Chapter 4

Mesh Generation

Chapter 4   Mesh Generation

Chapter 4   Mesh Generation

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

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

General procedure of mesh generation

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

1) Activate one of the mesh generation commands.
   Choose a mesh generation command from **Mesh** menu. A corresponding mesh generation dialog box appears. For some mesh generation commands, the proper object selection tool is automatically activated. For example, curve selection tool 🔧 is always activated when "4 edges" surface mesh generation command is issued.

2) Select curves or surface meshes, which are used as a seed of mesh generation.
   For example, a surface mesh may be generated by revolving a selected curve about a specified axis. In this case, the selected curve is used as a seed of the mesh generation.

3) Set the dialog items.
   The dialog has various selectable items to set the shape of element, number of nodes, input methods and so on. Some input text items are provided for defining characteristics of the mesh generation.

4) Input axis or path if necessary.
   The axis of rotation or path of sweeping is needed for some mesh generation schemes, and may be determined by selecting an existing curve or creating a new one.

5) Press appropriate buttons in the dialog.
   Buttons vary depending on the input stage. When everything is ready for

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

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

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

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

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

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

Mesh

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

Surface Mesh Generation

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

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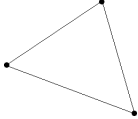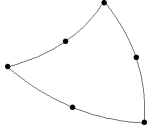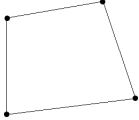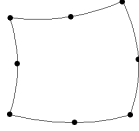- a seed mesh for generating a volume mesh.

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

- automatic triangulation
- mapping
- sweeping
- duplication
- projection

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

< Element types in surface meshes >

3 node triangle (T3)	6 node triangle (T6)
4 node quadrilateral (Q4)	8 node quadrilateral (Q8)

Automatic triangulation

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

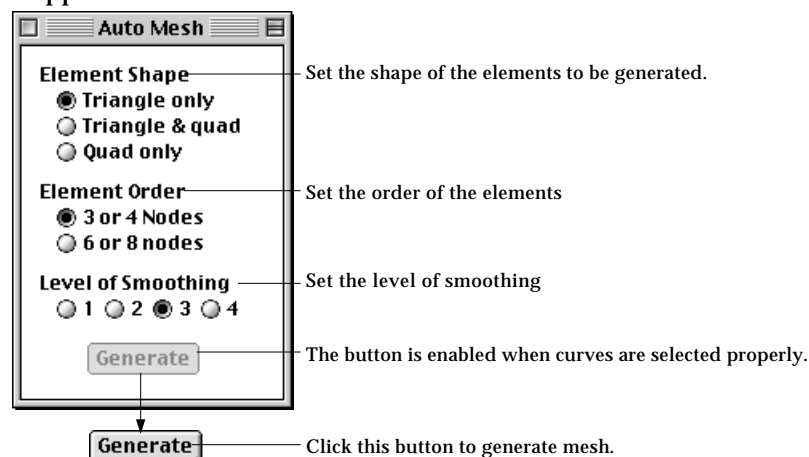### ■ Generating mesh on a plane by automatic triangulation
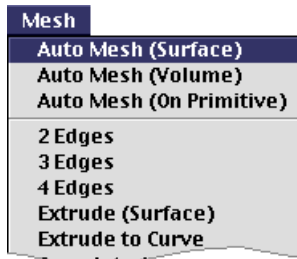
The easiest way of generating mesh on a plane is "unconstrained" automatic triangulation. An arbitrarily shaped surface mesh is generated simply by designating the curves of the mesh boundary and issuing a mesh generation command. A curved surface as well as a plane may be meshed by this method. However,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

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

1) Choose "Auto Mesh" from **Mesh** menu.
   The curve selection tool 🔖 is automatically activated, and "Auto Tri" dialog box appears.

Set the shape of the elements to be generated.

Set the order of the elements

Set the level of smoothing

The button is enabled when curves are selected properly.

Click this button to generate mesh.

2) Set the element shape using "Auto Tri" dialog.
   Click one of the buttons for element shape. There are following 3 options.
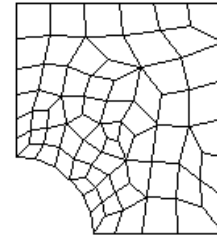   • "Triangle only" : Only triangular elements are generated.

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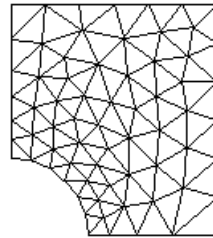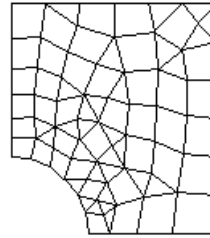
"Triangle only"   "Triangle and quad"   "Quad only"

< Mesh generated using different options of element type>

3) Set the element order.

   Click one of the buttons for the order of elements. Either linear or quadratic order can be selected.
   - "3 or 4 nodes" : Linear elements (3 node triangle or 4 node quadrangle) are generated.
   - "6 or 8 nodes" : Quadratic elements (6 node triangle or 8 node quadrangle) are generated.

4) Set the level of smoothing using "Auto Tri" dialog.

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

5) Select curves.

   All the selected curves should be divided. Some of the selected curves are used in defining the boundary of the mesh region. Others may be needed to control the mesh density and element boundaries inside the region.

6) Click [Generate] button.

   A surface mesh is generated by automatic triangulation, if the selected curves are compatible for automatic triangulation as described in the next section. Otherwise, the action is ignored with a message notifying that automatic triangulation cannot not be executed with the selected curves.

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

Selected curves

Boundary of the mesh region   Generated mesh
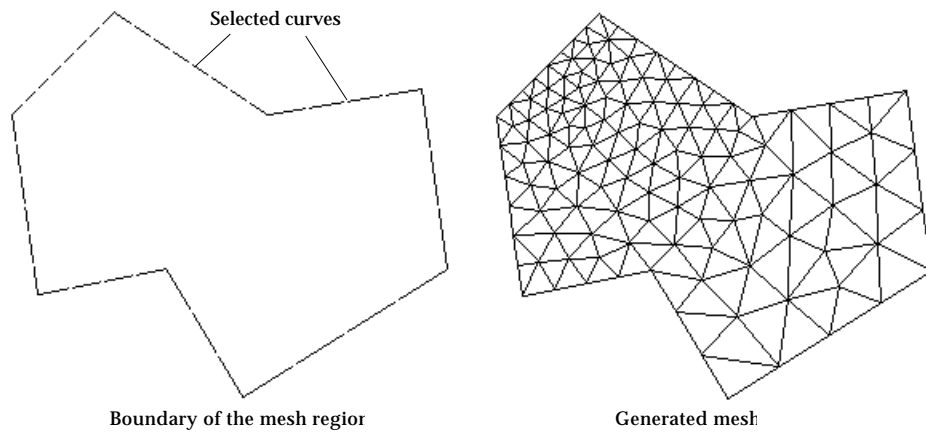
< Example of automatic triangulation >

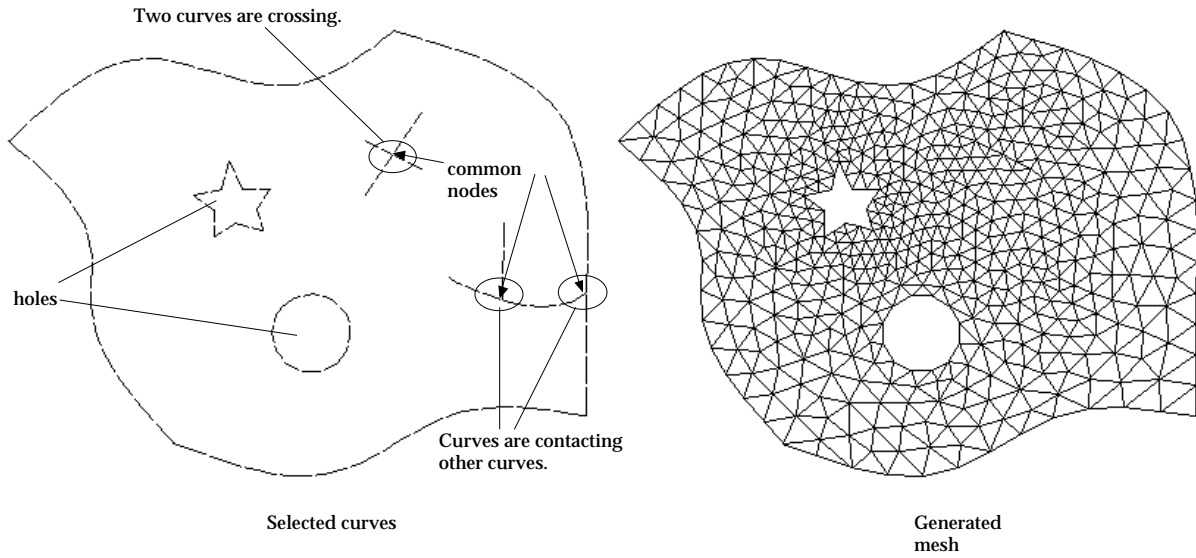## ■ Setting a compatible region for automatic triangulation

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

- The outer boundary of the region should be enclosed by serially connected curves.
- All the selected curves should be in one united region.
- The region boundary may be either convex or concave.
- The region may contain holes or control curves inside the boundary.
- Curves inside the region may cross or contact with each other, but they must have a common node at each crossing or contacting point.
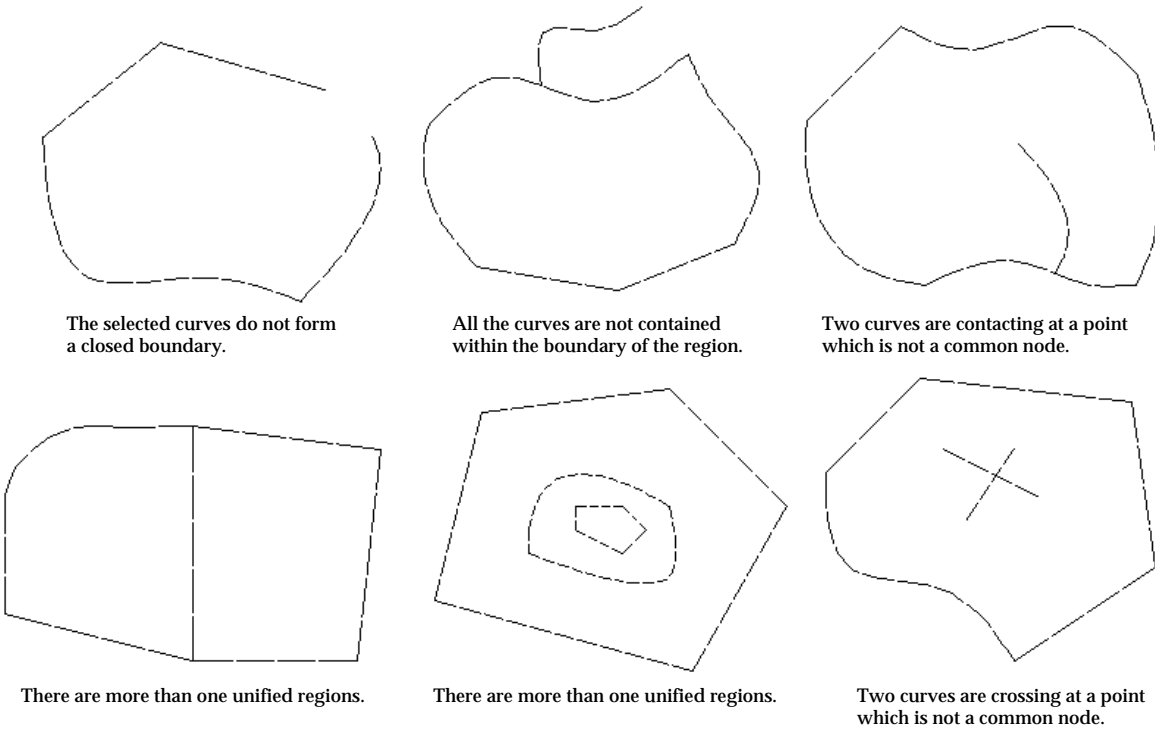
If you issue the automatic triangulation command with improper curve selection, you may receive one of the responses shown from VisualFEA depending on the state of the curve selection.

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

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

Two curves are crossing.

common nodes

holes

Curves are contacting other curves.

Selected curves

Generated mesh

< An example of compatible region for automatic triangulation >

The selected curves do not form a closed boundary.

All the curves are not contained within the boundary of the region.

Two curves are contacting at a point which is not a common node.

There are more than one unified regions.

There are more than one unified regions.

Two curves are crossing at a point which is not a common node.

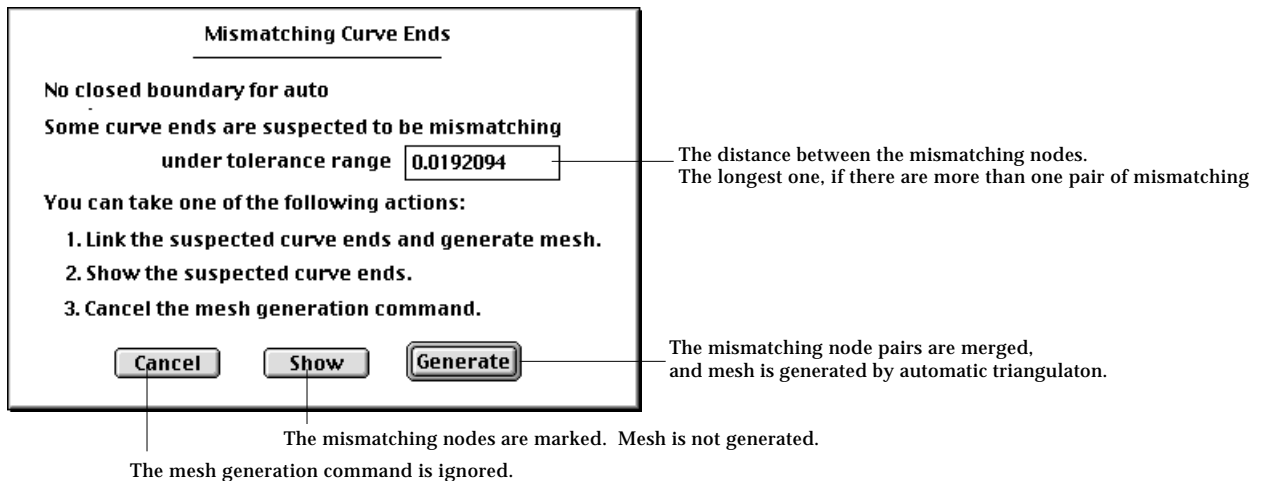**< Examples of incompatible region for automatic triangulation >**

■ Handling the incompatibility due to mismatching curve ends

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

Mismatching Curve Ends

No closed boundary for auto

Some curve ends are suspected to be mismatching

    under tolerance range `0.0192094` —— The distance between the mismatching nodes.
The longest one, if there are more than one pair of mismatching

You can take one of the following actions:

  1. Link the suspected curve ends and generate mesh.
  2. Show the suspected curve ends.
  3. Cancel the mesh generation command.

[Cancel]   [Show]   [Generate] —— The mismatching node pairs are merged,
and mesh is generated by automatic triangulaton.

The mismatching nodes are marked. Mesh is not generated.
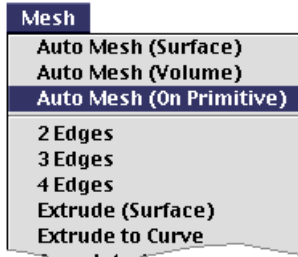
The mesh generation command is ignored.

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

- [Cancel] : cancel the mesh generation command.
- [Show] : display the suspected mismatching curve ends.
- [Generate] : Close the gap between the mismatching curve ends so that the boundary curves are close. You may set the tolerance range of the gap to be closed.
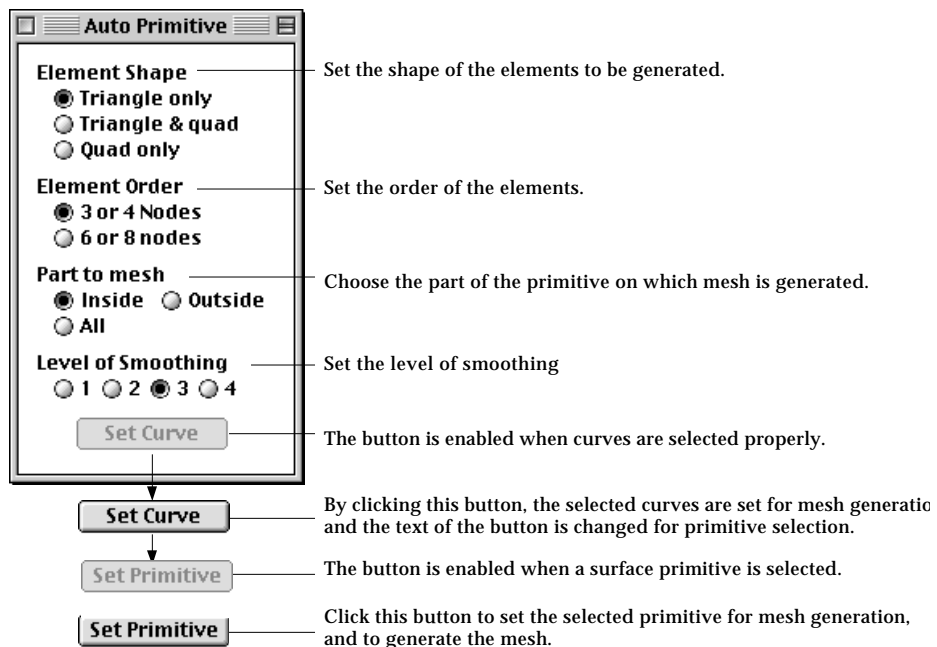
The suspected mismatching curve ends are indicated by a small red rectangle.
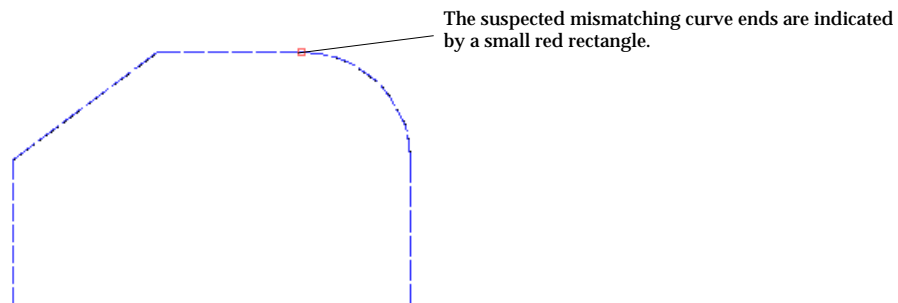
< A example of showing mismatching curve ends >

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

Mesh
| Auto Mesh (Surface) |
| Auto Mesh (Volume) |
| **Auto Mesh (On Primitive)** |
| 2 Edges |
| 3 Edges |
| 4 Edges |
| Extrude (Surface) |
| Extrude to Curve |

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

1) Choose "Auto Mesh on Primitive" from **Mesh** menu.
   The curve selection tool 🔼 is automatically activated, and "Auto Primitive" dialog box appears.

Auto Primitive

Element Shape — Set the shape of the elements to be generated.
- ◉ Triangle only
- ○ Triangle & quad
- ○ Quad only

Element Order — Set the order of the elements.
- ◉ 3 or 4 Nodes
- ○ 6 or 8 nodes

Part to mesh — Choose the part of the primitive on which mesh is generated.
- ◉ Inside ○ Outside
- ○ All

Level of Smoothing — Set the level of smoothing
- ○ 1 ○ 2 ◉ 3 ○ 4

Set Curve — The button is enabled when curves are selected properly.

Set Curve — By clicking this button, the selected curves are set for mesh generation and the text of the button is changed for primitive selection.

Set Primitive — The button is enabled when a surface primitive is selected.

Set Primitive — Click this button to set the selected primitive for mesh generation, and to generate the mesh.

2) Set the element shape.
   Click one of the buttons for element shape. There are 3 options.
   - "Triangle only" : Only triangular elements are generated.
   - "Triangle and quad" : The generated mesh is filled with mixture of triangular and/or quadrilateral elements.
   - "Quad only" : Only quadrilateral elements are generated. In order to use this option, the total number of divisions in the selected curves should be even.

3) Set the element order.
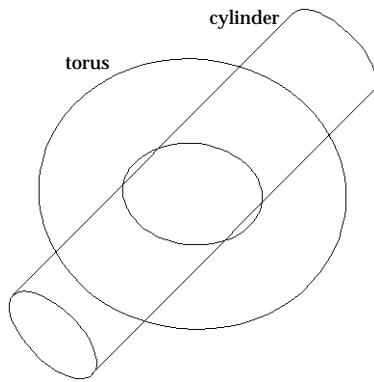   Click one of the buttons for the order of elements. Either linear or quadratic order can be selected.
   - "3 or 4 nodes" : Linear elements (3 node triangle or 4 node quadrangle) are generated.
   - "6 or 8 nodes" : Quadratic elements (6 node triangle or 8 node quadrangle) are generated.

4) Choose the part of the primitive on which mesh is to be generated.
A mesh may be generated inside, outside or both sides of the region
boundary of the surface primitive. Choose the part by turning on one of the
radio buttons "Inside", "Outside" and "All".

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

6) Select curves.
All the selected curves should be divided. Some of the selected curves are
used in defining the boundary of the mesh region. Others may be needed to
control the mesh density and element boundaries inside the region. The
initially dimmed [Set Curve] button is enabled, when one or more curves
are selected.

7) Click [**Set Curve**] button.
The selected curves are reserved for mesh generation, and the button changes
into [Set Primitive] indicating that selecting surface primitive is expected in the
next step. The primitive selection tool [🔣] is automatically activated.

8) Select a surface primitive.
The selected surface primitive is highlighted in bright red color. Only one
surface primitive should be selected. The dimmed [Set Primitive] button is
enabled when a surface primitive is selected.

9) Click [**Set Primitive**] button.
A mesh is generated on the selected surface primitive. [**Set Primitive**] button is
restored to [Set Curve] button. It is now ready for inputting another surface
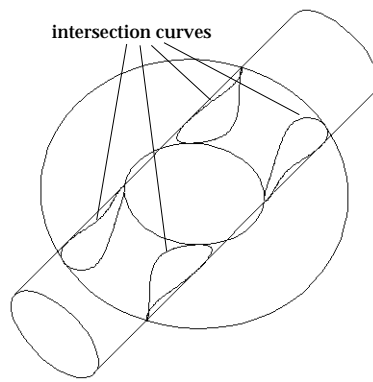mesh.

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

curve   surface primitive

"Inside"   "Outside"   "All"
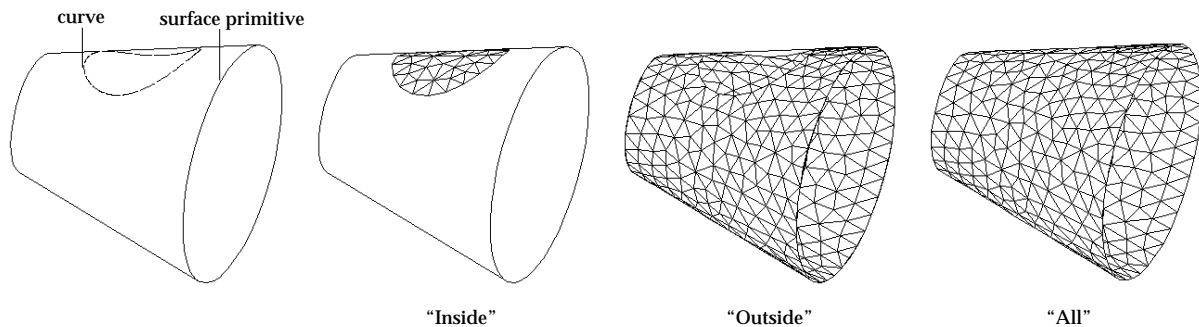
< Comparison of meshes generated using different options of part to mesh >

cylinder

torus

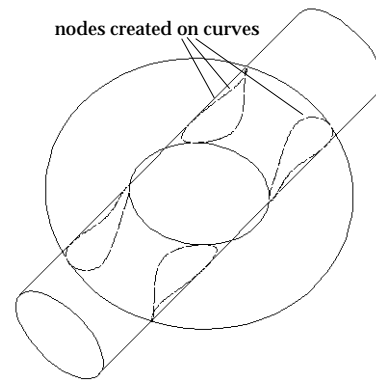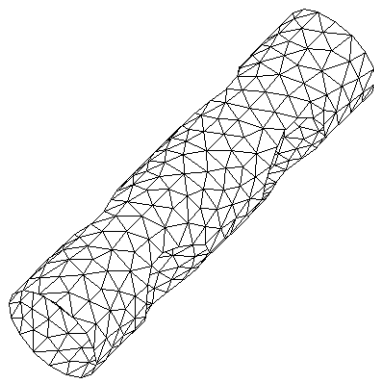intersection curves

nodes created on curves
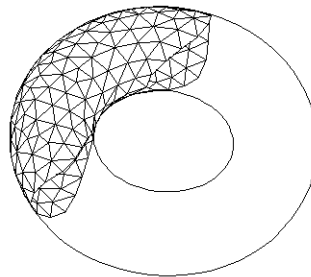
Create surface primitives.

Obtain intersection curves of
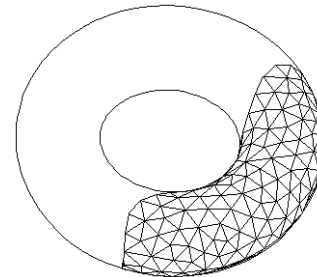the two surface primitives.
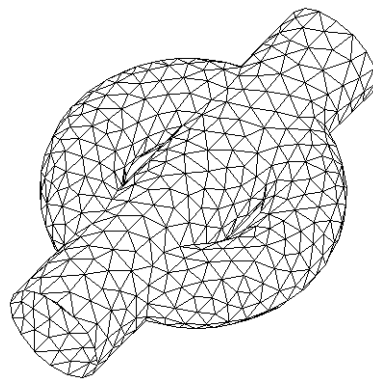
Divide the curves.

Generate mesh on the cylinder.

Generate mesh on one part of
the torus.

Generate mesh on the other
part of the torus.

Completed mesh

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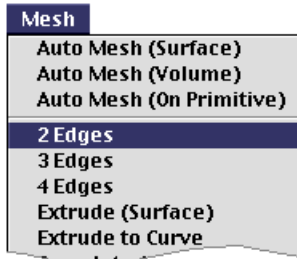
Surface mesh generation by mapping

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

- **2 edges**
- **3 edges**
- **4 edges**

The nodal points of the generated mesh are computed by mapping techniques called "lofting", "triangular mapping" or "transfinite mapping" respectively for each of the above 3 types.   These techniques are not explained in detail here. The commands generating mesh by mapping are provided as menu items in   **Mesh** menu.
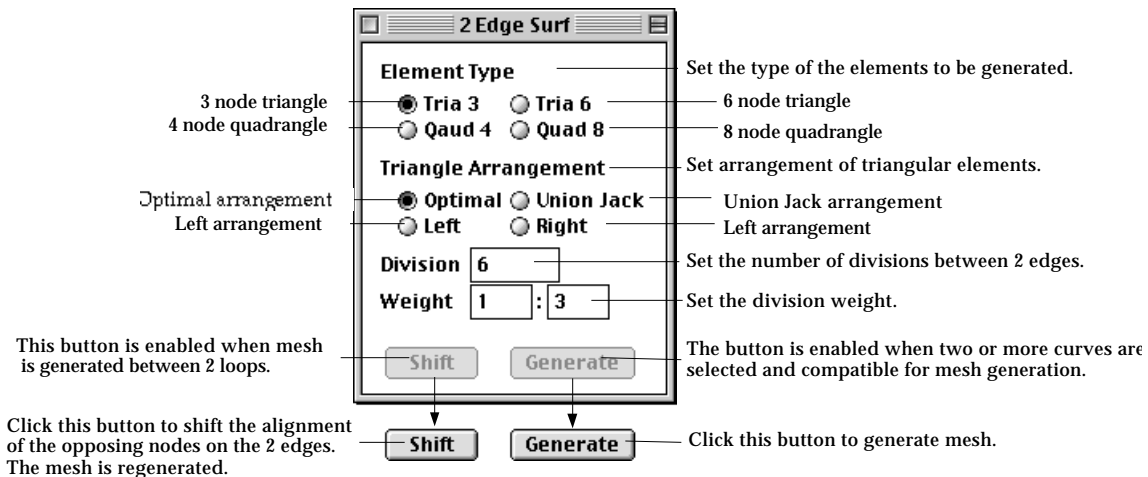
■ Generating mesh using 2 edges

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)
2 Edges
3 Edges
4 Edges
Extrude (Surface)
Extrude to Curve

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

1) Choose "2 Edges" from   **Mesh**   menu.

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

- 3 node triangle
- 4 node quadrangle
- Optimal arrangement
- Left arrangement
- This button is enabled when mesh is generated between 2 loops.
- Click this button to shift the alignment of the opposing nodes on the 2 edges. The mesh is regenerated.

2 Edge Surf
- **Element Type** — Set the type of the elements to be generated.
- Tria 3   Tria 6 — 6 node triangle
- Qaud 4   Quad 8 — 8 node quadrangle
- **Triangle Arrangement** — Set arrangement of triangular elements.
- Optimal   Union Jack — Union Jack arrangement
- Left   Right — Left arrangement
- **Division** 6 — Set the number of divisions between 2 edges.
- **Weight** 1 : 3 — Set the division weight.
- Shift   Generate — The button is enabled when two or more curves are selected and compatible for mesh generation.
- **Generate** — Click this button to generate mesh.

2) Set the element type.

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

3) Select the type of arrangement for triangular elements.

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

"Optimal"   "Union Jack"

"Left"   "Right"

<Arrangement of triangular elements>
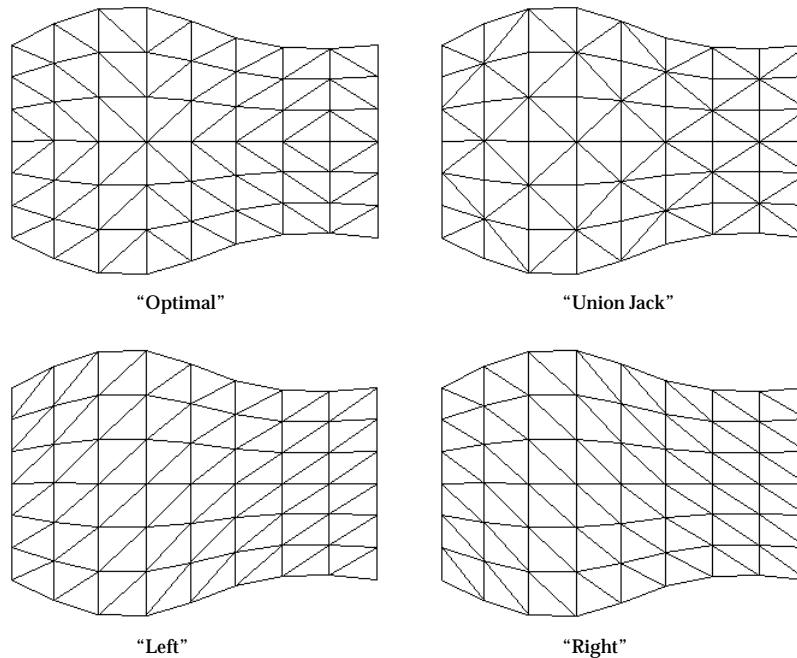
4) Set the number of divisions between two edges.
   It is the same as designating how many rows of elements to be generated between two edges. The number of divisions is denoted by n in the following example.
5) Set the weight of division density.
   The weight of division density is entered in the form of $w_1{:}w_n$, which is the ratio between the length of the first division l_1 and that of the last l_n as indicated in the figure below.
6) Select curves forming two edges.
   All the selected curves should be divided, and form two edges, each of which consists of one curve or serially connected curves.   The total divisions on each edge should be equal.   `Generate` button is enabled when two edges are formed.
   The order of selecting curves does not make any difference in the resulting mesh generation. It is not necessary to designate the direction of the edges, although most other software asks to match the directions of the two edges. VisualFEA does it automatically.
   *The division is directed from the first edge to the second. Which one is the first is*

> *determined by the input order of curves forming the edges.*

7) Click ⎣**Generate**⎦ button.

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

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

Selected curves   Generated mesh
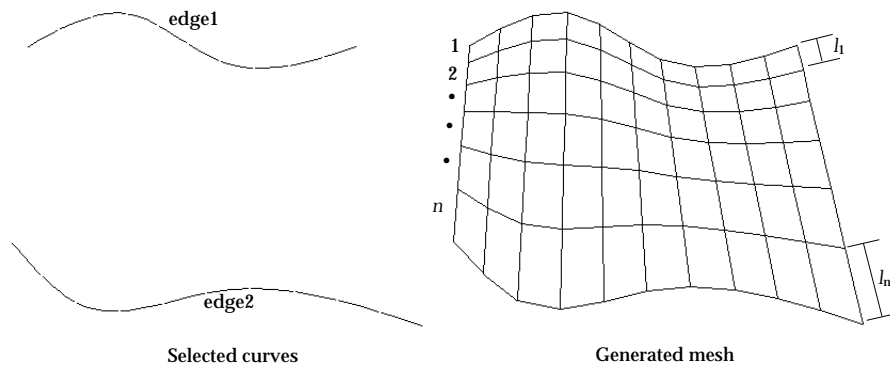
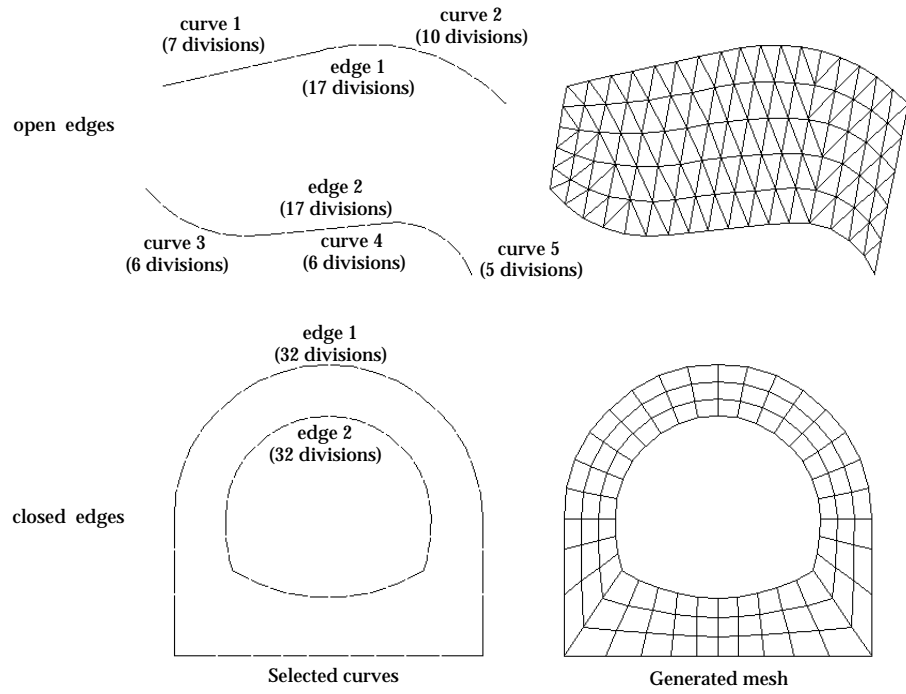< Example of surface mesh generation between 2 edges >

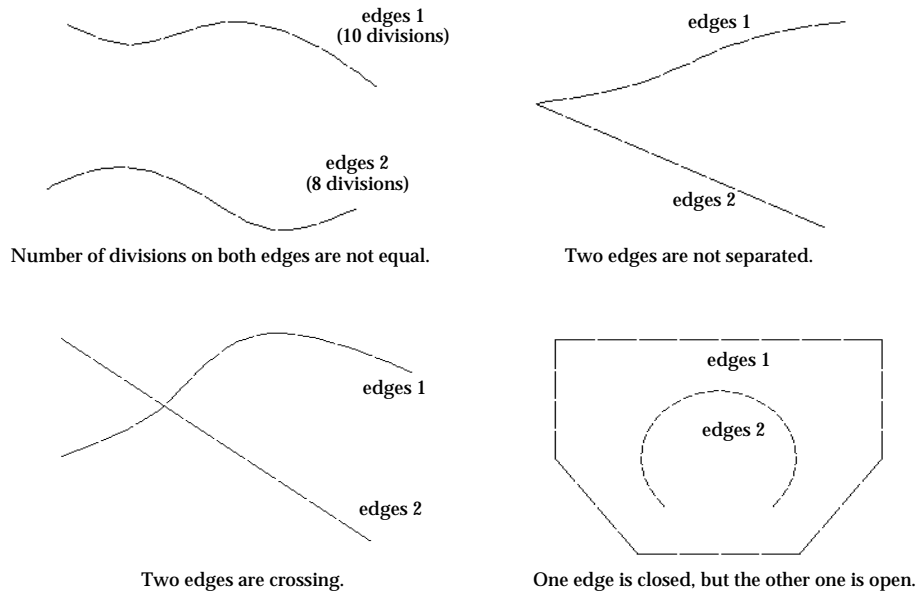## ■ Setting 2 edges compatible for mesh generation

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

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

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

curve 1
(7 divisions)

curve 2
(10 divisions)

edge 1
(17 divisions)

open edges

edge 2
(17 divisions)

curve 3
(6 divisions)

curve 4
(6 divisions)

curve 5
(5 divisions)

edge 1
(32 divisions)

edge 2
(32 divisions)

closed edges

Selected curves

Generated mesh

<Examples of 2 edge formation compatible for mesh generation>

edges 1
(10 divisions)

edges 1

edges 2
(8 divisions)

edges 2

Number of divisions on both edges are not equal.

Two edges are not separated.

edges 1

edges 2

edges 1

edges 2

Two edges are crossing.

One edge is closed, but the other one is open.

<Examples of 2 edge formation incompatible for mesh generation>

■ Shifting alignment of node pairing on 2 edges

If the two edges form closed loops respectively, the one-to-one pairing of the nodes on the edges are optimally determined by the software. However, it is rare but possible that the automatic pairing results in generation of elements with distorted or improper shape. The mesh shape can be improved by manually adjusting the alignment of the node pairing. This is achieved by pressing Shift button, in "2 Edge Surf" dialog, which is enabled when the mesh is generated. Shift button is valid only in case both edges form closed loops respectively. There are 7 different alignments, each of which is obtained one after another as the button is pressed consecutively. The sequence of the alignment is as follows:
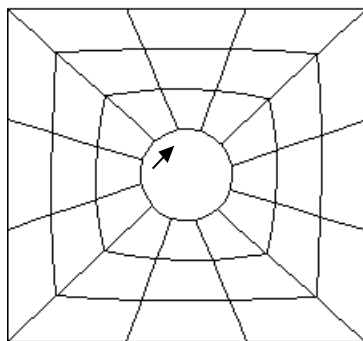
1) The outer loop shifts clockwise.
2) The inner loop shifts clockwise.
3) Both the inner loop and the outer loop shift clockwise.
4) The outer loop shifts counter-clockwise.
5) The inner loop shifts counter-clockwise.
6) Both the inner loop and the outer loop shift counter-clockwise.
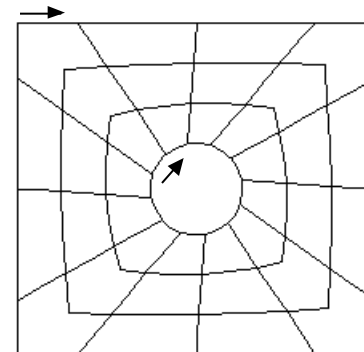7) The initial alignment is obtained again.
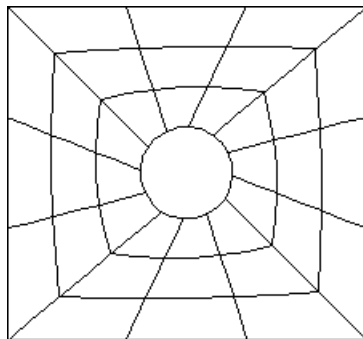
Initially generated mesh.
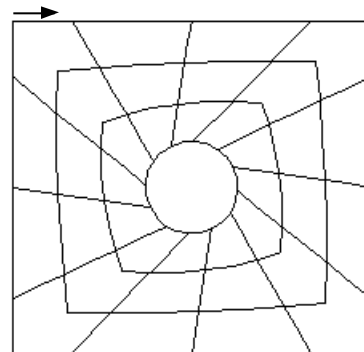
The outer loop shifts clockwise.
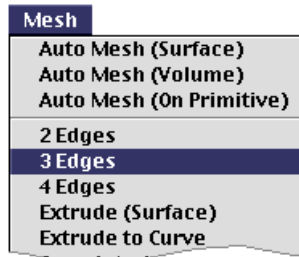("Shift" button is clicked once)

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

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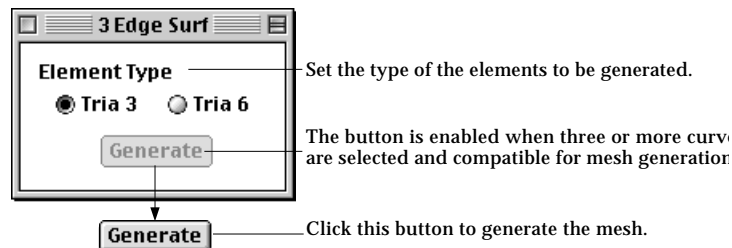("Shift" button is clicked three times)

< Example of alignment shifting in looped 2 edges >

■ Generating mesh using 3 edges

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)
2 Edges
3 Edges
4 Edges
Extrude (Surface)
Extrude to Curve

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

1) Choose "3 Edges" from **Mesh** menu.
The curve selection tool � is automatically activated, and "3 Edge Surf" dialog box appears.

3 Edge Surf

Element Type ——— Set the type of the elements to be generated.
◉ Tria 3   ○ Tria 6

[Generate] ——— The button is enabled when three or more curv[e] are selected and compatible for mesh generation

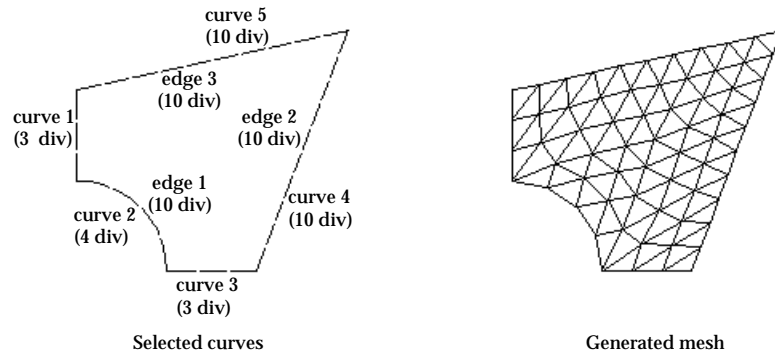[Generate] ——— Click this button to generate the mesh.

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

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

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

curve 5
(10 div)

edge 3
(10 div)

curve 1
(3 div)

edge 2
(10 div)

edge 1
(10 div)

curve 4
(10 div)

curve 2
(4 div)

curve 3
(3 div)

Selected curves   Generated mesh

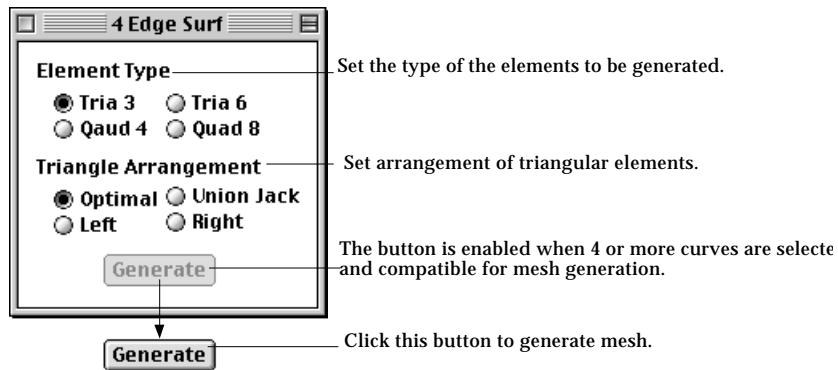< Example of surface mesh generation enclosed by 3 edges >

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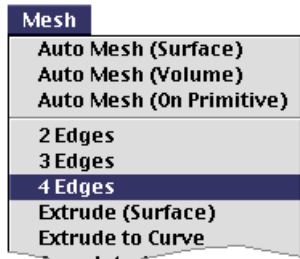
A surface mesh can be generated using 4 edges formed by a series of curves. The region enclosed by the 4 edges is filled with triangular or quadrilateral elements depending on your choice. The coordinates of the nodes on the mesh are determined by interpolating the nodal points on the 4 edges using transfinite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

1) Choose "4 Edges" from **Mesh** menu.
The curve selection tool ▣ is automatically activated, and "4 Edge Surf" dialog box appears.

Set the type of the elements to be generated.

Set arrangement of triangular elements.

The button is enabled when 4 or more curves are selected and compatible for mesh generation.

Click this button to generate mesh.

2) Set the element type.
You may choose one of 3 node triangle, 6 node triangle, 4 node quadrangle, and 8 node quadrangle element as the element type.
3) Select the type of arrangement for triangular elements.
The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section "Generating mesh using 2 edges".

4) Select curves forming 4 edges.

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

5) Click [Generate] button.

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

Selected curves   Generated mesh
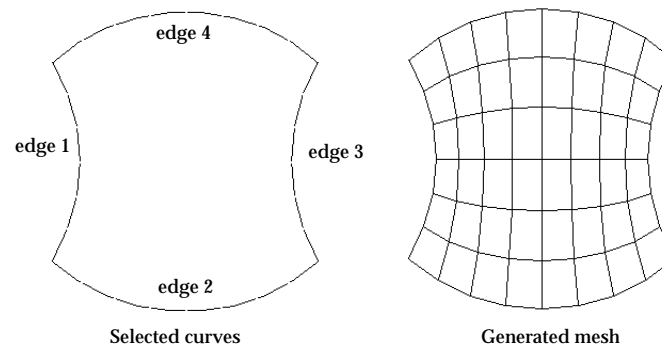
< Example of surface mesh generation enclosed by 4 edges >

■ **Setting 4 edges compatible for mesh generation**

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

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

  The 4 edges may be regarded as boundary lines of a quadrilateral region with two pairs of edges facing opposite to each other.
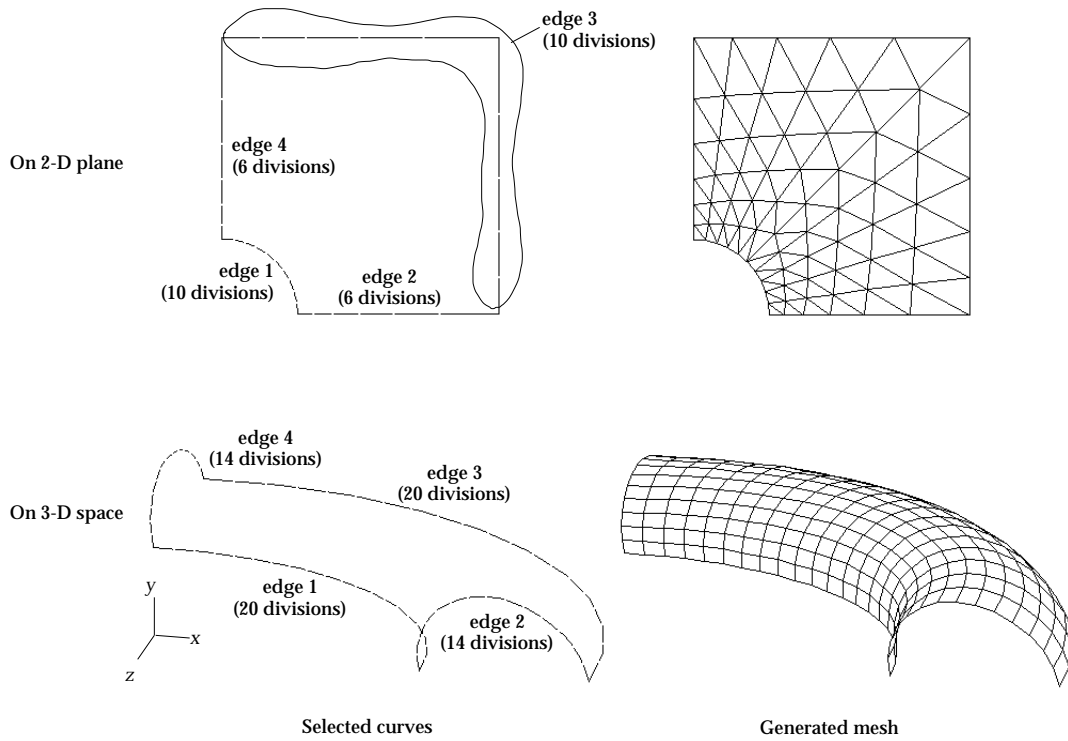
- **Each pair of edges should have an equal number of nodes on them.**

  Edges are constructed automatically by VisualFEA so that each pair of edges have equal number of nodes on them. If such construction is not possible, mesh generation is aborted after a message "Incompatible curve selection for 4 edge surface."
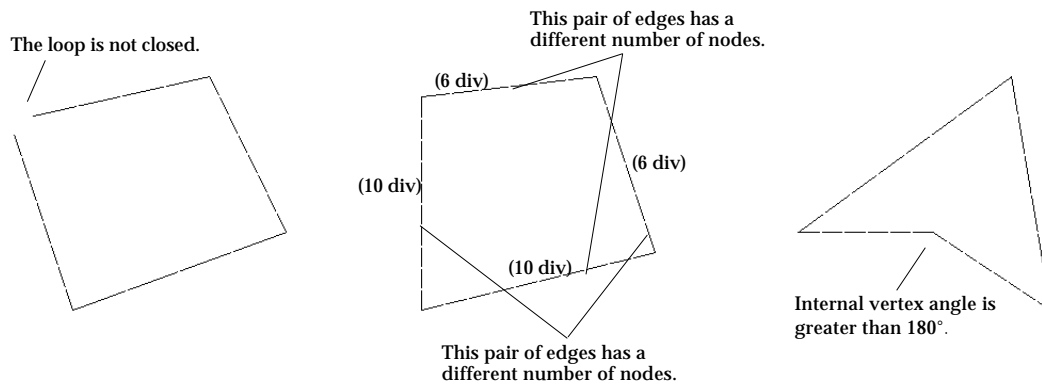
- **All the internal angles of vertexes made by two adjacent edges should be less than 180˚.**

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

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

edge 3
(10 divisions)

On 2-D plane

edge 4
(6 divisions)

edge 1
(10 divisions)

edge 2
(6 divisions)

edge 4
(14 divisions)

edge 3
(20 divisions)

On 3-D space

y

edge 1
(20 divisions)

edge 2
(14 divisions)

x

z

Selected curves

Generated mesh

<Examples of 4 edge formation compatible for mesh generation>

The loop is not closed.

This pair of edges has a
different number of nodes.

(6 div)

(6 div)

(10 div)

(10 div)

Internal vertex angle is
greater than 180°.

This pair of edges has a
different number of nodes.

**<Examples of 4 edge formation incompatible for mesh generation>**

Surface mesh generation by sweeping operations

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

- Extrusion
- Translation
- Revolution
- Twisting

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

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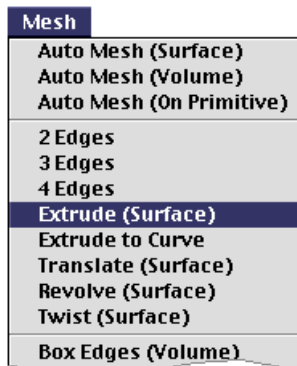
A surface mesh can be generated by extruding selected curves up to the specified height and in the specified direction. The height of the extrusion is entered using the "Extr Surf" dialog.

1) Choose "Extrude(Surface)" from **Mesh** menu.
   The curve selection tool [▲] is automatically activated, and "Extr Surf" dialog box appears.

Mesh
- Auto Mesh (Surface)
- Auto Mesh (Volume)
- Auto Mesh (On Primitive)
- 2 Edges
- 3 Edges
- 4 Edges
- Extrude (Surface)
- Extrude to Curve
- Translate (Surface)
- Revolve (Surface)
- Twist (Surface)
- Box Edges (Volume)

Extr Surf

Element Type — Set the type of the elements to be generated.
- ◉ Tria 3   ○ Tria 6
- ○ Qaud 4   ○ Quad 8

Triangle Arrangement — Set arrangement of triangular elements.
- ◉ Optimal   ○ Union Jack
- ○ Left   ○ Right

Extruding Direction — Set the direction of extrusion.
- ○ X   ○ Y   ○ Z
- ◉ Normal

Height [10] — Set the extrusion height.
Division [8] — Set the number of divisions for extrusion.
Weight [1] : [2] — Set the division weight.

The button is enabled when the mesh is generated. — [Reverse]   [Generate] — The button is enabled when one or more curves are selected and compatible for mesh generation.

Click this button to reverse the direction of extrusion. — [Reverse]   [Generate] — Click this button to generate mesh.

2) Set the element type.

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

3) Select the type of arrangement for triangular elements.

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

4) Set the direction of extrusion.

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

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

5) Enter the height of extrusion.

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

6) Enter the number of divisions for extrusion.

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

7) Enter the weight of division density.

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

8) Select curves forming seed curves for extrusion.

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

9) Click [Generate] button.

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

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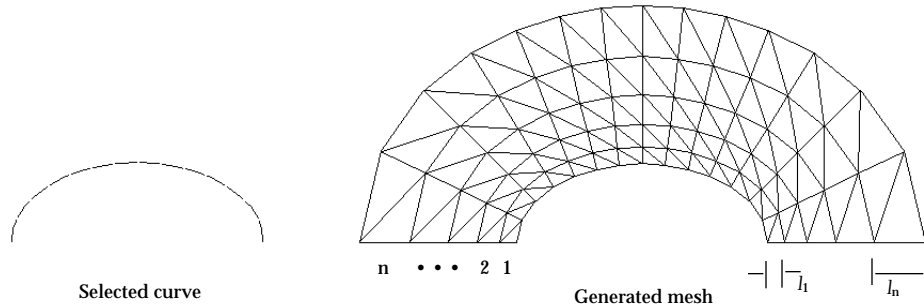
< A surface mesh generated by extrusion in normal direction >

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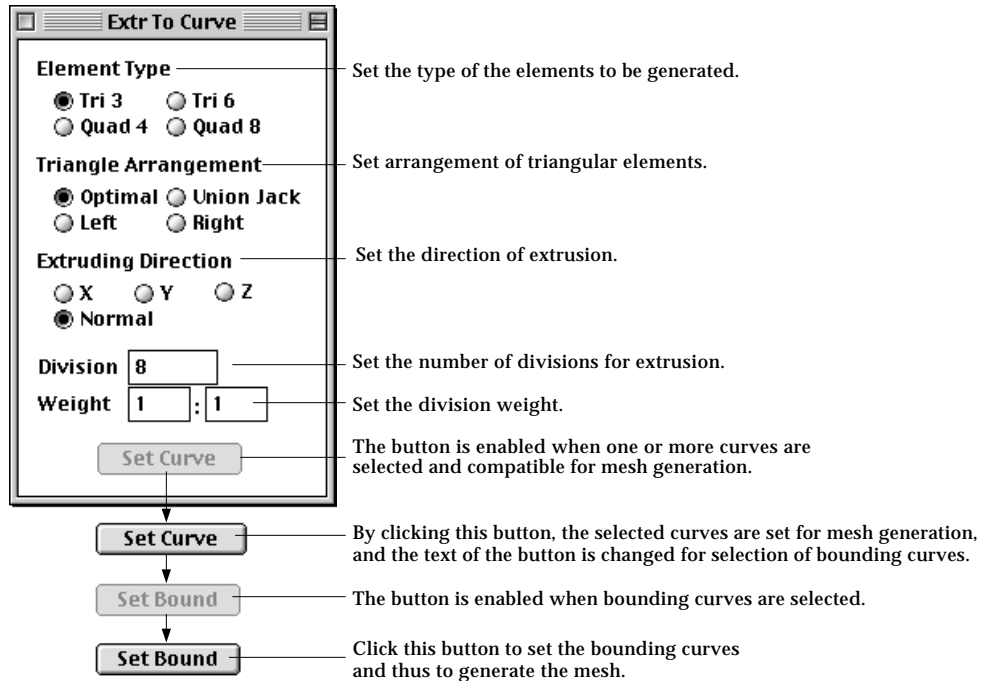

Selected curve   Generated mesh
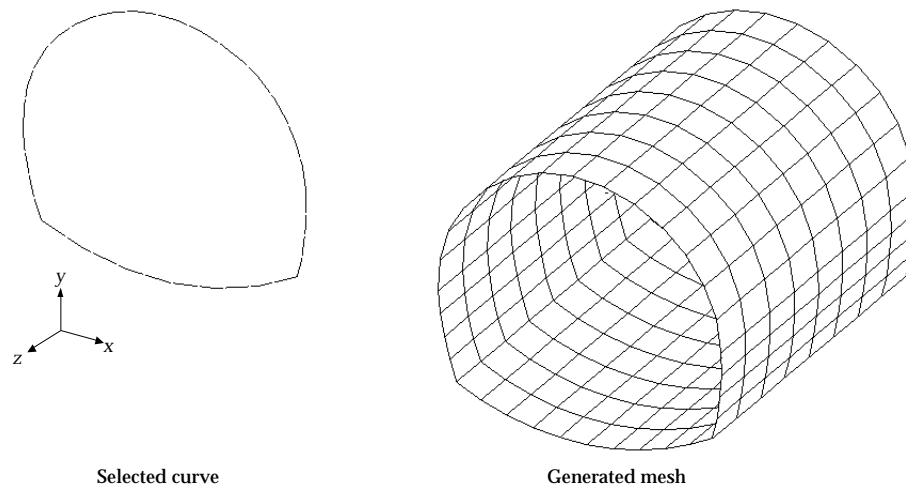
< A 3-D surface mesh generated by extrusion in an out-of-plane direction >

■ Generating mesh by extrusion up to bounding curves

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

1) Choose "Extrude to Curve" from **Mesh** menu.

The curve selection tool [icon] is automatically activated, and "Extr to Curve" dialog box appears.

Extr To Curve	
Element Type	Set the type of the elements to be generated.
● Tri 3 ○ Tri 6	
○ Quad 4 ○ Quad 8	
Triangle Arrangement	Set arrangement of triangular elements.
● Optimal ○ Union Jack	
○ Left ○ Right	
Extruding Direction	Set the direction of extrusion.
○ X ○ Y ○ Z	
● Normal	
Division `8`	Set the number of divisions for extrusion.
Weight `1` : `1`	Set the division weight.
Set Curve	The button is enabled when one or more curves are selected and compatible for mesh generation.

Set Curve — By clicking this button, the selected curves are set for mesh generation, and the text of the button is changed for selection of bounding curves.

Set Bound — The button is enabled when bounding curves are selected.

Set Bound — Click this button to set the bounding curves and thus to generate the mesh.

2) Set the element type.

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

3) Select the type of arrangement for triangular elements.

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

4) Set the direction of extrusion.

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

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

5) Set the number of divisions for extrusion.

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

6) Set the weight of division density.

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

7) Select curves forming seed curves for extrusion.

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

8) Click [Set Curve] button.

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

9) Select the bounding curves.

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

10) Click [Set Bound] button.

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

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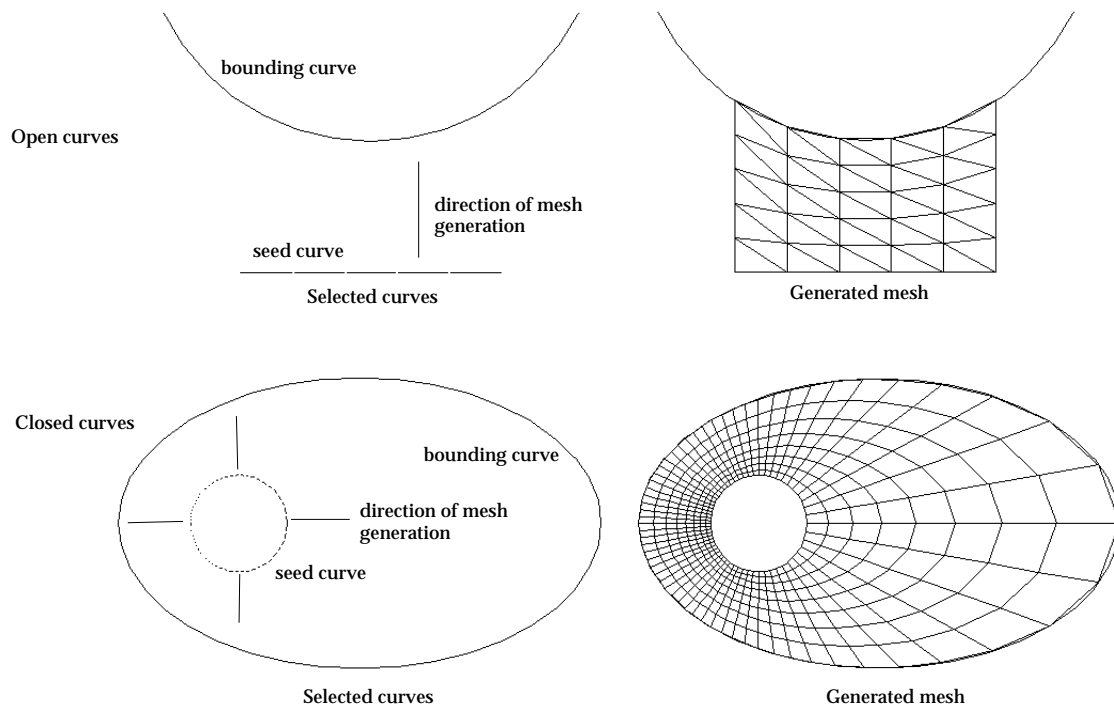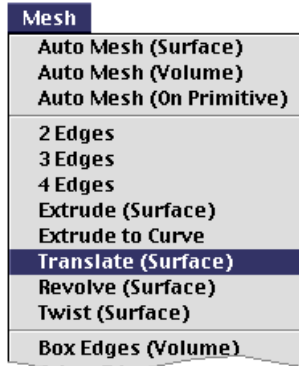
<Mesh generated by extrusion up to bounding curves>

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

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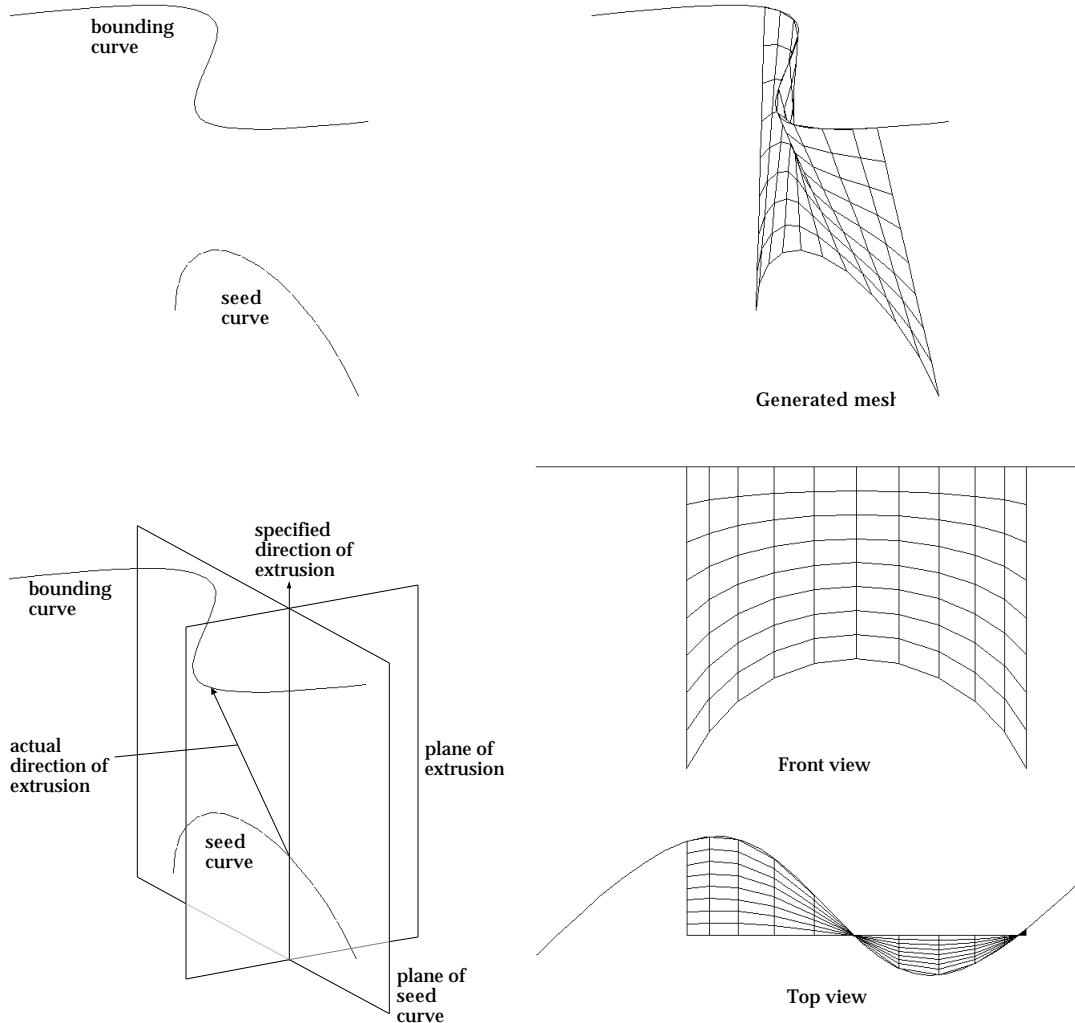
< Determination of extruding direction in case the seed curve
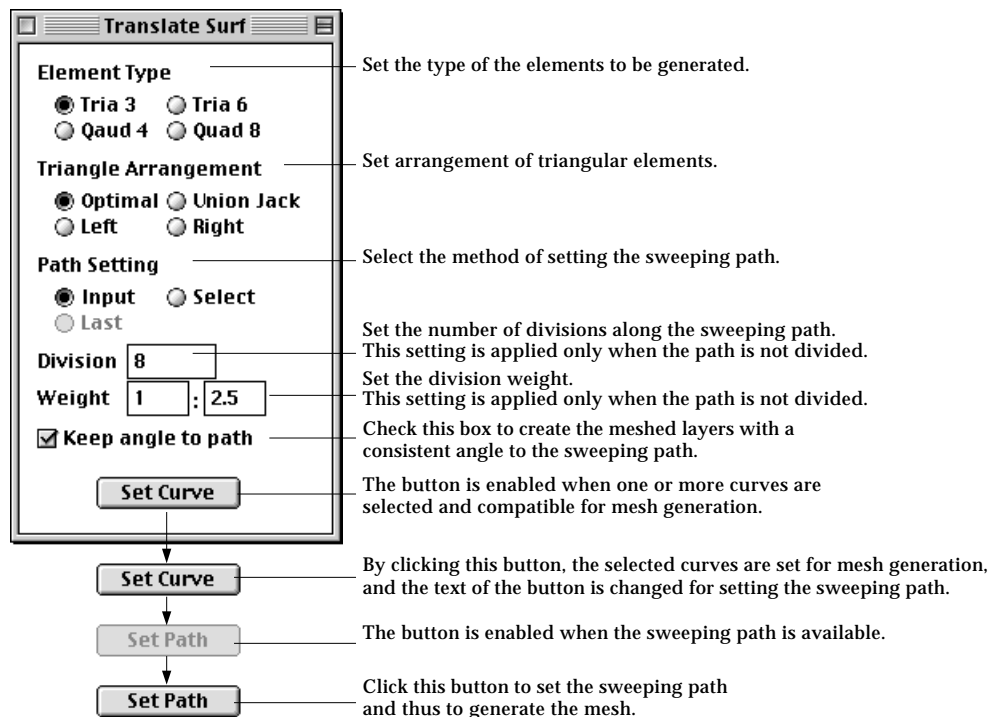and the bounding curve are not on the same plane >

■ Generating mesh by translation

Mesh

- Auto Mesh (Surface)
- Auto Mesh (Volume)
- Auto Mesh (On Primitive)
- 2 Edges
- 3 Edges
- 4 Edges
- Extrude (Surface)
- Extrude to Curve
- **Translate (Surface)**
- Revolve (Surface)
- Twist (Surface)
- Box Edges (Volume)

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

1) Choose "Translate(Surface)" from **Mesh** menu.

The curve selection tool [▸] is automatically activated, and "Translate Surf " dialog box appears.

Translate Surf

Element Type
- ◉ Tria 3   ○ Tria 6
- ○ Qaud 4   ○ Quad 8

— Set the type of the elements to be generated.

Triangle Arrangement
- ◉ Optimal ○ Union Jack
- ○ Left   ○ Right

— Set arrangement of triangular elements.

Path Setting
- ◉ Input   ○ Select
- ○ Last

— Select the method of setting the sweeping path.

Division 8

— Set the number of divisions along the sweeping path.
This setting is applied only when the path is not divided.

Weight 1 : 2.5

— Set the division weight.
This setting is applied only when the path is not divided.

☑ Keep angle to path

— Check this box to create the meshed layers with a consistent angle to the sweeping path.

Set Curve

— The button is enabled when one or more curves are selected and compatible for mesh generation.

Set Curve

— By clicking this button, the selected curves are set for mesh generation, and the text of the button is changed for setting the sweeping path.

Set Path

— The button is enabled when the sweeping path is available.

Set Path

— Click this button to set the sweeping path and thus to generate the mesh.

2) Set the element type.

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

3) Select the type of arrangement for triangular elements.

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

4) Select the option for path setting.

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

5) Set the number of divisions for translation.

Specify how many rows of elements are to be generated by translation. This setting is applied only when the sweeping path is not divided. If divided curves are selected as the sweeping path, their divisions will be applied regardless of this setting.

6) Check or uncheck "Keep angle to path" check box.

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

7) Set the weight of division density.

Enter the weight of division density in the form of $w_1:w_n$, which is the ratio between width of elements at the starting part and at the ending part of translation. This setting is also applied only when the sweeping path is not divided.

8) Select curves forming seed curves for translation.

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

9) Click $\boxed{\text{Set Curve}}$ button.

The selected curves are reserved as the seed curves for mesh generation, and the button changes into $\boxed{\text{Set Path}}$ indicating that setting the sweeping path is expected in the next step.

10) Set the sweeping path.

Set the sweeping path by the method selected at step 4).

- "Input " : If the method is set as "Input", one of the curve input tool is activated, and the cursor changes into $+$ shape. And, it is now ready for creating a sweeping path by inputting a new curve. At this step, line tool button $\boxed{\diagup}$ is activated so that straight lines may be entered. Any type of curves may be used as the sweeping path. In order to input desired types of curves, click the corresponding curve tool button.

- "Select" : If the method is set as "Select", the curve selection tool is activated, and thus cursor changes into $\blacktriangle$ shape. Select a curve which will be used as the sweeping path. Either divided or undivided curves are acceptable.

- "Last" : The sweeping path for last mesh generation is applied again. So, it is not necessary to input or select the sweeping path. This option can be used only when this method of mesh generation was applied at least once since the dialog box appeared.
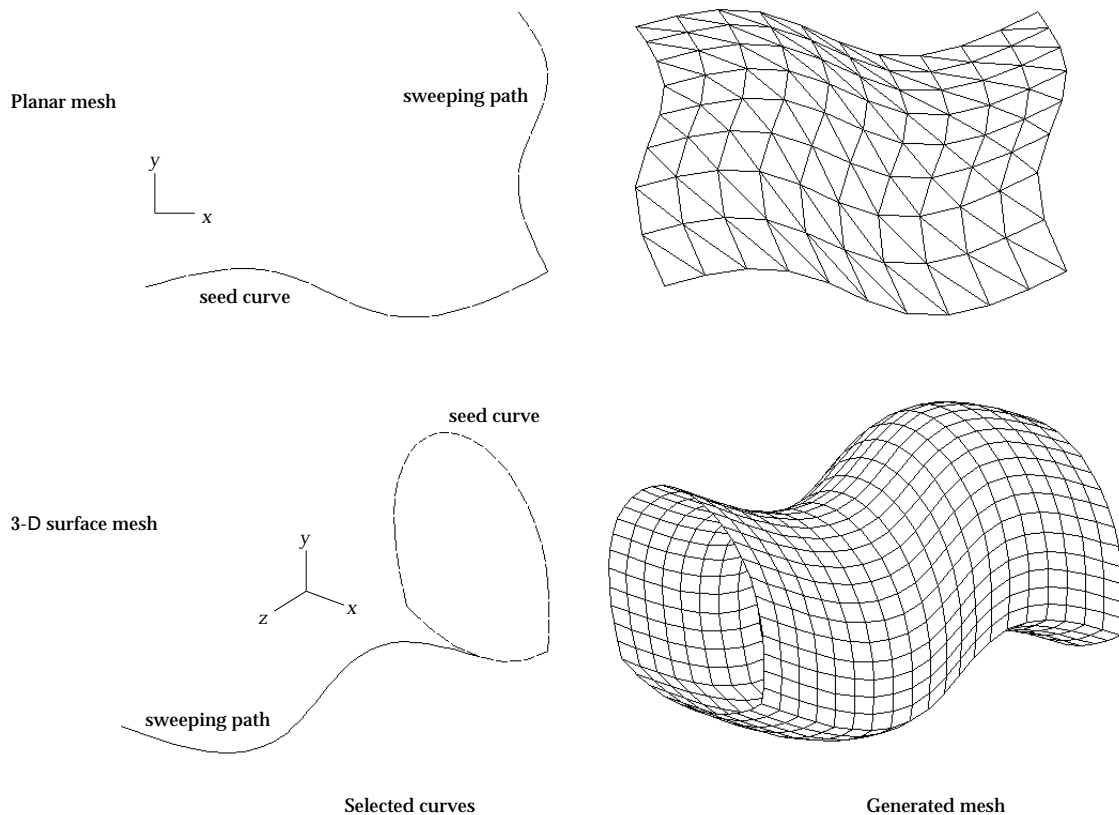
The sweeping path should be open, and its one end point should meet with a node on the seed curve. The sweeping path may be one curve or serially connected curves. If

the sweeping path consists of more than one curve, they must be either all divided or all undivided. Mixed use of divided and undivided curves for sweeping path is not allowed.
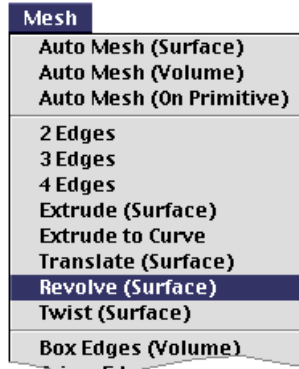
11) Click [**Set Path**] button.

A mesh is generated by translating the seed curves along the sweeping path. [**Set Path**] button is restored to [Set Curve] button. It is now ready for generating another surface mesh. The curve selection tool [🔖] is automatically activated, if it is not in action.

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

Planar mesh   sweeping path

y

x

seed curve

3-D surface mesh

seed curve

y

z   x

sweeping path

Selected curves   Generated mesh

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

■ Generating mesh by revolution

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)

2 Edges
3 Edges
4 Edges
Extrude (Surface)
Extrude to Curve
Translate (Surface)
Revolve (Surface)
Twist (Surface)

Box Edges (Volume)

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

1) Choose "Revolve (Surface)" from **Mesh** menu.
The curve selection tool 🔝 is automatically activated, and "Revolve Surf " dialog box appears.

Revolve Surf

Element Type ——————— Set the type of the elements to be generated.
● Tria 3   ○ Tria 6
○ Qaud 4   ○ Quad 8

Triangle Arrangement ——— Set arrangement of triangular elements.
● Optimal ○ Union Jack
○ Left   ○ Right

Axis Setting ——————— Select the method of setting the axis of revolution.
● Input   ○ Select
○ Last

Angle   120 ° ——————— Set the angle of revolution.
Division 20 ——————— Set the number of divisions along the circumferences created by revolution.
Weight   2 : 1 ——————— Set the division weight.

The button is enabled when —— [Reverse] [Set Curve] ——— The button is enabled when one or more curves are
the mesh is generated.   selected and compatible for mesh generation.

Click this button to reverse —— [**Reverse**] [**Set Curve**] ——— By clicking this button, the selected curves are set
the direction of revolution.   for mesh generation, and the text of the button is
                                                                changed for setting the axis of revolution.

[Set Axis] ——— The button is enabled when the axis of revolution
                is ready for setting.

[**Set Axis**] ——— Click this button to set the axis of revolution
                  and thus to generate the mesh.

2) Set the element type.
Choose one of the 4 element types given as radio buttons in the dialog.
3) Select the type of arrangement for triangular elements.
The triangular elements can be generated in the 4 different types of arrangement as explained in the previous section "Generating mesh using 2 edges".
4) Select the option for setting the axis of revolution.
Choose one of the 3 options setting the axis of revolution, "Input", "Select" and "Last" by clicking the corresponding radio button. The "Last" button is

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

5) Set the angle of revolution.

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

6) Set the number of divisions for revolution.

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

7) Set the weight of division density.

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

8) Select curves forming seed curves for revolution.

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

9) Click [Set Curve] button.

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

10) Set the axis of revolution.

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

- "Input " : If the method is set as "Input", the line tool button [⬚] is automatically activated, and the cursor changes into + shape. Input the axis of revolution following the same procedure as that of creating a straight line.

- "Select" : If the method is set as "Select", the curve selection tool is activated, and thus cursor changes into ↖ shape. Select a straight line which will be used as the axis of revolution.

- "Last" : The axis of revolution for last mesh generation is applied again. So, it is not necessary to input or select the axis. This option can be used only when this method of mesh generation was applied at least once since the program started.

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

11) Click $\boxed{\text{Set Axis}}$ button.

A mesh is generated by revolving the seed curves about the axis of revolution. $\boxed{\text{Set Axis}}$ button is restored to $\boxed{\text{Set Curve}}$ button. The program is now ready for generating another surface mesh. The curve selection tool $\boxed{\text{\char}}$ is automatically activated, if it is not in action.

$\boxed{\text{Reverse}}$ button is enabled, only when mesh generation is successful. Clicking $\boxed{\text{Reverse}}$ button reverses the sign of the revolution angle and regenerates the mesh.

You may repeat the above procedure of mesh generation without issuing the command again, while "Revolve Surf" dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.
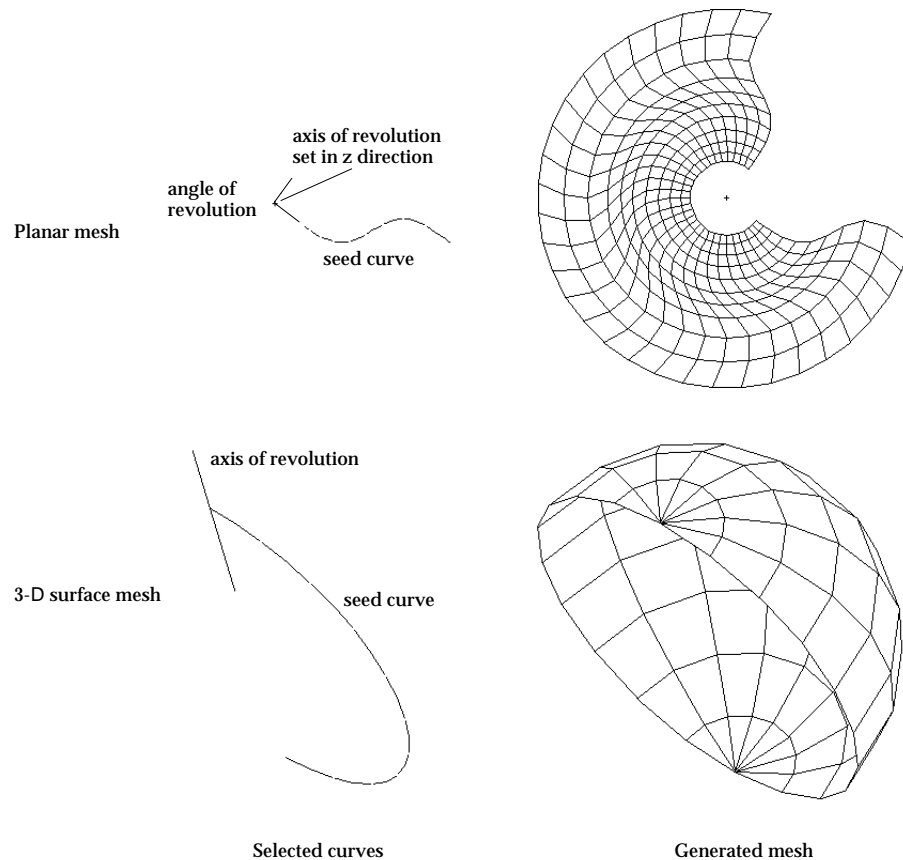
< Example of mesh generation by revolution in a plane and in 3-D space>

■ **Generating mesh by twisting**
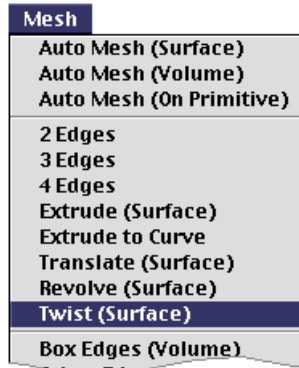
Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)

2 Edges
3 Edges
4 Edges
Extrude (Surface)
Extrude to Curve
Translate (Surface)
Revolve (Surface)
Twist (Surface)

Box Edges (Volume)

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

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

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

1) Choose "Twist (Surface)" from **Mesh** menu.

The curve selection tool [arrow icon] is automatically activated, and "Twist Surf " dialog box appears.

Twist Surf

Element Type ————— Set the type of the elements to be generated.

● Tri 3   ○ Tri 6
○ Quad 4 ○ Quad 8

Triangle Arrangement ————— Set arrangement of triangular elements.

● Optimal ○ Union Jack
○ Left   ○ Right

Axis Setting ————— Select the method of setting the axis of twisting.

○ Input   ● Select
○ Last

Height   10 ————— Set the extrusion height
Angle   360 ° ————— Set the twist angle
Division 20 ————— Set the number of divisions along the extrusion height.
Weight   1 : 3 ————— Set the division weight.

The button is enabled when the mesh is generated. ——— [Reverse] [Set Curve] ——— The button is enabled when one or more curves are selected and compatible for mesh generation.

Click this button to reverse the sign of twist angle, the mesh is regenerated. ——— [Reverse] [Set Curve] ——— By clicking this button, the selected curves are set for mesh generation, and the text of the button is altered for setting the axis of twisting.

[Set Axis] ——— The button is enabled when the axis of twisting is ready for setting.

[Set Axis] ——— Click this button to set the axis of twisting and thus to generate the mesh.

2) Set the element type.

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

3) Select the type of arrangement for triangular elements.

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

4) Select the option for setting the twist axis.

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

5) Set the extrusion height.

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

6) Set the twist angle.

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

7) Set the number of divisions.

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

8) Set the weight of division density.

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

9) Select curves forming seed curves for twisting.

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

10) Click $\boxed{\text{Set Curve}}$ button.

The selected curves are reserved as the seed curves for mesh generation, and the button changes into $\boxed{\text{Set Axis}}$ indicating that setting the twist axis is expected in the next step.

11) Set the twist axis.

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

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

program started.

12) Click [**Set Axis**] button.

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

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

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

twist axis

seed curve 3

seed curve 2

seed curve 1

seed curve 4

seed curve 5

twisted extrusion of curve 1

twisted extrusion of curve 2

twisted extrusion of curve 3

twisted extrusion of curve 4

twisted extrusion of curve 5
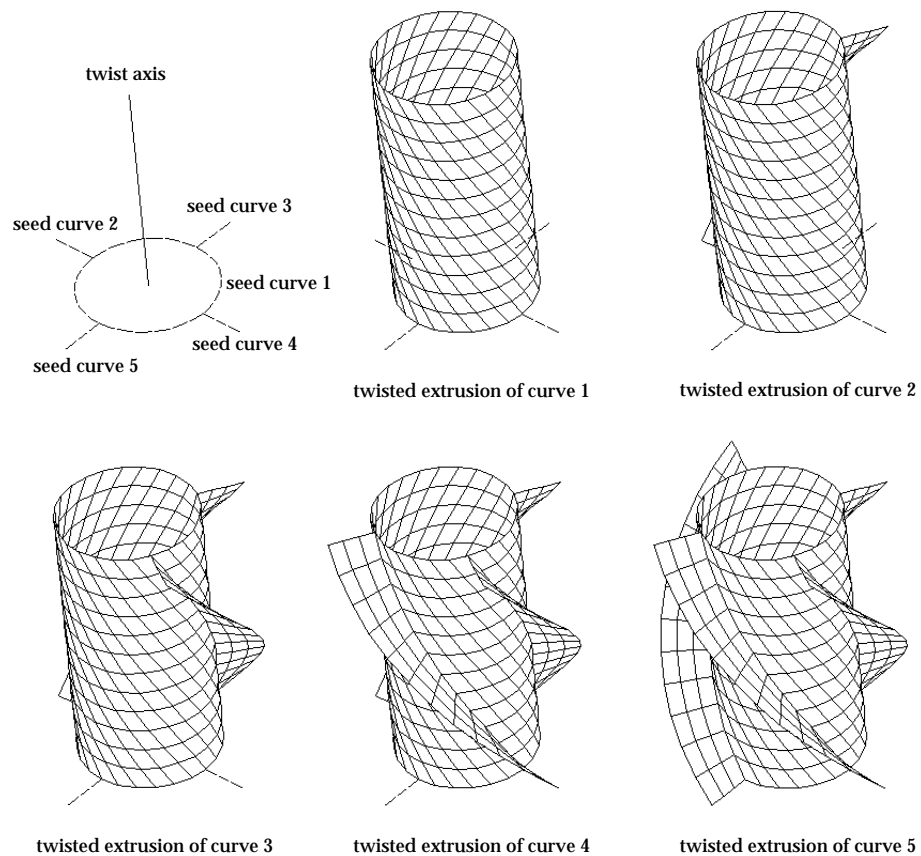
< Example of modeling 3-D surfaces by twisting>

Duplicating surface meshes

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

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

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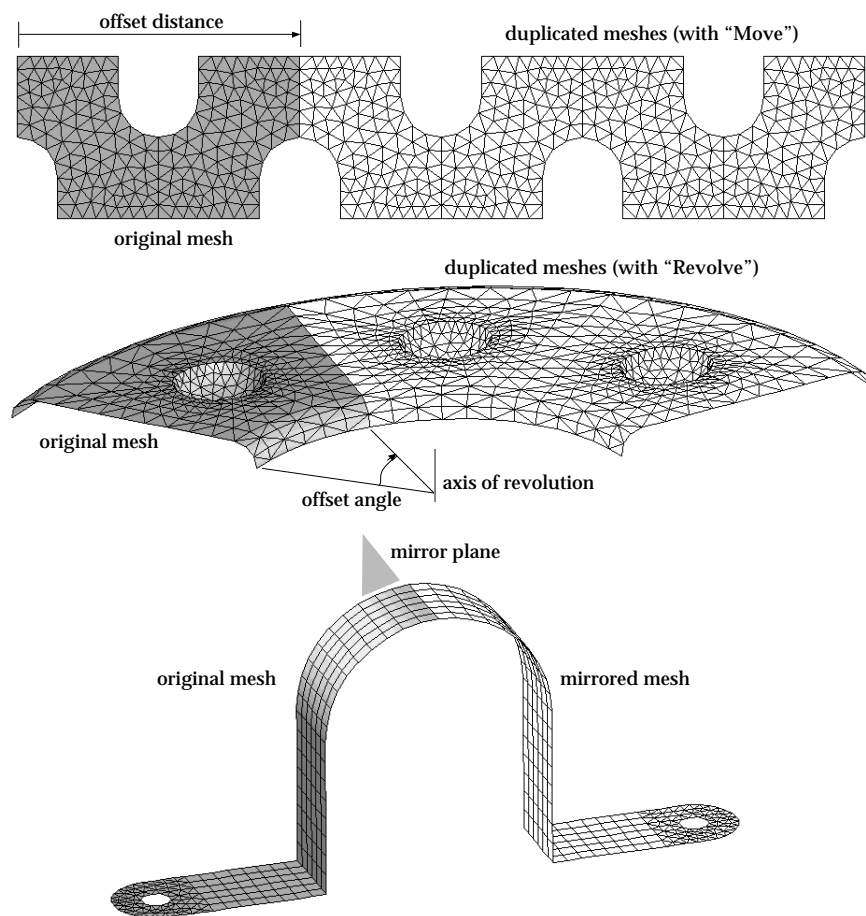
< Examples of duplicated surface meshes >

Projecting surface meshes

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

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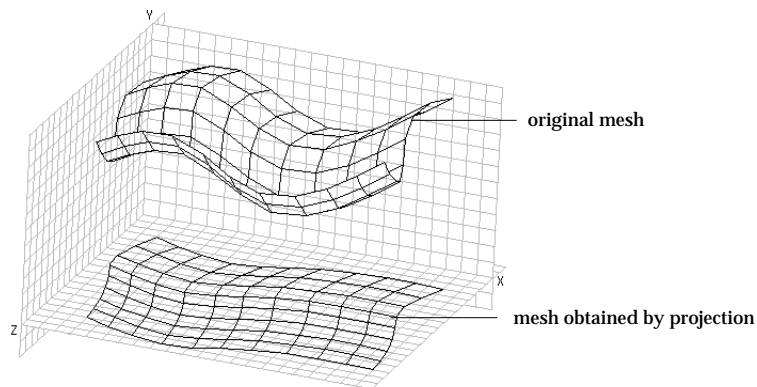
< Example of surface mesh projection >

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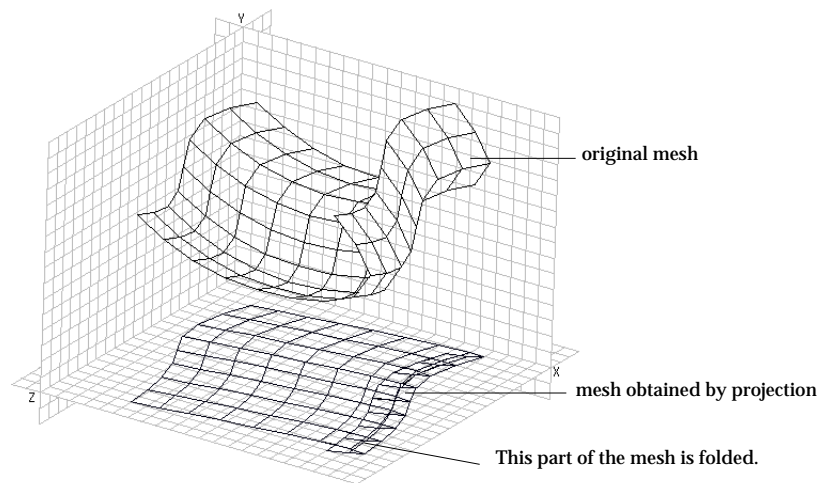
< Example of a faulty projection >

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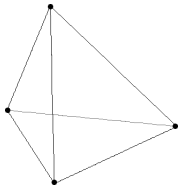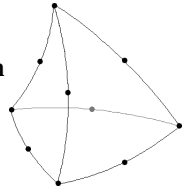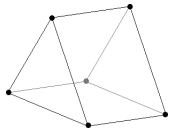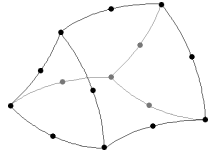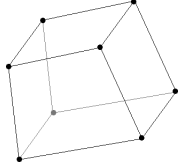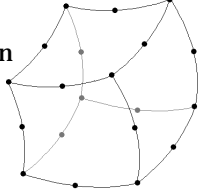
For 3-dimensional solid structural or heat conduction models, the analysis regions are modeled by volume meshes. The solid elements may be tetrahedra, prisms or hexahedra. VisualFEA provides a number of methods for generating volume meshes. They can be classified into the following 3 categories:

- automatic tetrahedronization
- mapping
- sweeping
- duplication

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

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

< Element types in volume meshes >

4 node tetrahedron	10 node tetrahedron
6 node prism	15 node prism
8 node hexahedron	20 node hexahedron

Volume mesh generation by automatic tetrahedronization

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

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

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

■ Generating mesh by automatic tetrahedronization

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

1) Choose "Auto Mesh (Volume)" from **Mesh** menu.
The surface selection tool 🔲 is automatically activated, and "Auto Mesh Volume" dialog box appears.

Check this box to detect interference of volume elements.

Check this box to examine aspect ratios of generated elements.

Check this box to visualize the mesh generation process.

Set the level of smoothing.

This button is enabled when boundary meshes are selected.

Click this button to generate mesh.
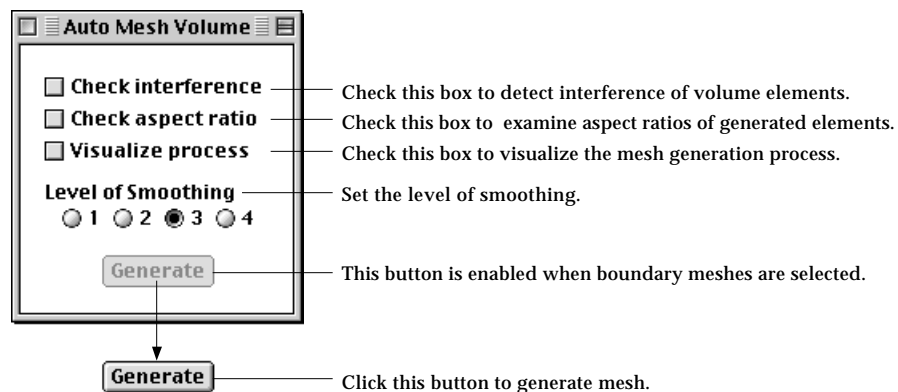
2) Set options for mesh generation process.
   - "Check interference" : If this option is checked, interference between elements is detected during mesh generation.
   - "Check aspect ratio" : The aspect ratios of generated elements are examined, and reported if they are not acceptable.
   - "Visualize process" : The recursive process of mesh generation is graphically visualized.
3) Set the level of smoothing.
   The shape of the individual element is polished through smoothing process. The level of smoothing is defined in four grades, 1, 2, 3 and 4.   Grade 4 takes longest time, but produces the best shaped elements.
4) Select surface meshes forming closed boundary.
   The selected meshes should form a closed surface boundaries. All the elements in the meshes should be triangular and have the same number of nodes. That is, they should consists of 3-node triangular elements only, or 6-node triangular elements only.   [Generate] button is enabled when acceptable boundary meshes are formed.
5) Click [Generate] button.
   A volume mesh is generated by automatic tetrahedronization, if the selected surface meshes are acceptable for automatic tetrahedronization as described in the next section. Otherwise, the action is ignored with a message notifying that automatic tetrahedronization cannot not be completed.

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

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

> **Processing mesh generation**
>
> [████████░░░░░░░░░░]
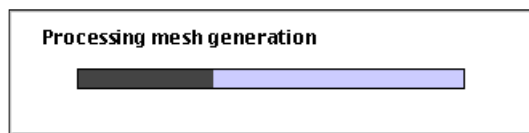
The progress bar is not to the actual time scale. The display indicates only the progress in the recursion stages. If "Visualize process" box is checked, the recursive process of mesh generation is also graphically visualized.

Boundary surface meshes   Volume mesh generated by automatic tetrahedronization
(exploded view)
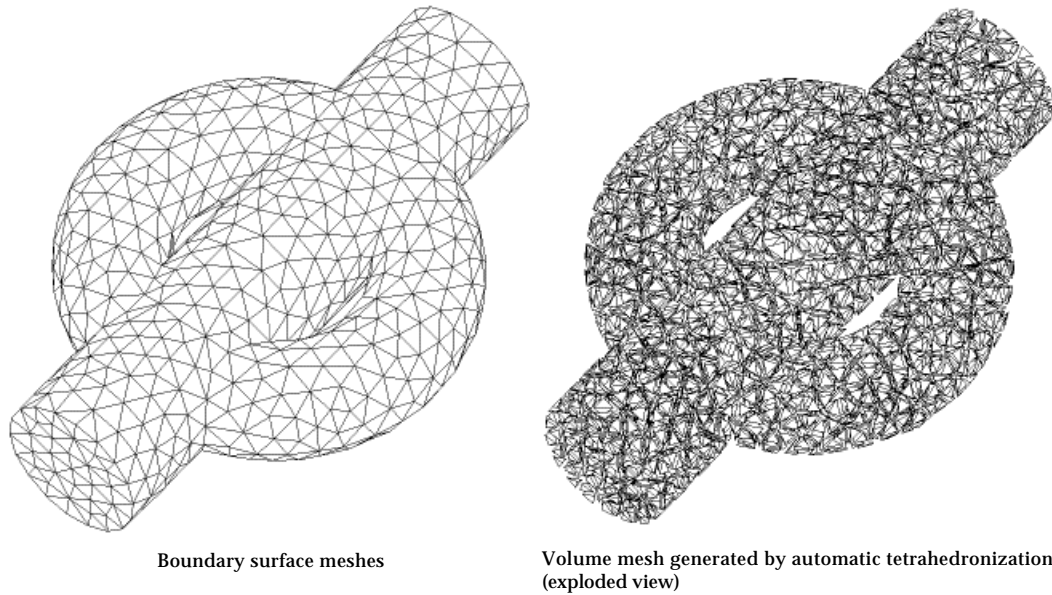
< Example of volume mesh generation by automatic tetrahedronization >

■ Forming boundary meshes for automatic tetrahedronization

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

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

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

Volume mesh generation by mapping

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

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

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

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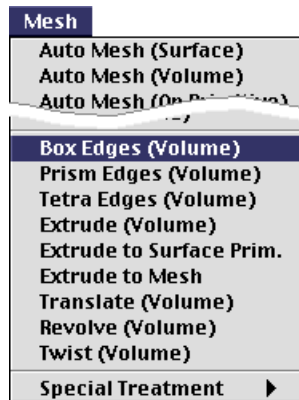
A volume mesh can be generated using transfinite mapping technique in a box-shaped region bounded by 12 edges, each of which consists of a curve or serially connected curves. The shape of the elements within the mesh may be chosen from tetrahedron and hexahedron. Prism elements are not included because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

1) Choose "Box Edges" from ▭ Mesh ▭ menu.

The curve selection tool ▭ is automatically activated, and "Box Edges" dialog box appears.
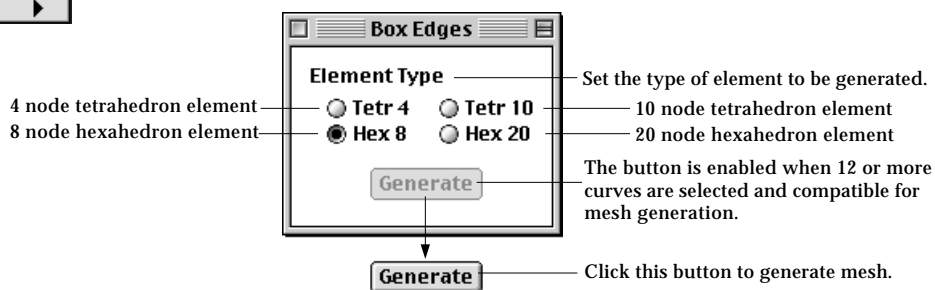
2) Set the element type.

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

3) Select curves forming box edges.

All the selected curves should be divided, and form compatible box edges. ▭ Generate ▭ button is enabled when more than equal to 12 separate curves are

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

4) Click Generate button.

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

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

Selected curves   Generated mesh
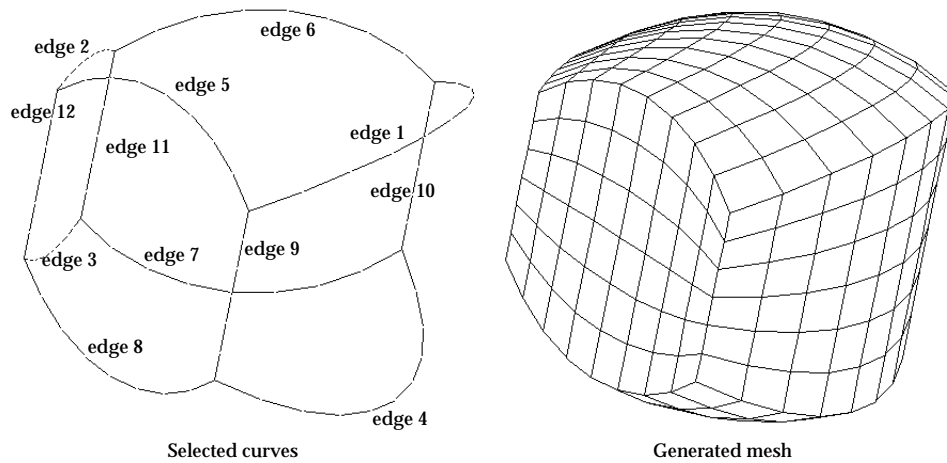
< Example of mesh generation in a box edge region >

■ Setting box edges compatible for mesh generation

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

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

**and edges 9, 10, 11 and 12 are parallel edges respectively.**
- **For element types "Tetra 4" and "Hexa 20", each edge should be divided in multiples of 2. And for "Tetra 10", each edge should be divided in multiples of 4.**

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

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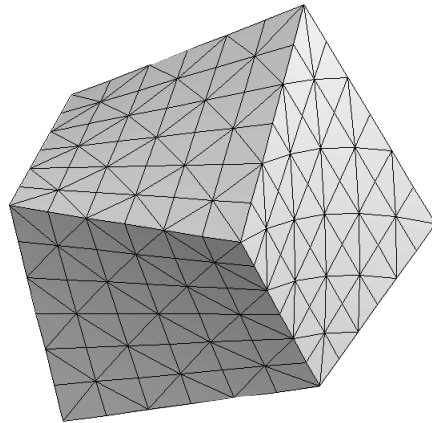
■ Generating mesh using prism edges
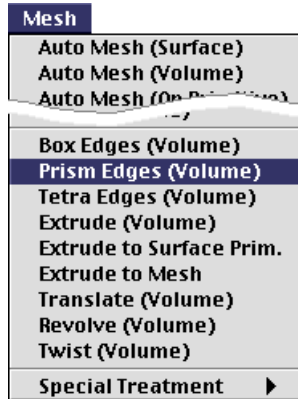
A volume mesh can be generated in a prism-shaped region bounded by 9 edges, each of which consists of a curve or serially connected curves. The shape of the elements within the mesh may be chosen from tetrahedron and prism. For mesh generation, transfinite mapping is applied in the longitudinal direction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

1) Choose "Prism Edges" from **Mesh** menu.

The curve selection tool 🔏 is automatically activated, and "Prism Edges" dialog box appears.
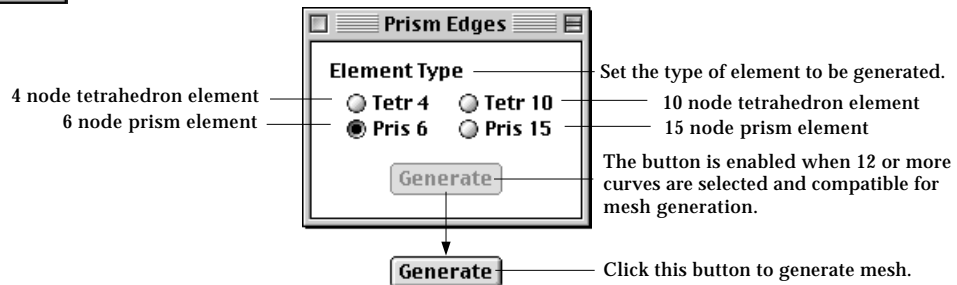
Element Type — Set the type of element to be generated.

4 node tetrahedron element — ○ Tetr 4   ○ Tetr 10 — 10 node tetrahedron element
6 node prism element — ● Pris 6   ○ Pris 15 — 15 node prism element

The button is enabled when 12 or more curves are selected and compatible for mesh generation.

Generate

Generate — Click this button to generate mesh.

2) Set the element type.

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

3) Select curves forming prism edges.

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

4) Click **Generate** button.

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

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

Selected curves   Generated mesh

**< Example of mesh generation in a prism edge region >**
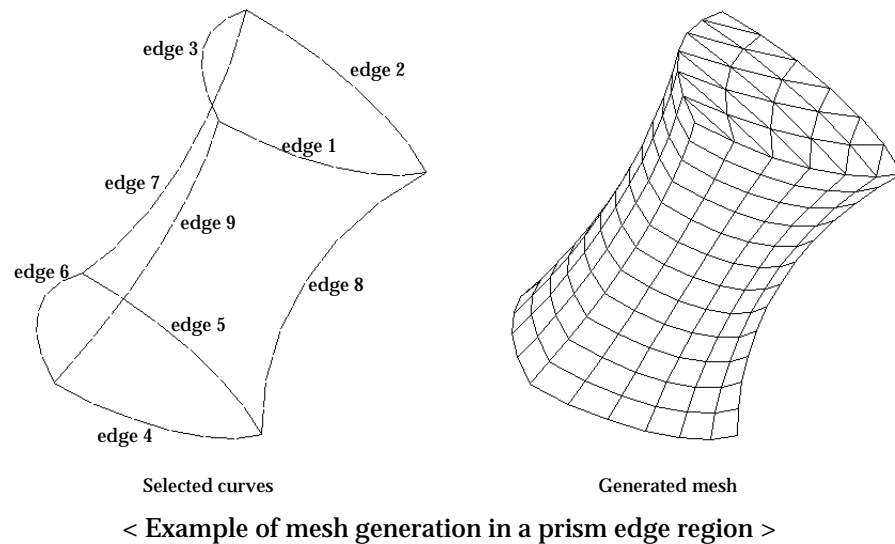
■ Setting prism edges compatible for mesh generation

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

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

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

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

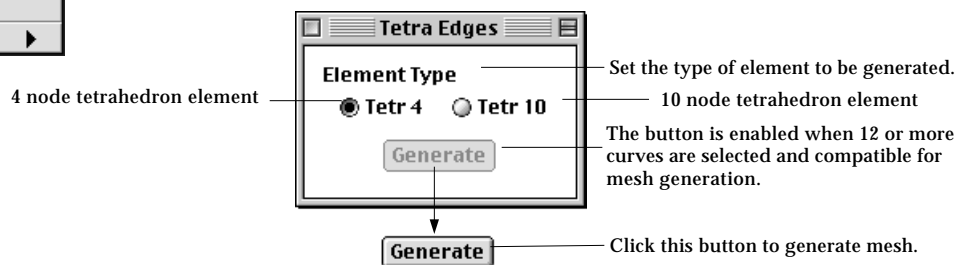<Formation of Union Jack on surfaces of a prism edge mesh>
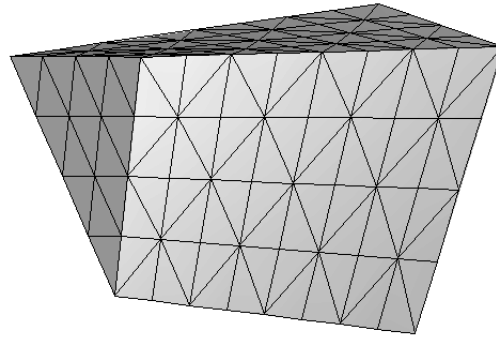
■ Generating mesh using tetrahedron edges
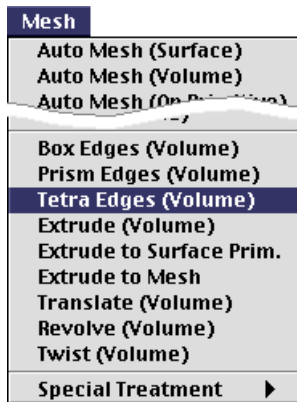
A volume mesh can be generated in a tetrahedron-shaped region bounded by 6 edges, each of which consists of a curve or serially connected curves. The shape of the elements generated by this method is always tetrahedron. For mesh generation, triangular mapping is applied on every triangular section of the tetrahedron.

1) Choose "Tetra Edges" from **Mesh** menu.
   The curve selection tool [▶] is automatically activated, and "Tetra Edges" dialog box appears.

Set the type of element to be generated.

10 node tetrahedron element

The button is enabled when 12 or more curves are selected and compatible for mesh generation.

4 node tetrahedron element

Click this button to generate mesh.

2) Set the element type.
   Choose one of the 2 element types given as radio buttons in the dialog. "Tetr 4" and "Tetr 10" represent element shape that are 4 and 10 node tetrahedron, respectively.

3) Select curves forming tetra edges.
   All the selected curves should be divided, and form compatible tetra edges. [Generate] button is enabled when more than equal to 6 separate curves are selected and a tetra edges are form by them.   The number of divisions on all

the edges should be equal.

4) Click $\boxed{\text{Generate}}$ button.

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

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

Selected curves   Generated mesh
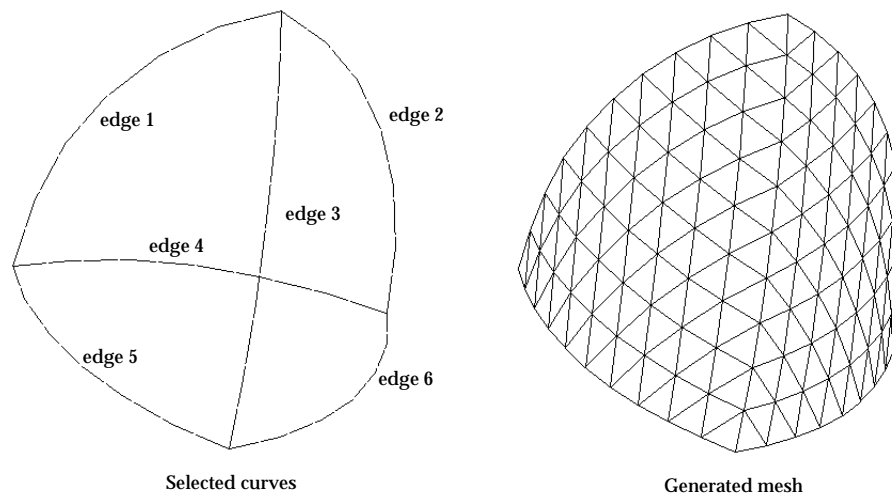
< Example of mesh generation in a tetra edge region >

■ Setting tetra edges compatible for mesh generation

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

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

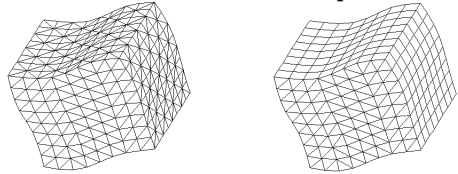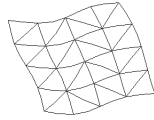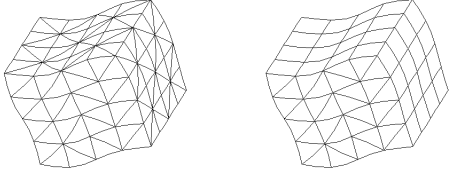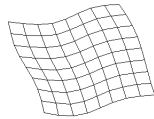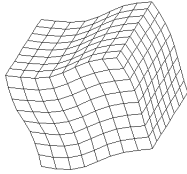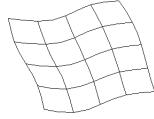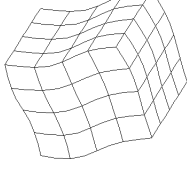## Volume mesh generation by sweeping operations
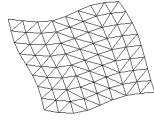
VisualFEA supports the following 4 types of sweeping operations for volume mesh generation:

- Extrusion
- Translation
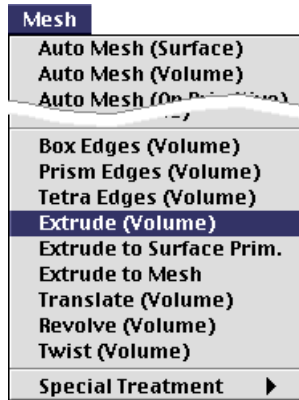- Revolution
- Twisting

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

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

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

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

■ Generating mesh by extrusion

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On ~~~~~~)
~~~~~~
Box Edges (Volume)
Prism Edges (Volume)
Tetra Edges (Volume)
Extrude (Volume)
Extrude to Surface Prim.
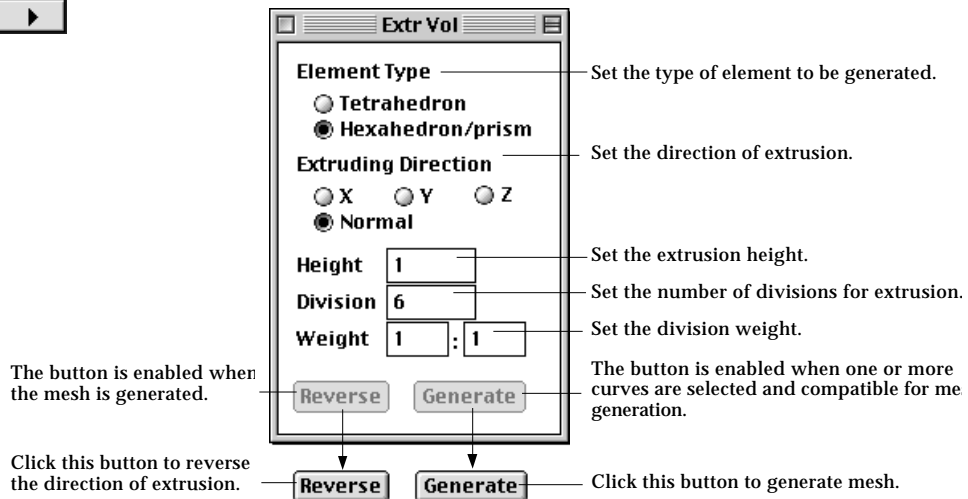Extrude to Mesh
Translate (Volume)
Revolve (Volume)
Twist (Volume)

Special Treatment ▶

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

1) Choose "Extrude(Volume)" from **Mesh** menu.
The curve selection tool 🔲 is automatically activated, and "Extr Vol" dialog box appears.

Extr Vol

Element Type ———————————— Set the type of element to be generated.
○ Tetrahedron
◉ Hexahedron/prism
Extruding Direction ———————————— Set the direction of extrusion.
○ X   ○ Y   ○ Z
◉ Normal
Height `1` ———————————— Set the extrusion height.
Division `6` ———————————— Set the number of divisions for extrusion.
Weight `1` : `1` ———————————— Set the division weight.

The button is enabled when the mesh is generated. ———— [Reverse]   [Generate] ———— The button is enabled when one or more curves are selected and compatible for mesh generation.

Click this button to reverse the direction of extrusion. ———— [Reverse]   [Generate] ———— Click this button to generate mesh.

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

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

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

5) Enter the number of divisions for extrusion.

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

6) Enter the weight of division density.

Enter the weight of division density in the form of $w_1{:}w_n$, which is the ratio between thickness of elements at the starting part, l_1 and at the ending part of extrusion, l_n.
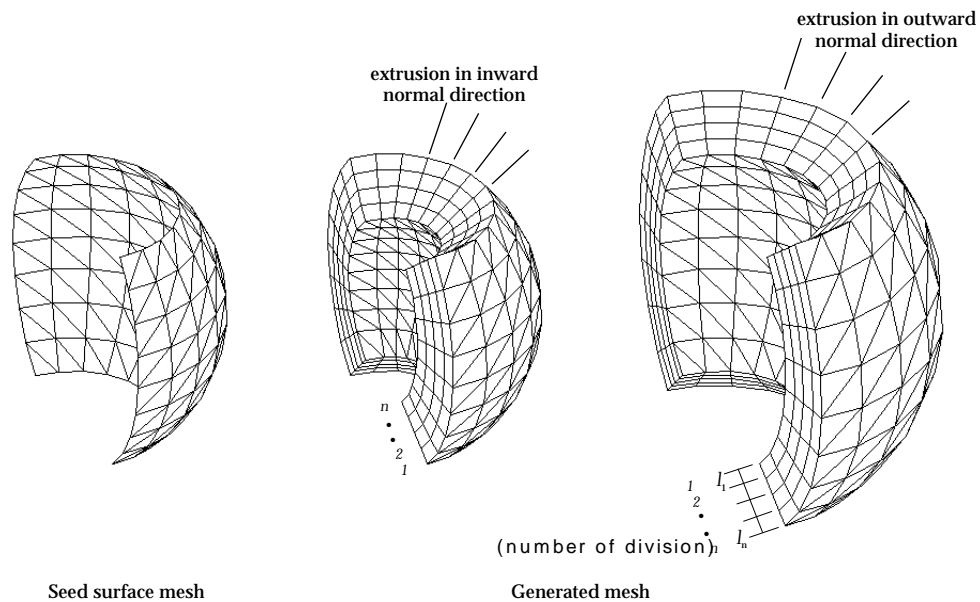
7) Select the seed surfaces for extrusion.

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

8) Click $\boxed{\text{Generate}}$ button.

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

You may repeat the above procedure of generating mesh by extrusion without issuing the command again, while "Extr Surf" dialog remains on the screen. This mesh generation command is terminated by closing the dialog box or issuing any other command.
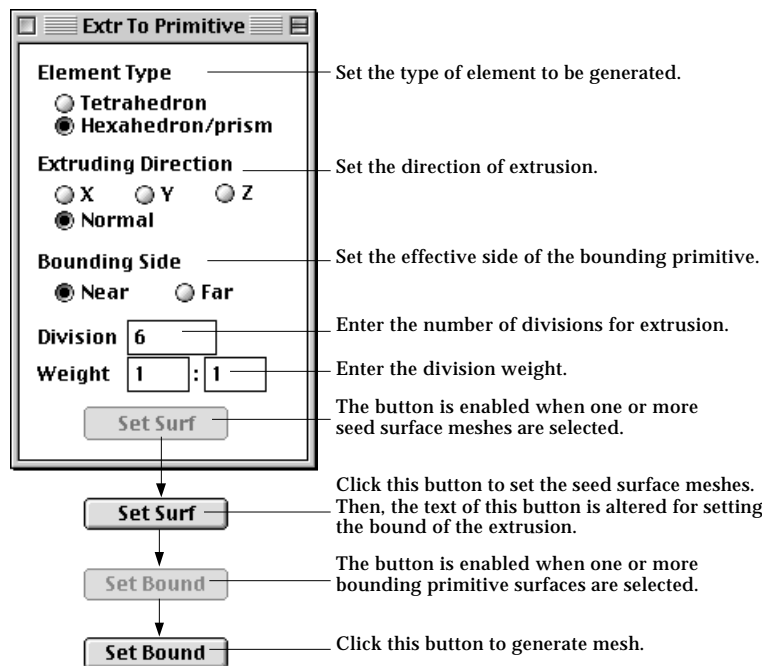
< A volume mesh generated by extrusion in normal direction >

■ Generating mesh by extrusion up to bounding surface primitives

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Pri......

Box Edges (Volume)
Prism Edges (Volume)
Tetra Edges (Volume)
Extrude (Volume)
Extrude to Surface Prim.
Extrude to Mesh
Translate (Volume)
Revolve (Volume)
Twist (Volume)

Special Treatment   ▶

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

1) Choose "Extrude to Surface Prim." from **Mesh** menu.

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

Extr To Primitive

Element Type ——————— Set the type of element to be generated.
  ○ Tetrahedron
  ◉ Hexahedron/prism

Extruding Direction ———— Set the direction of extrusion.
  ○ X   ○ Y   ○ Z
  ◉ Normal

Bounding Side ——————— Set the effective side of the bounding primitive.
  ◉ Near   ○ Far

Division [6] ——————— Enter the number of divisions for extrusion.

Weight [1] : [1] ——————— Enter the division weight.

[Set Surf] ——————— The button is enabled when one or more seed surface meshes are selected.

↓

[Set Surf] ——————— Click this button to set the seed surface meshes. Then, the text of this button is altered for setting the bound of the extrusion.

↓

[Set Bound] ——————— The button is enabled when one or more bounding primitive surfaces are selected.

↓

[Set Bound] ——————— Click this button to generate mesh.

2) Set the element type.

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

3   Set the direction of extrusion.

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

4) Set the side of the bound.

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

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

5) Set the number of divisions for extrusion.

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

6) Set the weight of division density.

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

7) Select surface meshes forming seed surfaces for extrusion.

[Set Surf] button is enabled when one or more surface meshes are selected.

8) Click [**Set Surf**] button.

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

9) Select the bounding surface primitives.

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

10) Click [**Set Bound**] button.

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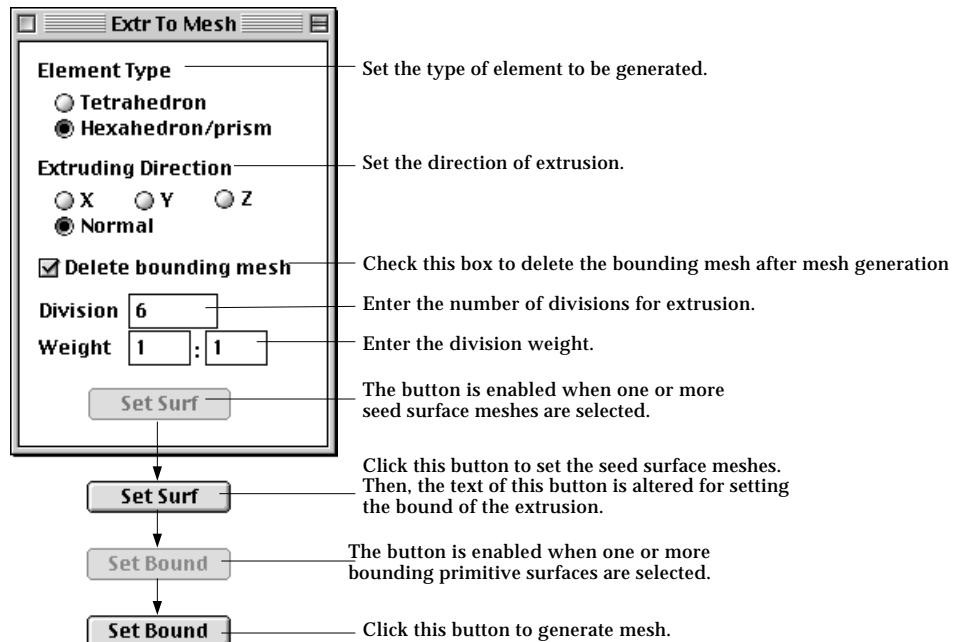

seed surface mesh   bounding surface primitive

near side

far side

Seed surface and bounding primitive   Mesh generated by extrusion upto "near" side   Mesh generated by extrusion upto "far" side

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

■ Generating mesh by extrusion up to surface meshes

Mesh
Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)

Box Edges (Volume)
Prism Edges (Volume)
Tetra Edges (Volume)
Extrude (Volume)
Extrude to Surface Prim.
Extrude to Mesh
Translate (Volume)
Revolve (Volume)
Twist (Volume)

Special Treatment   ▶

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

1) Choose "Extrude to Mesh" from **Mesh** menu.
The surface selection tool 🔶 is automatically activated, and "Extr to Mesh" dialog box appears.

Extr To Mesh

Element Type ———————— Set the type of element to be generated.
  ○ Tetrahedron
  ● Hexahedron/prism

Extruding Direction ———— Set the direction of extrusion.
  ○ X   ○ Y   ○ Z
  ● Normal

☑ **Delete bounding mesh** ———— Check this box to delete the bounding mesh after mesh generation

Division 6 ———————————— Enter the number of divisions for extrusion.

Weight 1 : 1 ———————— Enter the division weight.

Set Surf ———————— The button is enabled when one or more seed surface meshes are selected.

Set Surf ———————— Click this button to set the seed surface meshes. Then, the text of this button is altered for setting the bound of the extrusion.

Set Bound ———————— The button is enabled when one or more bounding primitive surfaces are selected.

Set Bound ———————— Click this button to generate mesh.

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

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

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

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

5) Set the number of divisions for extrusion.

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

6) Set the weight of division density.

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

7) Select surface meshes forming seed surfaces for extrusion.

[Set Surf] button is enabled when one or more surface meshes are selected.

8) Click [Set Surf] button.

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

9) Select the bounding surface meshes.

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

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

10) Click [Set Bound] button.

A volume mesh is generated by extruding the seed meshes up to the bounding meshes. [Set Bound] button is restored to [Set Surf] button, and the surface selection tool [🢑] is activated.

seed surface mesh   bounding surface mesh

Seed surface and bounding surface mesh   Generated volume mesh

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

■ **Generating volume mesh by translation**

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

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

1) Choose "Translate (Volume)" from **Mesh** menu.
   The surface selection tool 🔫 is automatically activated, and "Translate Vol" dialog box appears.

Element Type — Set the type of element to be generated.
○ Tetrahedron
◉ Hexahedron/prism

Path Setting — Set the direction of extrusion.
◉ Input   ○ Select
○ Last

Division [6] — Enter the number of divisions for extrusion.

Weight [1] : [1] — Enter the division weight.

☑ Keep angle to path — Check this box to create the meshed layers with a consistent angle to the sweeping path.

[Set Surf] — The button is enabled when one or more seed surface meshes are selected.

[**Set Surf**] — Click this button to set the seed surface meshes. Then, the text of this button is altered for setting the sweeping path.

[Set Path] — The button is enabled when the sweeping path is entered or selected.

[**Set Path**] — Click this button to generate mesh.

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

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

4) Set the number of divisions for translation.
   Specify how many rows of elements to be generated by translation. This

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

5) Set the weight of division density.

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

6) Check or uncheck "Keep angle to path" check box.

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

7) Select surface meshes for seed meshes.

[Set Surf] button is enabled when one or more surface meshes are selected.

8) Click [Set Surf] button.

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

9) Set the sweeping path.

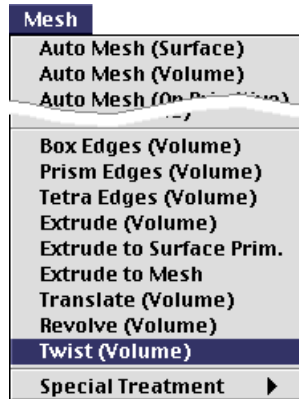Set the sweeping path by the method selected at step 3).   Tools corresponding to the method are automatically activated, and the cursor changes accordingly. The sweeping path should be open, and its one end point should meet with a node on the seed curve. The sweeping path may be one curve or serially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

10) Click [Set Path] button.

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

seed surface mesh

sweeping path

Seed mesh and sweeping path   Generated volume mesh

< Example of volume mesh generation by translation >

■ Generating volume mesh by revolution

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

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

1) Choose "Revolve (Volume)" from **Mesh** menu.
   The surface selection tool 🔾 is automatically activated, and "Revolve Vol " dialog box appears.

Mesh

Auto Mesh (Surface)
Auto Mesh (Volume)
Auto Mesh (On Primitive)

Box Edges (Volume)
Prism Edges (Volume)
Tetra Edges (Volume)
Extrude (Volume)
Extrude to Surface Prim.
Extrude to Mesh
Translate (Volume)
Revolve (Volume)
Twist (Volume)

Special Treatment ▶

Revolve Vol

Element Type ——— Set the type of element to be generated.
○ Tetrahedron
● Hexahedron/prism

Axis Setting ——— Set the direction of extrusion.
● Input   ○ Select
○ Last

Angle   150 ° ——— Enter the angle of revolution.
Division 12 ——— Enter the number of divisions for extrusion.
Weight 1 : 1 ——— Enter the division weight.

[Reverse] [Set Surf] ——— The button is enabled when one or more seed surface meshes are selected.

[Reverse] [Set Surf] ——— Click this button to set the seed surface meshes. Then, the text of this button is altered for setting the axis of revolution.

[Set Axis] ——— The button is enabled when the axis of revolution is selected or entered.

[Set Axis] ——— Click this button to generate mesh.

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

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

4) Set the angle of revolution.
   Insert the angle of revolution in the dialog box. The angle should be greater than or equal to -360° and less than equal to 360°. Both -360° and 360° make a full surface of revolution. The negative sign reverses the direction of

revolution.

5) Set the number of divisions for revolution.

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

6) Set the weight of division density.

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

7) Select surface meshes forming seed meshes for revolution.

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

8) Click **Set Surf** button.

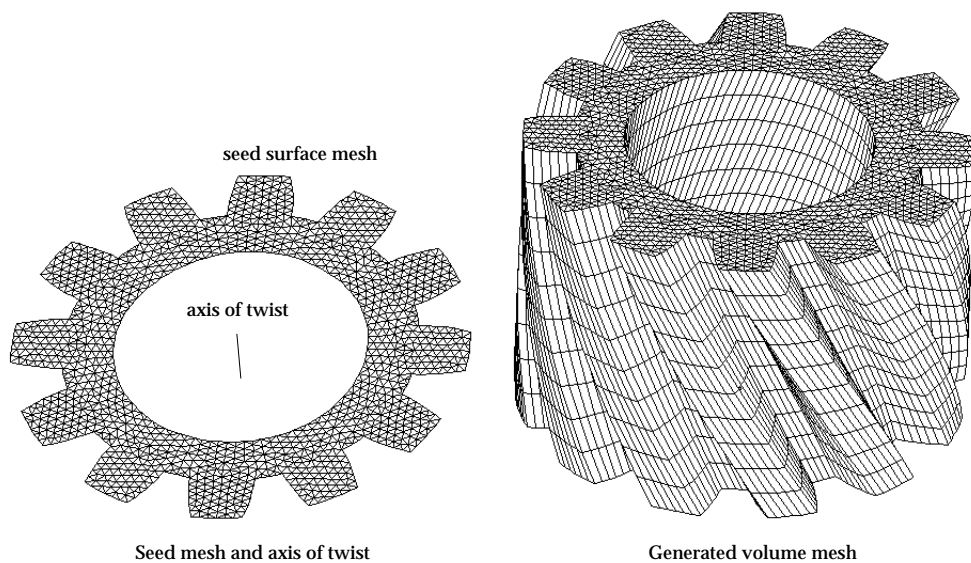   **The selected surfaces are reserved as the seed surfaces for mesh generation, and the button changes into Set Axis indicating that setting the axis of revolution is expected in the next step.**

9) Set the axis of revolution.

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

10) Click **Set Axis** button.

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

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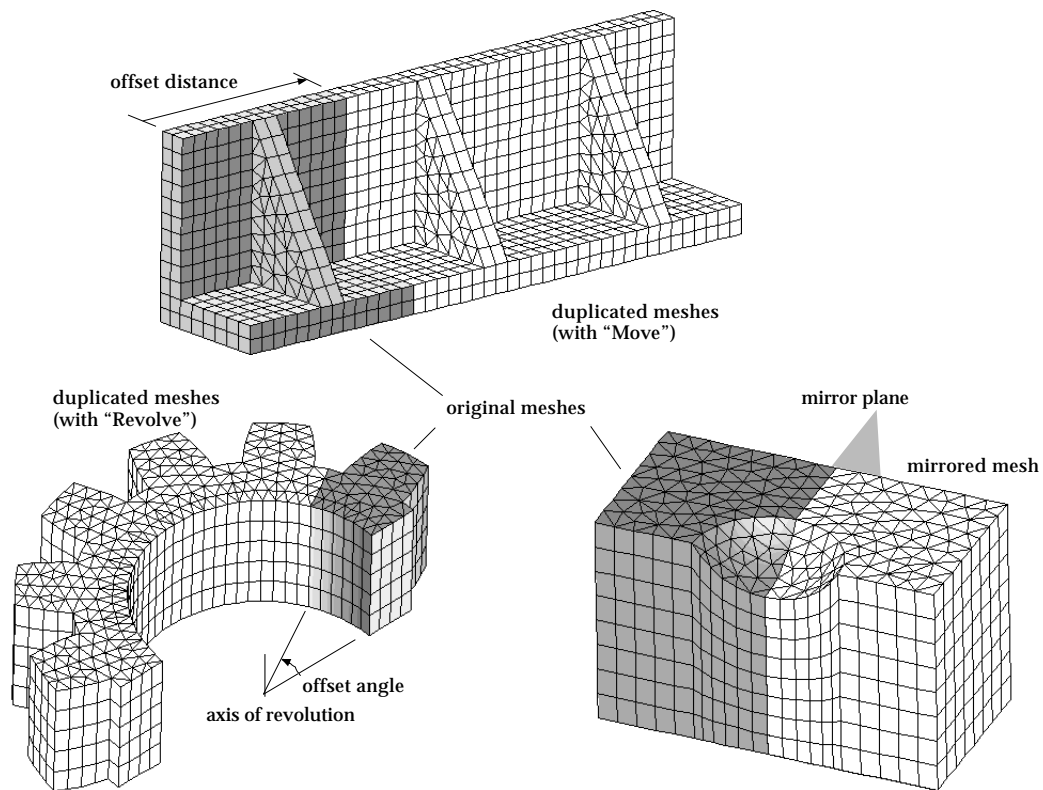
axis of revolution

seed surface mesh

Seed mesh and axis of revolution   Generated volume mesh

< Example of volume mesh generation by revolution >

■ Generating volume mesh by twisting

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

1) Choose "Twist (Volume)" from **Mesh** menu.
   The surface selection tool 🔆 is automatically activated, and "Twist Vol" dialog box appears.

Set the type of element to be generated.

Set the direction of extrusion.

Enter the height of extrusion.

Enter the twist angle.

Enter the number of divisions for twist.

Enter the division weight.

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

Click this button to set the seed surface meshes. Then, the text of this button is altered for setting the twist axis.

The button is enabled when the axis of twist is selected or entered.

Click this button to generate mesh.

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

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

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

the second.

5) Set the twist angle.

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

6) Set the number of divisions.

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

7) Set the weight of division density.

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

8) Select surface meshes forming seed surfaces for twisting.

[Set Surf] button is enabled when one or more surface meshes are selected.

9) Click [Set Surf] button.

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

10) Set the twist axis.

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

11) Click [Set Axis] button.

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

seed surface mesh

axis of twist

Seed mesh and axis of twist   Generated volume mesh

< Example of modeling 3-D solid by twisting>

Duplicating volume meshes

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

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

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

< Examples of duplicated volume meshes >

Frame Element Generation

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

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

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

< Types of frame element >

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

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

Creating frame elements using straight lines or curves

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

This curved member is not divided yet.
So, no elements are assigned on it.

This straight member is modelled
by 2 elements for convenience.

One element is automatically assigned
on these straight members.

This curved member has been
divided into 8. So, 8 elements
are assigned on it.

< Example of element generation on frame members >

■ Creating a frame element by inputting a straight line

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

■ Generating frame elements by dividing curves

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

This curved member is approximated by 4 elements.

This curved member is approximated by 8 elements.

< Bending moment diagrams with different approximation of curved member >

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

This straight member is divided into 3 and assigned with a distributed load on its middle part.

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

Creating frame elements using mesh generation functions

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

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

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

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

Seed curves are set for mesh generation

The mesh is generated by "2 edges"

Redundant members are removed.

Some members are added.

< Example of constructing frame model using mesh generation >

Duplicating frame elements

New frame elements may be created by duplicating existing ones. The position and orientation of the duplicated frame elements are determined by moving, revolving or mirroring coordinates of original meshes with specified offsets.
Duplicating frame elements is identical to duplicating curves except that not only the curves but also the elements are duplicated. However, individual elements cannot be duplicated independently. In order to start duplicating frame elements, click the curve selection tool ⬚ if it is not in action. Then, select one of the submenu items for duplications: "Move", "Revolve" or "Mirror". Further details of duplication procedure are explained in Chapter 3, and are not repeated here.
The following figure shows examples of duplication of frame elements with moving, revolving or mirroring.
You should be careful not to create overlapping members by duplication. It is advisable to avoid duplicating members on the axis of revolution or mirror plane. That is the reason why the members on the axis of revolution or mirror plane are absent in the following examples. Those members can be added manually before or after duplication.

< Examples of duplicated frame members >

Mesh Editing

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

General mesh editing commands

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

■ Undoing mesh generation

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

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

■ Deleting meshes

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

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

■ Copying meshes

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

■ Cutting meshes

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

■ Pasting meshes

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

■ Merging meshes

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

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

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

Modifying nodal coordinates

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

■ Dragging nodes using mouse

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

1) Click node selection tool ⬚ , if it is not in action.
2) Double click the node to drag.
   The node is marked with a small blue rectangle, and "Node Info" dialog box appears. The dialog has the basic information on the marked node. If "Node Info" dialog is already on the screen, you don't have to double click to invoke the dialog. Instead, click the node only once. Then, the contents of the dialog are renewed.

3) Position the screen cursor over the node and press the mouse button.
4) Move the mouse with the button pressed.
   The screen cursor moves, and the nodal point moves along with the cursor movement. While the node is moving, the updated nodal coordinates are continuously echoed as texts in the "Node Info" dialog. At the same time, the boundary lines of the elements connected to the node are also modified. Accordingly the modified shape of the elements are drawn on the screen.
5) Release the mouse button.
   The node is settled the the point where the mouse button is released.
6) Repeat step 3), 4), 5) until the node is placed on the desired position.

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

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

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

Move the mouse with mouse button pressed.

Release the mouse button.

< Process of dragging a node >

■ Modifying nodal coordinates by keyboard input

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

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

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

■ Changing node number

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

■ Absorbing nodes

Edit
Undo	⌘Z
Redo	
Cut	⌘X
Copy	⌘C
Paste	⌘V
Clear	⌘K
Duplicate and	▶
Link	⌘L
Absorb Nodes	
Separate	⌘M
Intersect	

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

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

Set the tolerance of absorption by editing this text.

Absorption Tolerance 0.001

Cancel   Absorb

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

Transforming meshes

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

■ Moving meshes

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

1) Click surface selection tool ⬚ or volume selection tool ⬚ depending on the type of mesh to move.
2) Select the meshes to move.
   Selected meshes are included in the list of the meshes to move.
3) Click object movement tool ⬚ .
   A bounding box surrounding the selected meshes is drawn on the screen. The bounding box may be a rectangle or a right hexahedron, as shown in the following example, depending on the spatial dimension formed by the meshes to move. If all the nodes within the meshes have an identical coordinate in any one of X, Y or Z, the bounding box is reduced to a rectangle.

The bounding box is a rectangle.   The bounding box is a right hexahedron

< Bounding boxes surrounding planes and 3-D volumes >

   At this stage, editable text boxes displaying the distances of movement in X, Y and Z directions appear at the bottom of the tool palette as shown at left.
4) Place the screen cursor on one edge of the bounding box, and press the mouse button.
   The direction of movement is set as parallel to the selected edge.
5) Move the mouse with the button pressed.
   The bounding box moves by sliding on the selected edge.   The distance of movement is determined by the cursor movement, and is displayed on the text boxes at the bottom of the tool palette.
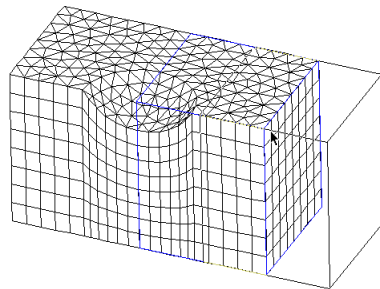
6) Release the mouse button.

The bounding box settles at the final position where the mouse button is released. The selected meshes also are moved to the new position.
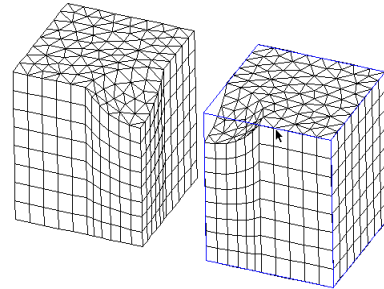
The meshes can be moved to the desired position by repeating step 4), 5) and 6) with different edges. Or, the desired movement may be achieved at once by directly entering the offset distances in the editable text boxes of the tool palette.

Place the cursor over an edge of bounding box parallel to the direction of movement. And press the mouse button.

Drag the bounding box to the desired position.

Release the mouse button. The selected meshes together with bounding box settles at the position.

< Moving meshes by dragging the bounding box >

■ Rotating meshes

You may rotate selected meshes either by turning the bounding box using the mouse or by entering the angles of rotation about X, Y and Z axis as explained in the following:

1) Click surface selection tool [⊕] or volume selection tool [⊕] depending on the type of mesh to rotate.
2) Select the meshes to rotate.

Selected meshes are included in the list of the meshes to rotate.

3) Click object rotation tool [⊚].

A bounding box surrounding the selected meshes is drawn on the screen. Editable text boxes displaying the angles of rotation about X, Y and Z axes appear at the bottom of the tool palette as shown left.

Object Angle

θ_x	0
θ_y	51.9
θ_z	0

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

The axis of rotation is set to be parallel to the selected edge, and to pass through the centroid of the bounding box.
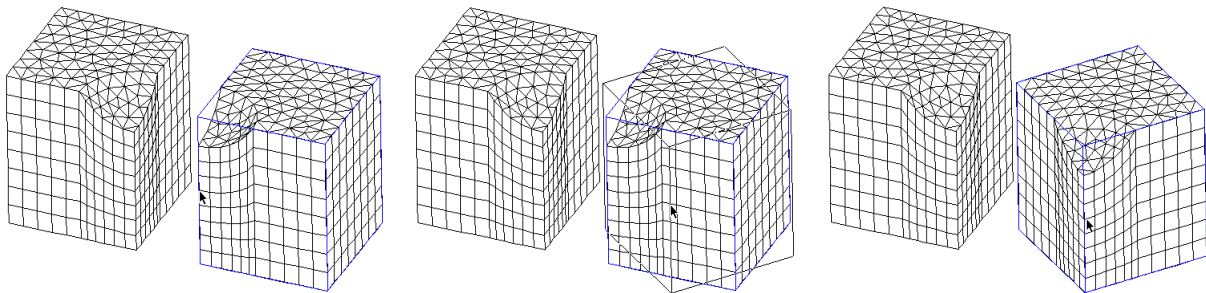
5) Move the mouse with the button pressed.

The bounding box rotates about the coordinate axis parallel to the selected edge.   The angle of the rotation is determined by the edge moving along the cursor movement. The angle of rotation is continuously updated on the text boxes at the bottom of the tool palette, while the bounding box is being rotated.

6) Release the mouse button.

The bounding box settles with the final orientation where the mouse button is released. The selected meshes are also rotated to the new orientation in accordance with the bounding box.

The meshes can be rotated to the desired orientation by repeating step 4), 5) and 6) with different edges. Or, the desired rotation may be achieved at once by directly entering the angles of rotation about each coordinate axis in the editable text boxes of the tool palette.

Place the cursor over an edge of bounding box parallel to the axis of rotation. And press the mouse button. **Drag the edge. Then, the bounding box rotates along with the edge. Rotate the box to the desiredorientation.** **Release the mouse button. The selected meshes are rotated to fit within the rotated bounding box.**

< Rotating meshes by dragging an edge of the bounding box >

■ Resizing meshes

You may resize the selected meshes either by rubber-banding the bounding box or by entering the relative scale in X, Y and Z axis as explained in the following:

1) Click surface selection tool 🖾 or volume selection tool 🖾 depending on the type of mesh to resize.

2) Select the meshes to resize

Selected meshes are included in the list of the meshes to resize.

3) Click object resizing tool 🖾 .

A bounding box surrounding the selected meshes is drawn on the screen. Editable text boxes displaying the relative scales of the meshes in X, Y and Z axes appear at the bottom of the tool palette as shown left.   The relative scales are set to 1 at the beginning.

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

The direction of the resizing is set as parallel to the selected edge.
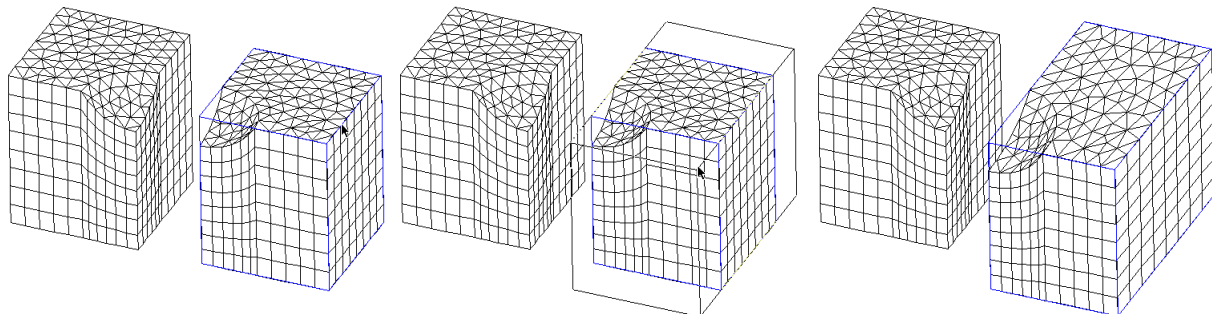
5) Move the mouse with the button pressed.

The bounding box expands or shrinks in the direction parallel to the selected edge.   The relative scale is determined by the movement of the cursor on the edge. If the cursor moves toward the center of the edge, the relative scale is reduced and the selected meshes shrink. If the cursor moves toward the ends

Object Scale

r_x	1
r_y	1
r_z	1.7

of the edge, the meshes expand. The relative scale is continuously updated on the text boxes at the bottom of the tool palette, while the bounding box is being resized.

6) Release the mouse button.

The bounding box settles with the final scale designated by the cursor position where the mouse button is released. The selected meshes are also resized to the new scale in accordance with the bounding box.

The meshes can be resized to the desired scale by repeating step 4), 5) and 6) with different edges. Or, the desired resizing may be achieved at once by directly entering the relative scale in each coordinate axis in the editable text boxes of the tool palette.
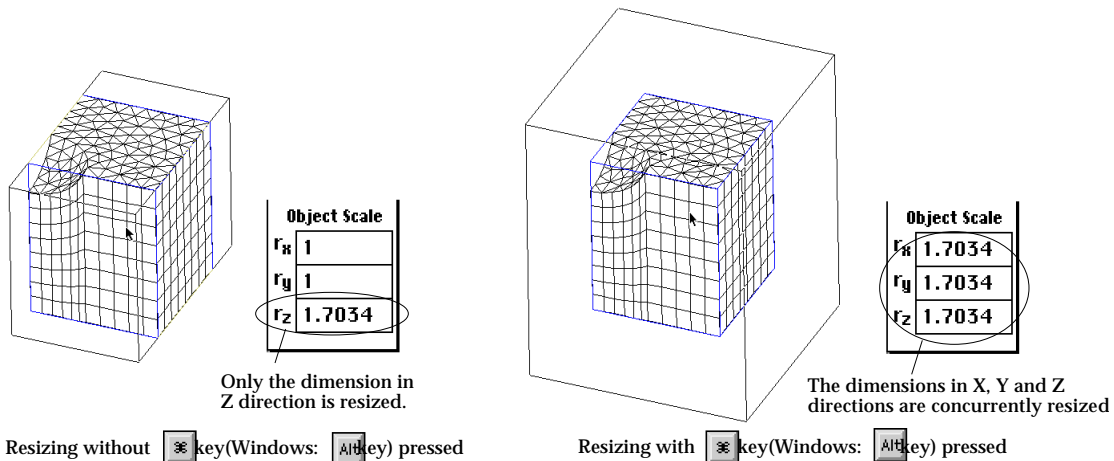

Place the cursor over an edge of bounding parallel to the direction of resizing.   And press the mouse button.

Slide the cursor over the edge. The bounding box expands or shrinks along the movement of the cursor.

Release the mouse button. The meshes shrink or expand to fit within the resized bounding box.

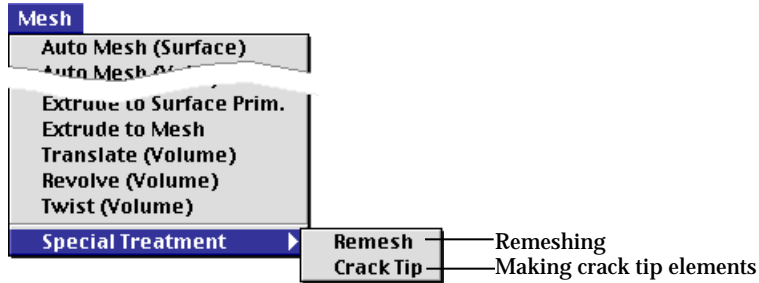< Resizing meshes by rubber-banding an edge of the bounding box >

Instead of resizing only in one edge direction, you may resize the meshes with equal relative scale in X, Y Z direction. This can be done by pressing ⌘ key (Windows : Alt key) while resizing the meshes following the above procedures.

Only the dimension in Z direction is resized.

The dimensions in X, Y and Z directions are concurrently resized

Resizing without ⌘ key(Windows: Alt key) pressed

Resizing with ⌘ key(Windows: Alt key) pressed

< Resizing meshes with and without ⌘ key (Windows : Alt key) pressed >
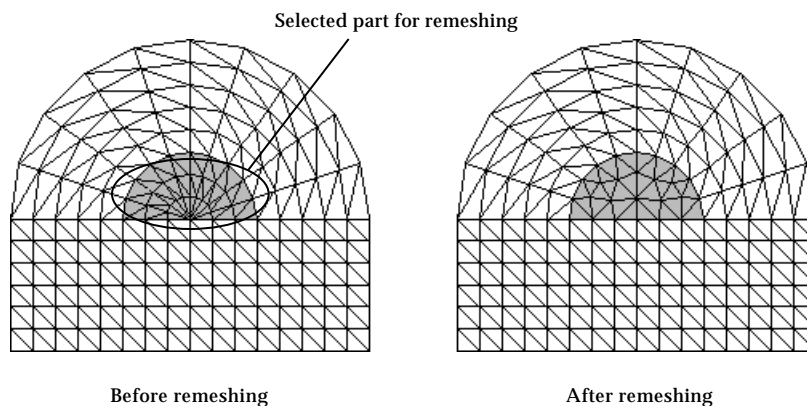
Other mesh treatments

The are functions for special treatments of meshes. The current version of VisualFEA supports 2 functions: creating a crack tip and remeshing.

■ Remeshing

A selected part of a surface mesh can be remeshed. Automatic mesh generation algorithms are used in remeshing operation.

1) Click the element selection tool 🔾, if it is not in pressed state.
   There appears **Select** menu at the right end of the menu bar. "Select Surface Elem." item of the menu should be checked.
2) Select the region of elements to remesh.
   The selected elements should be within a surface mesh and form a unified region. Otherwise, remeshing will not work.
3) Choose "Remesh" item from **Special Treatment** submenu.
   If all the selected elements have quadrilateral shape, quadrangle elements are generated by remeshing. Otherwise, triangular elements will be generated.

Selected part for remeshing

Before remeshing   After remeshing

< Example of remeshing >

Remeshing cannot be undone, and therefore the original mesh cannot be recovered. It is a good strategy to save the model before remeshing, so that the original mesh can be recovered in case the remeshing produces unsatisfactory result.
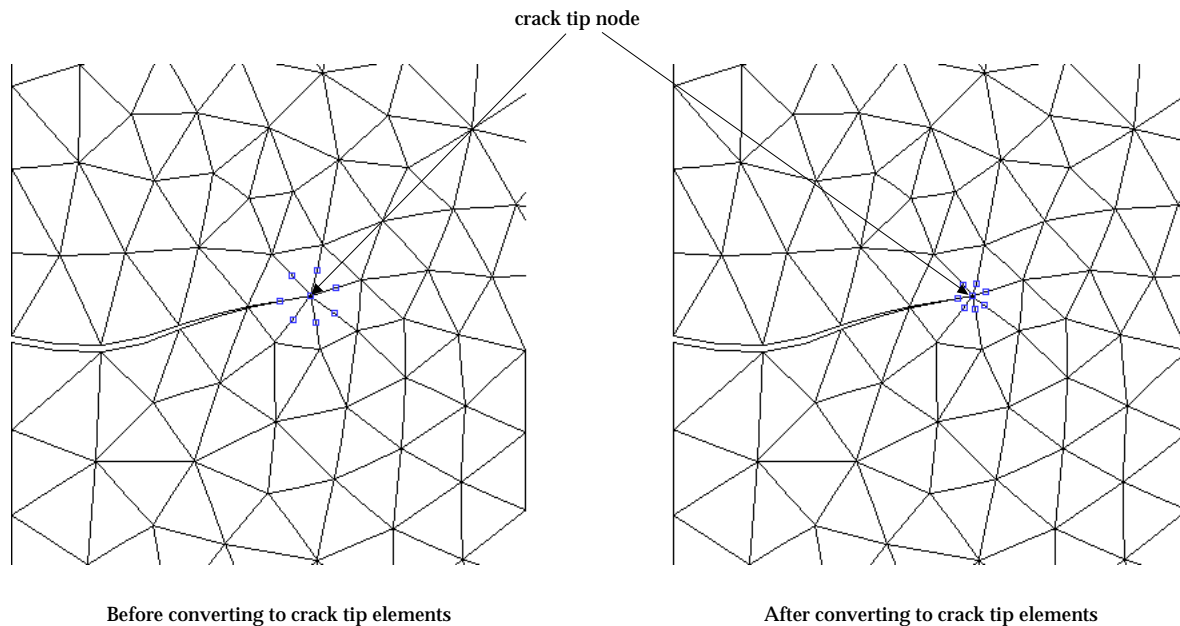
■ Making crack tip elements

The $1/\sqrt{r}$ stress singularity at a crack tip is usually modeled by using quarter-point elements in which the mid-side nodes are positioned at one quarter of the edge length from the crack tip node. The elements surrounding a crack tip can be converted into quarter-point crack tip elements by the following steps:

1) Click the node selection tool ⬛, if it is not in pressed state.
2) Select the crack tip node.
   The crack tip node should be at the tip of the crack, and the surrounding elements should be of quadratic order, i.e., **6 node triangular or 8 node quadrilateral element.**
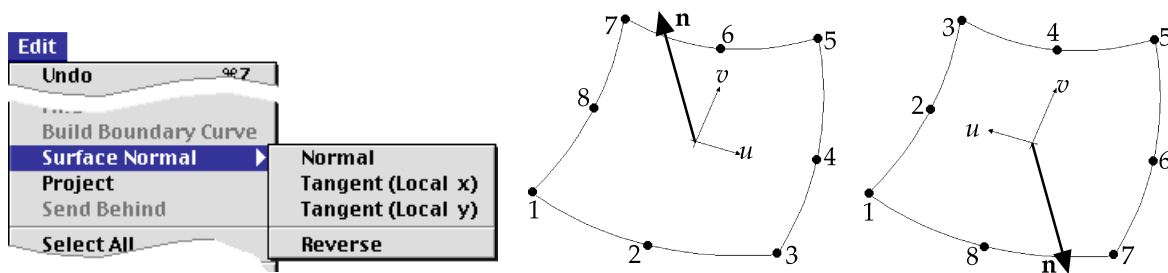3) Choose "Remesh" item from **Special Treatment** submenu.
   The surrounding elements are converted into quarter point crack tip elements. The mid-side nodes of the elements move to the quarter points along the edges toward the crack tip.

crack tip node

Before converting to crack tip elements   After converting to crack tip elements

< Example of creating crack tip elements >
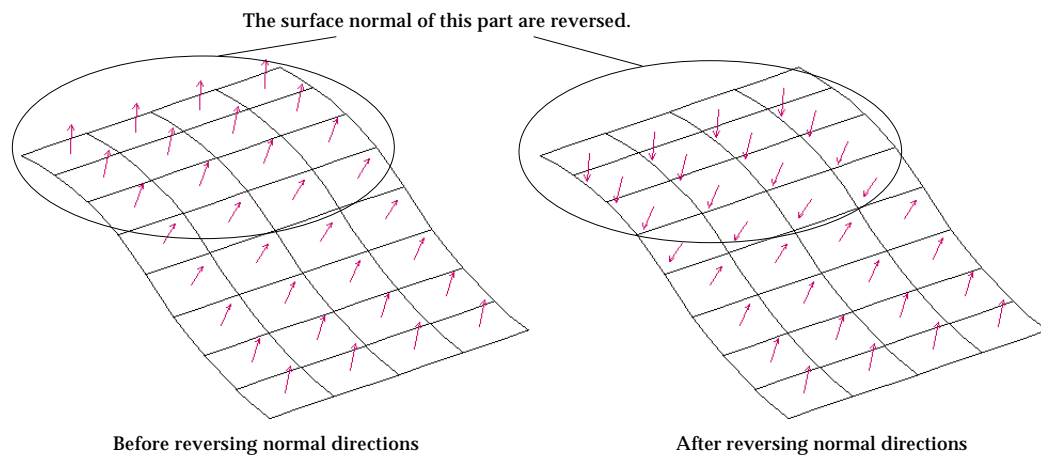
Surface normal and tangent directions

VisualFEA uses surface normal directions for various purposes including model rendering, force assignment, local coordinates of a shell element and so on. The normal direction at a specific point on a surface mesh is determined by the element containing the point. It is related to the node numbering within the element. The tangent directions are especially important for a shell element with its local coordinates based on the surface normal and tangent directions.   They are defined by the surface normal and the global coordinate axes. You can identify the surface normal and tangent directions, and reverse the directions if necessary.

<Node numbering and normal direction on an element>

■ Displaying the surface normal direction

To display the surface normal, select surface meshes whose normal directions are to be displayed. And choose the "Normal" item of **Surface Normal** submenu. Then, a surface normal vector is drawn at the center of each element constituting the surface meshes.

The surface normal of this part are reversed.

Before reversing normal directions   After reversing normal directions

<Example of reversing surface normal direction>

■ Displaying surface tangent directions

There are two tangent directions which makes 3 orthogonal axes along with the surface normal. The tangent directions are also the local coordinates axes used in shell elements. To display the directions tangent to the surface, select surface meshes whose tangent directions are to be displayed. And choose the "Tangent (Local x)" or "Tangent (Local y)" item of **Surface Normal** submenu. The local x direction of a shell element is displayed by selecting "Tangent (Local x)" item, and the local y direction by "Tangent (Local y)".

■ Reversing surface normal directions

The surface normal directions can be reversed by selecting surface meshes and choosing "Reverse" item of **Surface Normal** submenu. Reversion of a normal vector on an element results in reversing the node numbering within the element.