the control point to move.
- 3) Edit the coordinates of the selected point in the text boxes in the tool palette.
The coordinates of the selected point are displayed in the editable text boxes. Select and edit the X, Y and Z coordinates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

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

■ Moving an entire curve or surface primitive

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

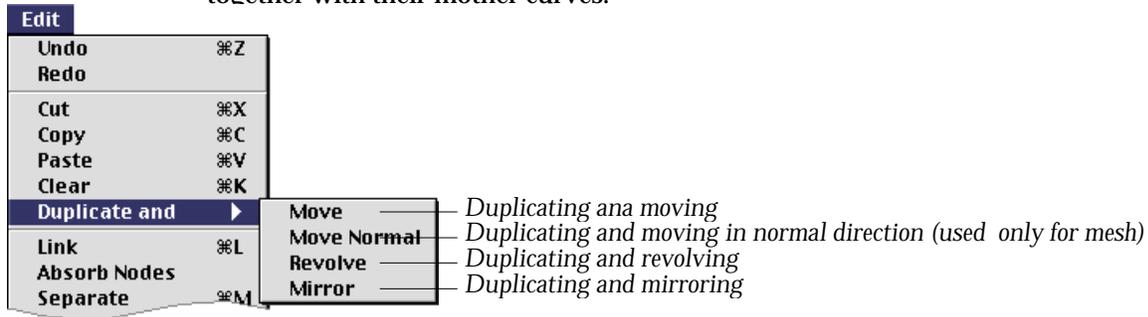
- 1) Select a curve or a surface primitive to modify.
- 2) Click the control point to move.
- 3) Press *shift* key.
- 4) Place the cursor over the  marked point, and press the mouse button.
- 5) Drag the point to the desired position, and release the mouse button.
The entire curve or primitive moves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



< Movement of an entire surface primitive >

Duplicating curves and surface primitives

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

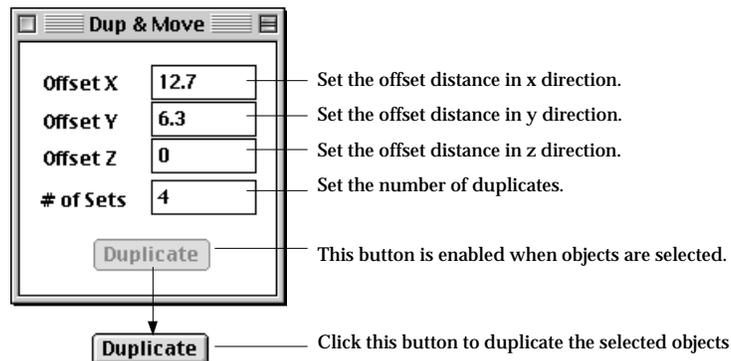


■ Duplicating curves and surface primitives, and moving

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

- 1) Choose “Move” from **Duplicate and** submenu.

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

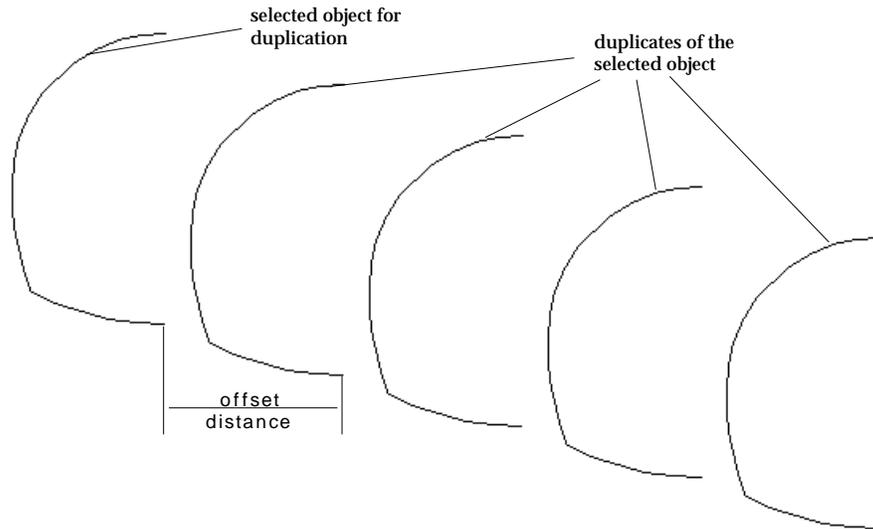


- 2) Set the offset distances in x, y, z directions.
The duplicates are spread in parallel with the specified offset distances. If all the offset distances are set to 0, no duplication will be done.
- 3) Set the number of duplicates to make.
The number of duplications is set by entering in this text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

one or more objects are selected.

- 5) Click **Duplicate** button.

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



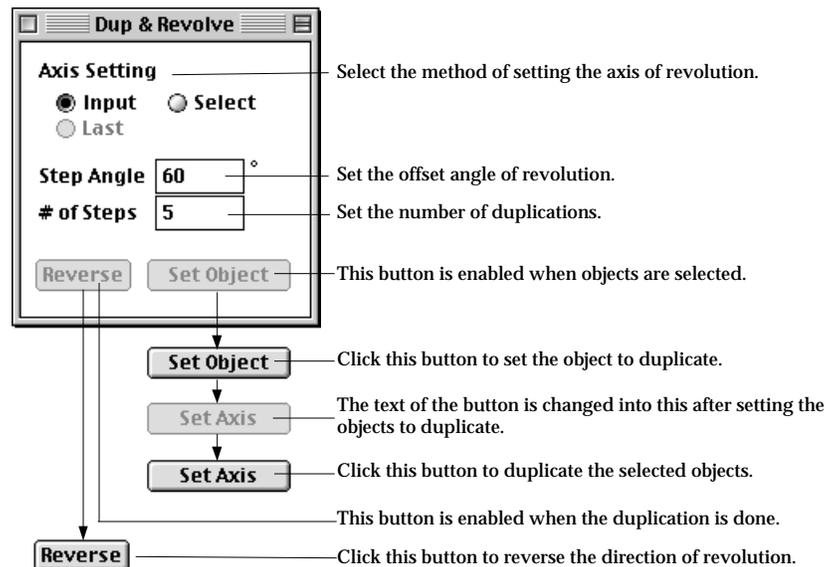
<Duplicating the selected curves and moving >

■ Duplicating curves and surface primitives, and revolving

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

- 1) Choose “Revolve” from **Duplicate and** submenu.

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



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

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

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

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

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

- 4) Set the number of duplicates to make.

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

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

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

- 6) Click button.

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

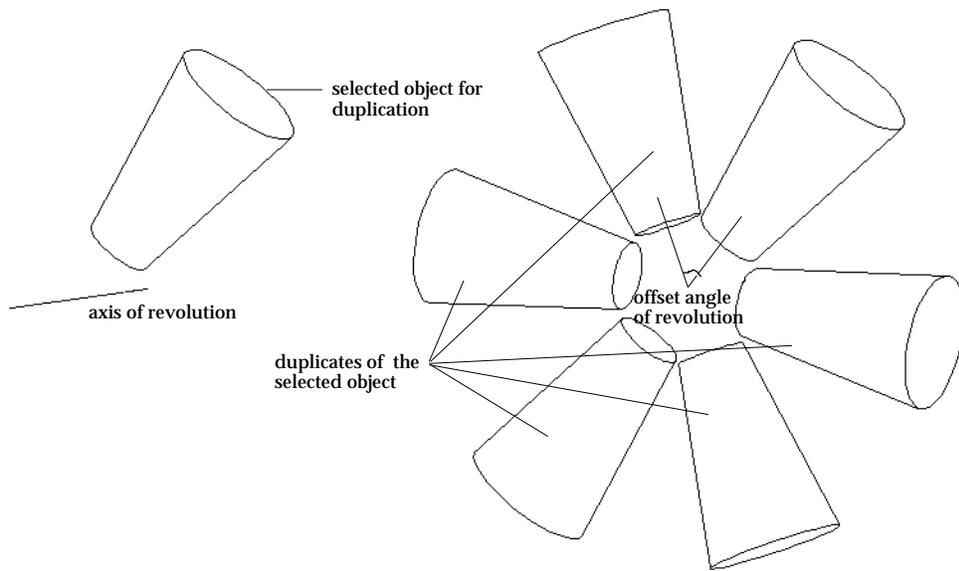
- 7) Input or select the axis of revolution.

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

- 8) Click button.

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

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

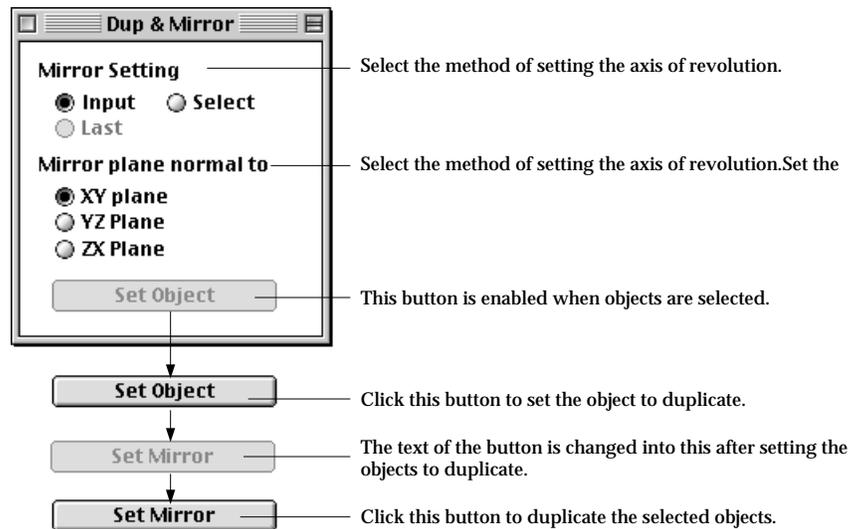


< Duplicating a surface primitive and revolving >

■ Mirroring curves and surface primitives

Duplicate of selected objects can be created by mirroring them.

- 1) Choose "Mirror" from **Duplicate and** submenu. One of the selection tools is activated if not currently, and "Dup and Mirror" dialog box appears.



- 2) Select the method of setting the mirror. The mirror plane for duplication may be set by one of the following 3 methods.
 - Input : input the mirror plane by entering a new straight line just as

creating a new line.

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

3) Select the plane normal to the mirror

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

4) Select the curves or surface primitives to duplicate.

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

5) Click button.

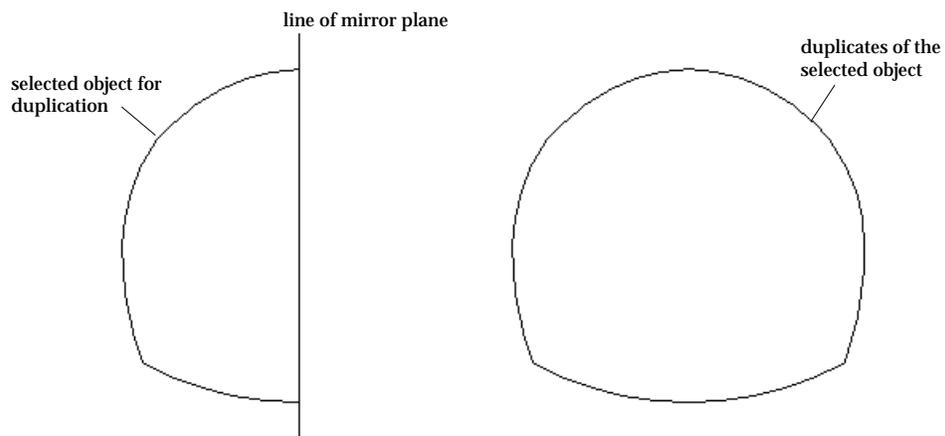
The selected objects are set for duplication. The text of the button is changed into . If “Mirror Setting” is set to “Last”, the button is enabled. Otherwise, it is disabled.

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

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

7) Click button.

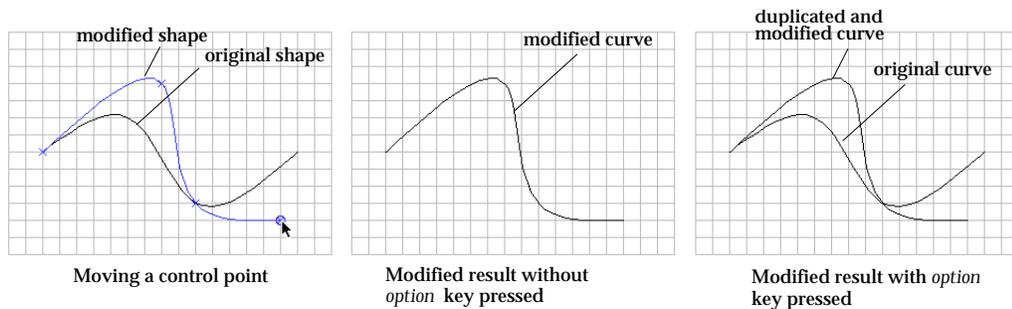
The duplication is processed, and the text of the button is changed into which indicates it is ready again for another duplication process.



< Mirroring a curve >

■ Modifying a curve or a primitive with its duplicate

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



< Modifying a curve or its duplicate >

■ Modifying attributes of a primitive surface

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

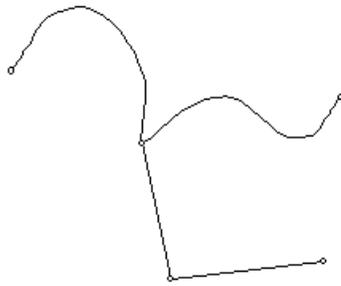
Processing curves and surface primitives

■ Linking curves

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

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

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



The selected curves are not connected serially.



The selected curves are not connected.

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

■ Separating curves

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

■ Filletting two straight lines

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

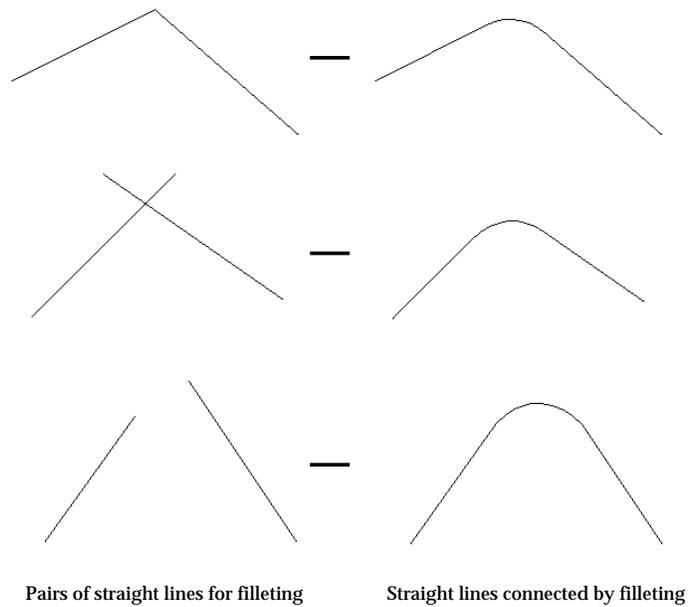
- 1) Activate curve selection tool , if it is not in action.
- 2) Select two straight lines for filletting.
"Fillet" item of **Edit** menu is enabled only when 2 straight lines are selected.
- 3) Choose "Fillet" from **Edit** menu.
"Fillet" dialog appears on the screen.
- 4) Enter the fillet radius using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

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

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



< Fillet dialog >



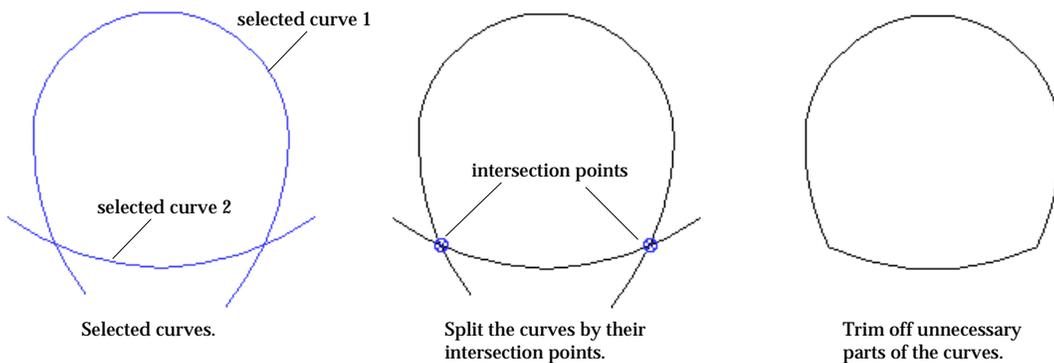
< Examples of filleting >

■ Splitting curves by their intersection points

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

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

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



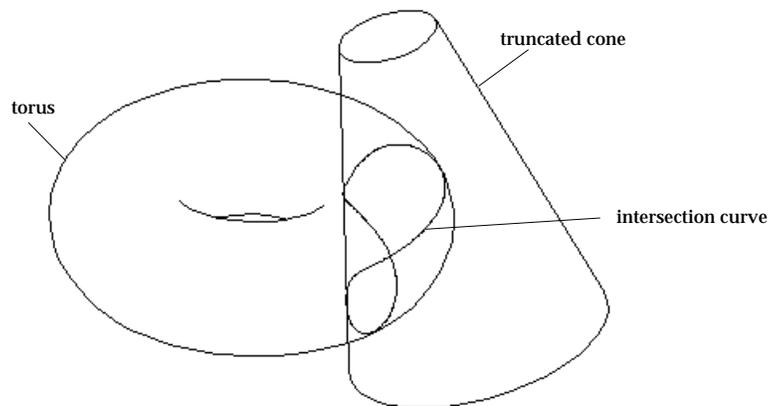
Example of processing curves using their intersection points

■ Obtaining intersection curves between surface primitives

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

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

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

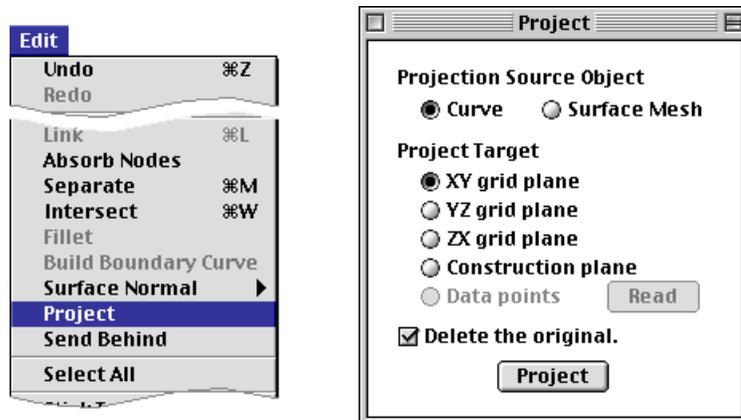


< Example of intersecting surface primitives >

■ Projecting curves

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

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



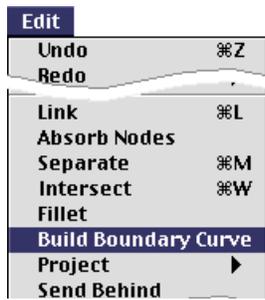
- 2) Turn on the radio button "Curve" under "Projection Source Object."
The curve selection tool  is automatically activated.

- 3) Set the projection target.
There are radio buttons "XY grid plane", "YZ grid plane", "ZX grid plane" and "Construction plane" which represent projection targets. Select one of them.
- 4) Select the curves for projection.
When one or more curves are selected, **Project** button is enabled. The selected curves are the mother curves for projection.
- 5) Click **Project** button.
The the selected curves are projected on the target object.

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

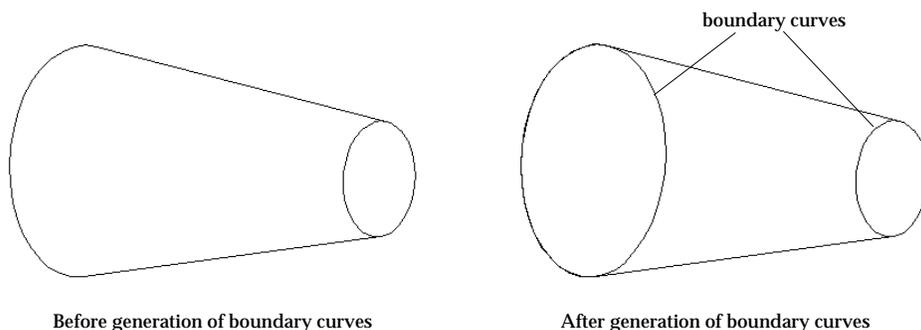
■ Building boundary curves of a surface primitive

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



- 1) Activate surface primitive selection tool  if it is not in action.
- 2) Select surface primitives, the boundary curves of which are to be generated.
When one or more surface primitives are selected, "Build Boundary Curve" item is enabled.
- 3) Choose "Build Boundary Curve" from **Edit** menu.
This command is effective only for open surface primitive. Boundary curves are created on all the effective surface primitives.

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



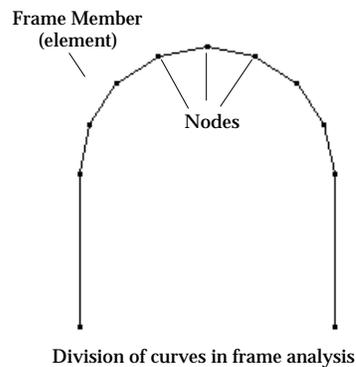
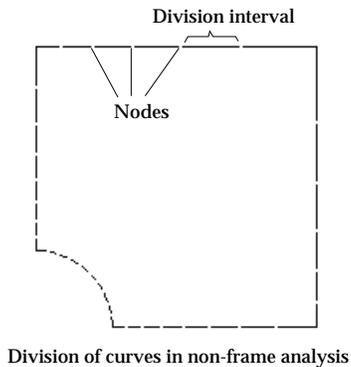
< Building boundary curves of surface primitives >

Curve Division

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

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

Divide	
Divide 	Divide the selected curves.
Divide None	
Divide 1	
Divide 2	
Divide 3	
Divide 4	Set the number of divisions.
✓ Divide 6	
Divide 8	
Divide 10	
Divide 20	
Other Divisions...	Specify the number of divisions which is not in the menu items.
✓ W= 1:1	
W= 1:2	Set the weight of intervals.
W= 1:3	
W= 1:4	
Other Weight...	Specify the number of divisions which is not in the menu items.
Reverse Weight	Reverse the weight of the interval for the last divided curve.
Division Items...	Add or delete the menu items setting the number of divisions.
Weight Items...	Add or delete the menu items setting the wight of division density.

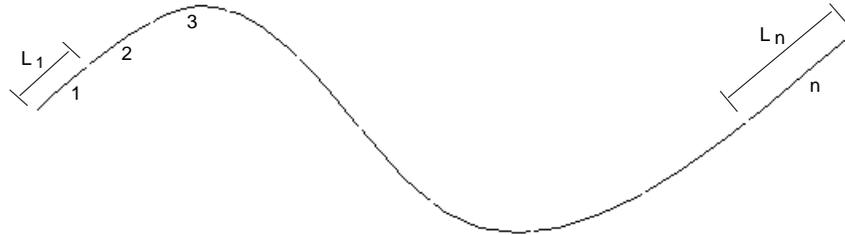


< Division of curves >

Dividing curves

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

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



< Number of divisions and weight of division density >

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

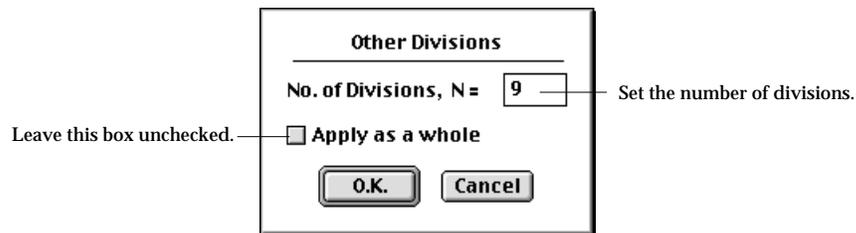


■ Dividing curves individually

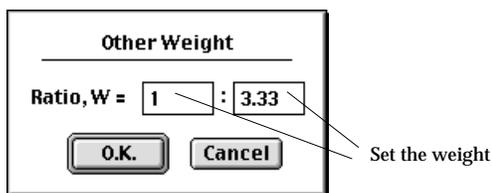
A group of selected curves as well as a single curve may be divided at once, with individual application of the specified number of divisions and weight of division density. This can be achieved simply in the following steps.

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

- "Divide" item
The number of divisions and the weight of division density with marks are applied in dividing the selected curve(s).
- An item for the number of divisions
The selected curves are divided by the number of divisions represented by the menu item.
- "Other Divisions..." item
"Other Divisions" dialog box appears. This dialog is useful for dividing the curve(s) by a number of divisions not shown as a menu item. Set **No. of Divisions, N =** item, and click button. Be sure **Apply as a whole** item is not checked.



- An item for the weight of division density
The selected curves are divided with the weight represented by the menu item.
- "Other Weight..." item
This dialog is useful for dividing the curve(s) with weight of division density not shown as a menu item. Set **Ratio, W =** : items, and click button.



- 4) Select "Reverse Weight" item, if necessary.
If the weight is applied in the direction contrary to your intention, select "Reverse Weight" menu item. Then, the direction is reversed.

■ Dividing curves as a whole

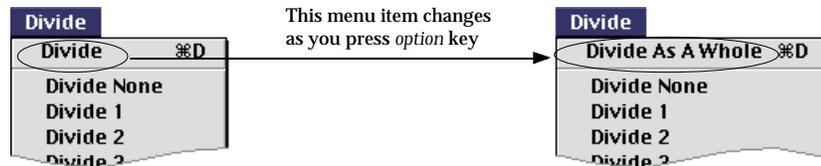
A group of selected curves may be divided as a whole. In this case, the number of divisions is applied as the total number of divisions in all the selected curves. This can be achieved by the following steps.

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

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

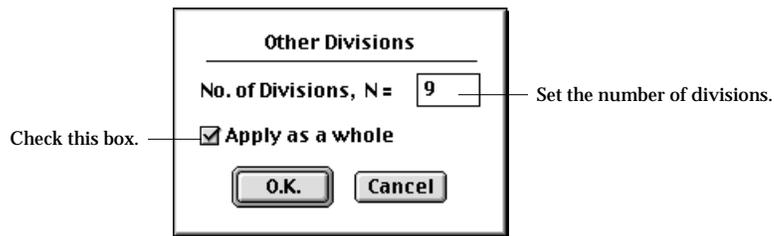
- "Divide As a Whole" item

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

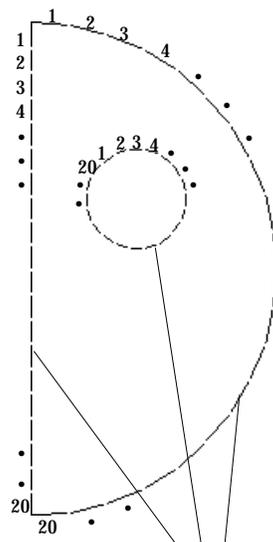


- "Other Divisions..." item

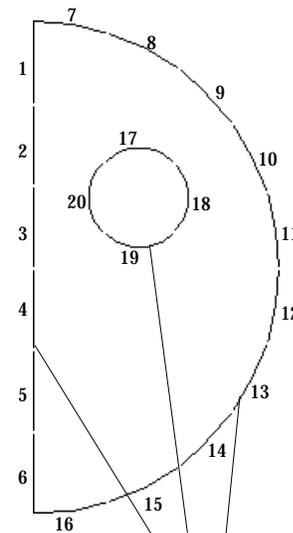
"Other Divisions" dialog box appears. This dialog is useful for dividing the curve(s) by a number of divisions not shown as a menu item. Set **No. of Divisions**, N = item, and click button. Be sure that **Apply as a whole** item is checked.



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



Each of these 3 curves are individually divide into 20.



These 3 curves are divide into 20 segments as a whole

< Comparison of dividing curves individually and as a whole >

■ Setting the number of divisions

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

■ Setting the weight of division density

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

■ Removing divisions from divided curves

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

■ Dividing curves using F keys

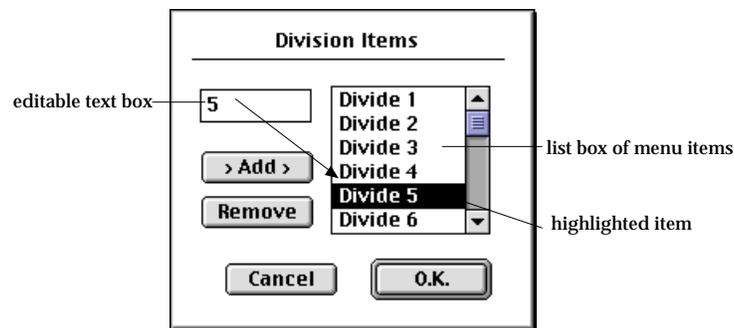
A single curve or a group of selected curves may be divided using F keys(F1 - F24). The curve division is achieved simply by pressing one of the F keys after selecting curve(s). The F number corresponds to the number of divisions. For example, pressing F3 will divide the selected curve(s) into 3. If key (Windows : Alt key) is pressed at the same time, "Apply as a whole" option applies.

Changing **Divide** Menu Items

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

■ Modifying the menu items for number of divisions

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



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

In order to change a menu item,

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

In order to add a menu item,

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

In order to delete a menu item,

- 1) Click the item in the list box.

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

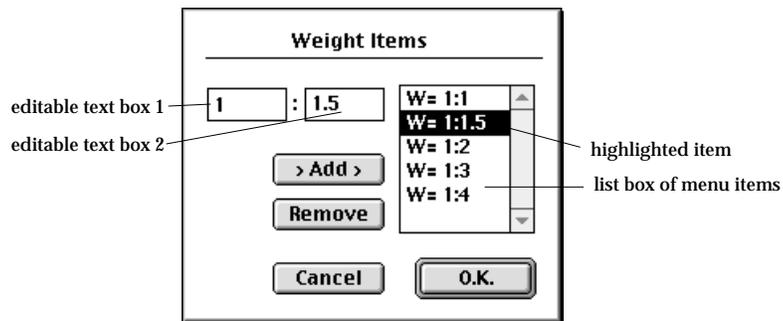
- 2) Click **Remove** button.

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

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

■ Changing the menu items for weight of division density

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



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