

OSM

Menu & Dialog

Reference

Version 2.6

2003

1 Introduction

This manual is a reference for the OSM menus and associated dialog boxes. OSM is a simple object oriented geometric modeler designed to generate geometric descriptions of objects suitable for importing into FRANC3D. It has a simple, but flexible object description language that is used to store object descriptions in the .mod restart files. OSM can be used to read .mod files generated by hand or by other programs.

1.1 Basic Interaction

1.1.1 Mouse Buttons

OSM uses a three-button mouse. The functions assigned to the buttons are as follows:

Left Button

Is used to select menu commands, to select objects (see below), to resize windows, and to display axis boxes and pulldown menus.

Center Button

Is used to “pop” windows or menus that are partially covered to the forefront so that they are not covered by any other windows or menus.

Right Button

Is used to move windows or menus. Holding down the mouse button and dragging the window or menu moves the item to a new location.

1.1.2 Menus

The OSM menus are displayed at all times. Individual menu commands can be either active or inactive. Inactive menu commands are displayed in a contrasting color, and cannot be selected. The status of all menu commands is updated continually. Most menu commands act on the currently selected objects in the currently active geometry window.

1.1.3 Selections

Most OSM menu commands operate on the currently selected objects. Selected objects are displayed in a highlighted color. There are two ways to select objects, click selects and drag selects.

Click Select

A click select is activated by a click of the left mouse button with the mouse pointer positioned over an object. Normally only one object at a time is selected this way. However, if the Shift key is held down during a click select, the new object is added to the selection. If you Shift-Click over an object that is already selected, the object is removed from the selection.

Drag Select

A drag select is activated by pressing down the left mouse button and, with the button still down, dragging the mouse to a new position. A “rubber-band” highlight will be displayed to show the drag box. When the mouse button is released, all objects that fall completely within this drag box will be selected. The Shift key can be used with a drag select to add to the current selection. Objects cannot be removed from the selection by a drag select.

2 OSM Widgets

In addition to the menus, described in section 3, there are four widgets that are always part of the OSM screen (Figure 2.1). These are:

- one or more geometry modeling windows,
- the view manipulation buttons that allow the user to rotate, zoom, pan and clip the current geometry window,
- the Select–Display box that allows the user to change what classes or types of objects are selected and displayed,
- the title box, which can be altered in the 'Osm' resources file (see section 5), and
- the Current Collection box.

2.1 Geometry Modeling Windows

A geometry modeling-window is created every time you select the **Read OSM Model File** or the **Create New Model** command. A geometry modeling-window is shown in Figure 2.2. Normally there are six parts of the geometry window visible. The parts are:

Axis Box

When depressed, this pops up a small window showing the current orientation of the Cartesian coordinate axis

Label Bar

This region displays the name of the .mod file name associated with this geometry. In addition, if this region is depressed and shaded, this indicates that this window is the currently active window.

Resize Box

When this box is depressed, moving the mouse while keeping the mouse button down allows you to resize the window. A “rubber-band” outline of the new window size is displayed.

Pulldown Menu Bar

This region displays the labels for the pulldown menus (currently there is only one standard pulldown menu, the View menu described below). Selecting the menu label displays the associated menu.

Display Area

This is the majority of the window and displays the current geometry.

Window Kill Box

When depressed, a dialog box is presented stating that all data will be removed. Select Ok to remove the window and the model data contained in it. The kill box is not shown in Figure 2.2, but is shown in Figure 2.1; it is located next to the axis box.

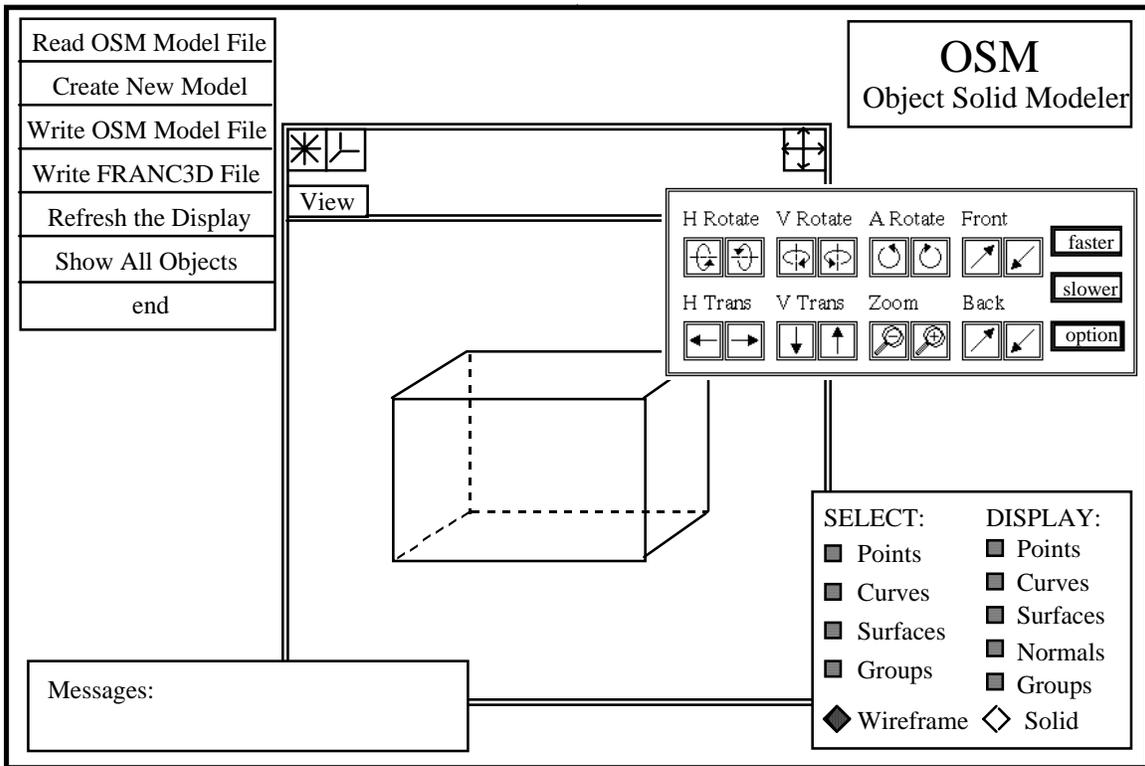


Figure 2.1. OSM modeling environment. Note that not all the menus are shown here.

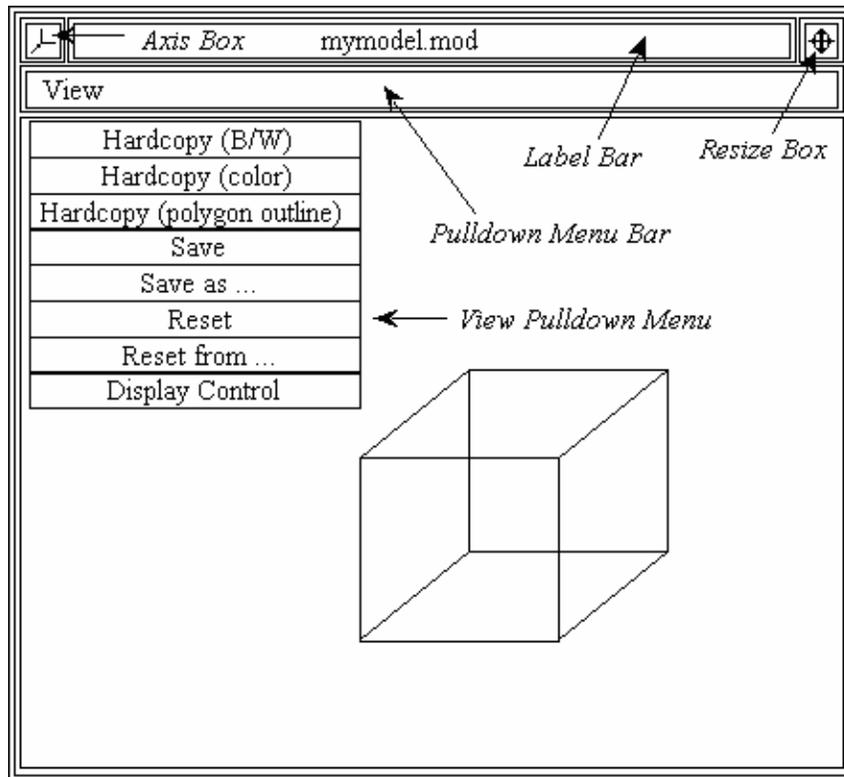


Figure 2.2. OSM modeling window.

2.1.1 Main Modeling Window Pull-Down Menus

There is one standard pull-down menu associated with the OSM geometry windows, the **View** menu shown in Figure 2.2. Specific commands on this menu are:

Hardcopy (B/W)

Generates a black and white postscript file of the current view. A file selector box is presented with a list of all the .ps files. You can overwrite an existing file or create a new file by entering the name in the Filename text field. The .ps extension will be added for you if you do not type it. These files may be printed directly on a postscript printer.

Hardcopy (color)

Generates a color postscript file of the current view. See Hardcopy (B/W) for details.

Hardcopy (polygon)

Generates a postscript file of the current view with surfaces shaded and hidden lines behind the shaded surfaces not shown. See Hardcopy (B/W) for details.

Save

Save the current camera position as the default.

Save as...

Save the current camera position to a file. A file selector box is presented with a list of all the .cam files. You can overwrite an existing file or create a new file by entering the name in the Filename text field. The .cam extension will be added for you if you forget to type it.

Reset

Reset the view based on the default camera position stored by **Save**.

Reset from...

Reset the view based on the camera position from a file. A file selector box is presented with a list of all the .cam files. You can select a file by double clicking on the file name or by highlighting the file name and then selecting **Accept**.

Display Control

Allows the user to turn on and off the display of vectors and polygons.

2.2 Select/Display Specification

The Select–Display widget (Figure 2.3) allows you to specify which items in the geometry windows are selected and displayed.

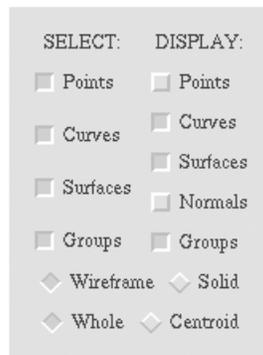


Figure 2.3. Select–Display widget.

There are four parts to this widget. The select options allow you to specify individually if Points, Curves, Surfaces, and Groups will be selected; remember you can either click-select or drag-select. The selection sensitivity for each type of option can be specified individually.

The display options allow you to turn the display of Points, Curves, Surfaces, surface Normals, and Groups on and off individually. Note however, that the display status of a class of objects does not affect their selection sensitivity.

The Wireframe/Solid toggle allows you to specify if surfaces are to be displayed by wireframe outlines or as shaded solid surfaces. The Whole/Centroid toggle allows the user to pick entities using a bounding box to select either the entire object or just the centroid of the object. Note that centroid-based picks require the user to hold the Ctrl key down while holding the left mouse button down and dragging the mouse over the entities in the modeling window.

2.3 View Specification Buttons

The view of the object or the camera position in the main modeling window is controlled with the use of the buttons in the view control panel (Figure 2.4). These buttons allow the object to be rotated and translated. The camera can be zoomed in on a portion of the object. Front and back clipping planes can be used to eliminate portions of the object from the current view (Figure 2.5). Each of the buttons in Figure 2.4 is now described.

The object is considered to be at a 'reference position' that is viewed from a 'camera position'. Changing the relative position of the camera with respect to the reference position controls the view. In addition, front and back 'clipping planes' can be toward or away from the camera and reference positions to cut away portions of the object that can be seen on the screen.

H Rotate

Rotates the model about a horizontal axis. The left button rotates the model upwards and the right button rotates the model downwards.

V Rotate

Rotates the model about a vertical axis. The left button rotates the model to the left and the right button rotates the model to the right.

A Rotate

Rotates the model in a clockwise/counterclockwise direction (about an axis that extends out of the screen). The left button rotates the model counterclockwise and the right button rotates the model clockwise.

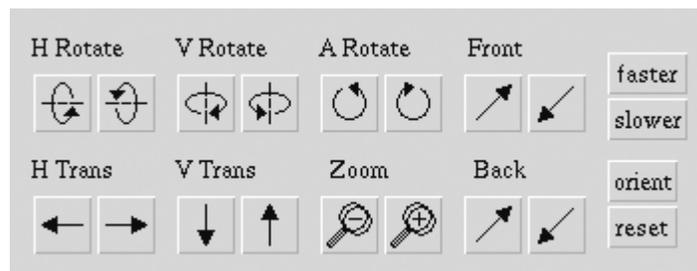


Figure 2.4. View control panel.

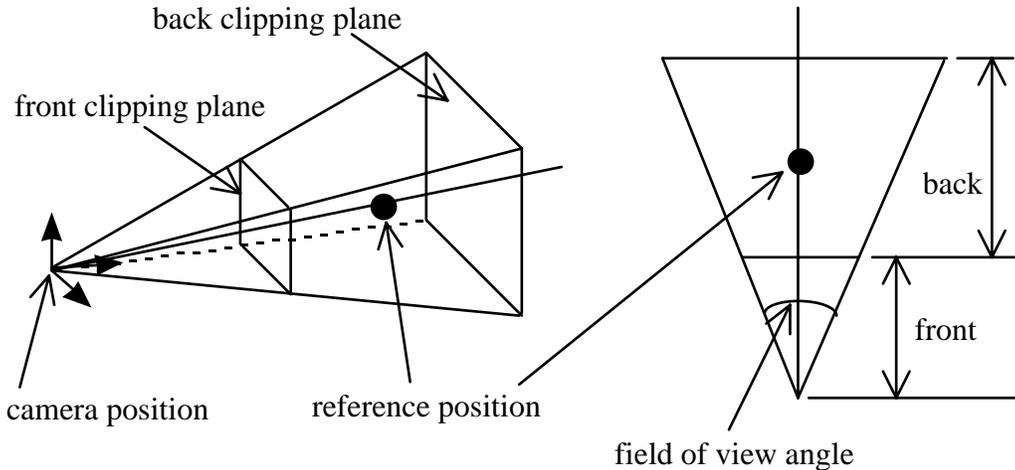


Figure 2.5. Reference and camera positions and front and back clipping planes.

Front

Moves the front clipping (cutting) plane. The left button, , pushes the front plane in towards the model; the right button, , pulls the front plane out away from the model towards the camera position. Pushing the front plane towards the model allows you to cut away the front portion of the object in order to see the interior of the model.

H Trans

Translates the model in the horizontal direction. The left button moves the model to the left and the right button moves the model to the right.

V Trans

Translates the model in the vertical direction. The left button moves the model downward and the right button moves the model upward.

Zoom

The left button zooms out (pushes the model away making it appear smaller) and the right button zooms in.

Back

Moves the back clipping (cutting) plane. The left button  pushes the back plane away from the model and the right button  pulls the plane forward towards the model. Pulling the back plane towards the model and camera position allows you to cut away the back of the object so that you can see the front portion more clearly.

faster/slower

These two buttons control the speed of the above buttons. The speed is incremented each time one of the buttons is pressed.

option

This button presents a dialog box that allows the user to snap the model into a view along one of the axes.

reset

Resets the view of the model to the initial default view.

The view can also be controlled with the mouse along with the Shift and Ctrl buttons. Holding the right mouse button down and dragging it allows the user to translate the model in the drag direction. Holding the Shift button down while holding the right mouse button down and dragging the mouse allows the user to rotate the model. Holding the Ctrl button down while holding down the right mouse button and dragging the mouse allows the user to zoom in and out. Holding the Shift button down and holding the middle mouse button down moves the front cutting plane forward and backward. Holding the Ctrl button down and holding the middle mouse button down moves the back cutting plane forward and backward.

2.4 Current Collection Dialog

The number of entities that are currently collected is provided in a dialog box, Figure 2.6 that is initially located in the lower right corner of the OSM screen.

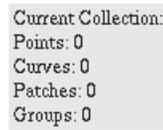


Figure 2.6. The Current Collection dialog.

3 Menus

3.1 OSM File Menu

This is the main menu in OSM (Figure 3.1), allowing you to open up new modeling windows, save files and redraw objects in the modeling windows.



Figure 3.1. Main OSM menu.

Concepts for this menu:

Geometry file formats - OSM reads and writes geometry in the ascii .mod format and in the portable binary .osm format. The format for the .mod file can be found in the OSM Model File Format section of this manual. The binary .osm file is portable and also stores additional attribute information that the .mod file cannot store. For instance, interior surfaces are stored with an extra attribute in the .osm file so that OSM recognizes these as interior when reading the file. OSM can write surface geometry in the FRANC3D .dat input format.

Specific Commands on this menu:**Read OSM Model File ...**

A sub-menu, Read Ascii File / Read Binary File is presented. For ascii .mod files, see the OSM Model File Format section for detailed information of the file format.

- OSM displays a list of all .mod or .osm files found in the current directory inside of a file selector box.
- The user selects the file to be read by either double clicking on the file name with the left mouse button or by highlighting the file name (single click with the left mouse button) and selecting **Accept**.
- A new model window is created and the model is displayed in this window.

Create New Model

Creates a new empty modeling window.

Write OSM Model File ...

A sub-menu, Write Ascii File / Write Binary File is presented. The geometry information in the currently active geometry window is written to a file.

- OSM prompts for the name of the file to be written by presenting a file selector box. Overwrite a file by selecting an existing file name or enter a new file name in the text entry field. The extension will be added if you do not type it.

Write FRANC3D File ...

Writes surface geometry information into a .dat file in a format that can be read by the FRANC3D program. There are two filing options depending on whether the model is a solid or shell.

3D Solid ...

This subcommand saves the information for a closed 3D solid.

Thin Shell Model ...

This subcommand saves the information for a shell surface (or 2D) model.

- OSM prompts for the name of the file to be written by presenting a file selector box. Overwrite a file by selecting an existing file name or enter a new file name in the text entry field. The .dat extension will be added automatically if you do not type it.

Refresh the Display

Redraws the current geometry window, but first re-scales the point boxes and surface normals to a size appropriate to the current view.

Show All Objects

Selects a view for the currently active geometry window such that all objects contained in that geometry are seen and redraws the objects based on this view.

end

Exits the OSM program.

3.2 Select Menu

This menu (Figure 3.2) allows the user to select, copy, cut, paste, group, and ungroup objects or sets of objects.



Figure 3.2 Select menu.

Concepts for this menu:

Selection - Before modeling operations can be performed, one or more objects must be selected. Selected objects are drawn in a highlight color.

Cut Buffer - The cut buffer is a temporary storage space that is used by the **Cut**, **Copy**, and **Paste** commands. Objects must be selected before they can be placed in the cut buffer. Objects in the cut buffer can be “pasted” into any geometry window.

Groups - objects can be grouped together. Grouped objects act as one object for the purposes of selection. Groups can be hierarchical. That is, groups can contain other groups.

Specific Commands on this menu:

Select All

Selects all the objects in the currently active geometry window.

Clear Selection

Clears the selection so that nothing is selected.

Cut

Removes the currently selected objects from the geometry window and places them in the cut buffer.

Copy

Makes a copy of the objects that are currently selected in the geometry window and places these copies in the cut buffer.

Paste

Makes a copy of the objects in the cut buffer and pastes these into the currently active geometry window.

Paste Special

Allows you to translate and rotate the objects in the cut buffer before they are pasted into the currently active geometry window. There are two sub-options for this command (Figure 3.3).



Figure 3.3. Paste Special sub-menu.

Cartesian Translate ...

This subcommand presents the following dialog box (Figure 3.4) that allows you to specify a translation along the three Cartesian directions.



Figure 3.4. Cartesian translation dialog box.

Rotate About Axis ...

This subcommand presents the following dialog box (Figure 3.5) that allows you to specify a rotation about one of the Cartesian axes or about an arbitrary axis specified by two points.

ROTATE ABOUT AN AXIS			
Amount of Rotation	0		degrees
axis	<input checked="" type="radio"/> x axis	<input type="radio"/> y axis	<input type="radio"/> z axis
arbitrary axis	<input checked="" type="radio"/> x	<input type="radio"/> y	<input type="radio"/> z
point 1	0	0	0
point 2	0	0	0
Accept		Cancel	

Figure 3.5. Rotate About Axis dialog box.

Group

Puts all the currently selected objects into a new group.

Ungroup

Takes all the objects in any currently selected group and converts them back into individual objects again.

3.3 Modeling Menu

This menu (Figure 3.6) contains commands to create entities in the active geometry window.



Figure 3.6. Modeling menu.

Specific Commands on this menu:

Type In Points

Allows the user to use the keyboard to enter new points. When this command is selected, the dialog box shown below is displayed (Figure 3.7). You enter the coordinates of the points into the spreadsheet and press Accept to add these points into the current model window. You can enter points directly into the spreadsheet or you can enter the coordinates in the lower x-y-z coordinate boxes

and select **Add** to add the coordinates into the spreadsheet. This is useful if one or two of the coordinates is constant. You can delete a row from the spreadsheet by highlighting the row (select a box in the row) and pressing **Delete**. Select **Import** to read points from a file. Select **Clear** to remove all points from the spreadsheet. Select **Cancel** to remove the dialog box without adding any points to the modeling window. Note that if duplicate point coordinates are entered, then duplicate points will be created. Also, only the entries in the main spreadsheet are added as points to the active model window.

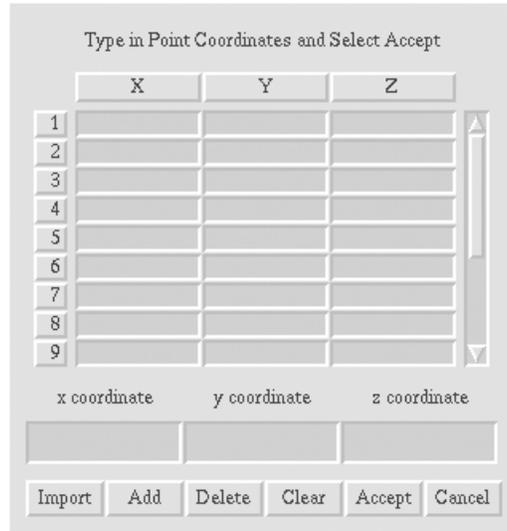


Figure 3.7. Type In Points dialog box.

Read Points From File

Allows you to read point coordinates from a file. The file should contain the coordinates for one point on each line. Each line contains the x, y, and z coordinates of the point. A simple example of 3 points is shown below:

```
1.0  0.0  3.0
2.5  3.2  3.5
0.54 0.3e-1 3.5
```

Create A Curve

Creates a spline curve that interpolates the selected points. The points are ordered to minimize the arc length of the generated curve. If only two points are selected, a straight line is added between the points.

Create A Circular Arc

Creates a curve that is an arc of a circle. This command becomes active if three points are selected. There are three possible cases for the relative distances between the points. 1) If the distances between one point and the other two are the same, this point is used as the center of the circle, and the common distance is

the radius. 2) If the distances between all three points are the same, the program will put up a dialog box that asks you which point to use as the center of the arc. 3) If the distance between all three points is different, an error message is displayed. Note that the arc is ambiguous if the angle is exactly 180 degrees, thus, no arc is created.

Create Line Segments

Adds a series of straight-line segments through the selected points. The points are ordered to minimize the total length of all the segments.

Create A Patch by Corners

Creates a triangular (if three points are selected) Bezier or quadrilateral (if four points are selected) bicubic spline surface patch. If curves joining any of the points exist, these curves are used for the corresponding patch edges. If curves do not exist, a straight-line geometry is assumed and created.

Create A Patch by Curves

Creates a triangular (if three curves are selected) Bezier or quadrilateral (if four curves are selected) bicubic spline surface patch.

Create Patch: Curves & Corners

Creates a triangular (if three corners are selected) Bezier or quadrilateral (if four corners are selected) bicubic spline surface patch. The user collects the curves and the corner points of the patch. Multiple curves can be used to define on edge of the patch.

Create Planar Polygon

Creates a planar polygon surface patch from the selected set of points. A plane is fit to the set of points based on a least-squares fit and a polygon is created based on the order of the collected points.

3.4 Operation Menu

This menu (Figure 3.8) contains commands to perform constructive modeling operations in a geometry window.

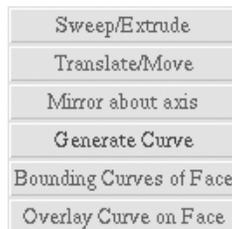


Figure 3.8. Operation menu.

Specific Commands on this menu:

Sweep/Extrude

Allows you to sweep the selected objects along a straight or circular path to create new geometry. Points are swept to create curves and curves are swept to create surfaces. There are two subcommands for this option, Figure 3.9.



Figure 3.9. Sweep submenu.

Cartesian Sweep...

This subcommand presents the dialog box seen in Figure 3.4 that allows you to specify a sweep translation along the three Cartesian directions.

Sweep About Axis ...

This subcommand presents the dialog box seen in Figure 3.5 that allows you to specify a sweep about one of the Cartesian axis, or about an arbitrary axis specified by two points.

Translate/Move

This command allows you to translate (move) the selected objects along a straight path or to rotate them about an axis. There are two subcommands for this option, Figure 3.10.



Figure 3.10. Translate submenu.

Cartesian Translate ...

Presents the dialog box shown above for a *Cartesian Sweep* (Figure 3.4), and moves the selected objects by the specified amounts.

Translate About Axis ...

Presents the dialog box shown above for a *Sweep About Axis* (Figure 3.5), and rotates the selected objects by the specified amount about the specified axis.

Mirror about axis

This command creates mirror copies of the selected objects about planes that are parallel to the Cartesian axis. The following dialog box (Figure 3.11) is displayed that allows you to specify the Mirror plane:



Figure 3.11. Mirror Plane dialog box.

Generate Curve

A submenu is presented (Ellipse points... / Cancel) allowing the user to generate points based on predefined functions. If the user selects **Ellipse points**, a dialog box (Figure 3.12) is presented to define the number of points, the major and minor axis lengths and the amount of the full (360 degree) ellipse to generate.

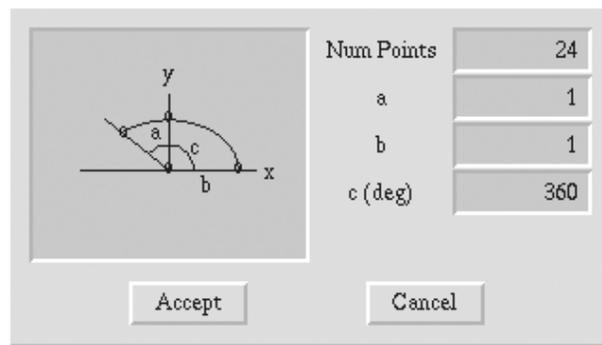


Figure 3.12. Ellipse points dialog box.

Bounding Curves of Face

This command determines the bounding curves of a surface patch and creates edges to represent the boundary of the patch if the curves do not exist.

Overlay Curve on Face

This command overlays a curve onto a surface patch. The user must collect a surface patch and two points on the patch. A curve that lies on the surface patch is created between the two points.

3.4 Information Menu

This menu (Figure 3.13) provides options for getting point coordinates, angles, and evaluating points along curves.

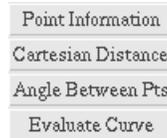


Figure 3.13. Information menu.

Specific Commands on this menu:

Point Information

Displays the coordinates of the selected point.

Cartesian Distance

Displays the distance between the two selected points.

Angle Between Pts

Displays the angle between the three selected points.

Evaluate Curve

Allows you to evaluate a selected curve at a specified number of points and write this information to a file or to the **Type in Points...** dialog box. A submenu is presented allowing the use to choose *to File* or *to Dialog*. Once selected, this command puts up a dialog box that allows you to specify the number of points at which the curve will be evaluated. If *to File* is chosen, a dialog box is then presented to enter the file name (Figure 3.14); specify the name of the file for the evaluated points. If *to Dialog* is chosen, the evaluated points are presented in the dialog box shown in Figure 3.7 and the user is then able to edit these points before adding them to the model.

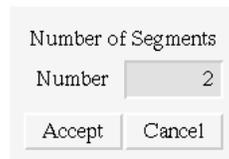


Figure 3.14. Number of Segments dialog box.

3.4 Special Operations Menu

This menu (Figure 3.15) provides options for merging or splitting curves, removing duplicate nodes, cutting curves, reversing surface normal directions, and defining interior surfaces.

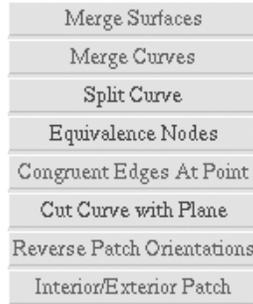


Figure 3.15. Special operations menu.

Specific Commands on this menu:

Merge Surfaces

Allows you to merge two adjacent surfaces into a single surface. Currently, only surfaces that are defined by exactly four bounding curves can be merged together. The original surfaces are retained along with the new surface.

Merge Curves

Allows you to merge two or more selected curves that form a continuous chain into a single curve.

Split Curve

Allows you to split a selected curve into multiple segments. There are two subcommands for this option, Figure 3.16.



Figure 3.16. Split Curve submenu.

At Defined Point ...

Presents a dialog box for entering the x-y-z coordinates of the point where the curve is to be split. By default the mid-point of the curve is given initially. Two new curves are created and the original curve is deleted.

In Equal Segments...

Presents the dialog box seen in Figure 3.14 for entering the number of new curve segments. Note that multiple curves can be selected for this subcommand, and all curves will be split accordingly. The specified number of new curves are created, and the original curve is deleted.

Equivalence Nodes

Allows you to remove duplicate nodes (points). There are two subcommands for this option, Figure 3.17.

Determine Closest Nodes
Remove Duplicate Nodes

Figure 3.17. Equivalence Nodes submenu.

Determine Closest Nodes

Computes the distance between all pairs of nodes in the model and displays the distance between the pair that is closest in an acknowledge box as well as highlighting the pair of nodes.

Remove Duplicate Nodes

Presents the dialog box seen in Figure 3.18 for entering the tolerance to be used in removing duplicate nodes. All pairs of nodes that are within the defined tolerance are merged to remove one node from the pair.

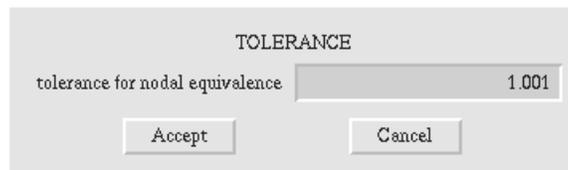


Figure 3.18. Equivalence Nodes tolerance dialog box.

Congruent Edges At Point

Makes a set of edges congruent at a single point. Edge geometry is stored independently of points that define the edges. To ensure that all edges use the exact point location in their geometry a user can pick a set of edges and a single point and make the edges join exactly at the point.

Cut Curve With Plane

A curve or set of curves can be cut by a user defined plane. The plane is defined by entering three points (using the Add Points dialog box, Figure 3.7). The selected curve(s) are cut into two curves at the intersection of the curve and plane; the original curve is removed.

Reverse Patch Orientation

Reverses the orientation (direction of the normal) of all selected patches. There are two subcommands for this option, Figure 3.19.

Reverse Patch Geometry
Reverse Patch Color

Figure 3.19. Reverse Patch submenu.

Reverse Patch Geometry

Reverses the actual geometric description of the surface.

Reverse Patch Color

Switches the color from red to green or green to red. The patch colors are seen when viewing the model with surface normals on in Solid mode, Figure 2.3.

Interior/Exterior Patch

Switches the surface patch attribute to create interior or exterior surfaces. There are two subcommands for this option, Figure 3.20.

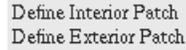


Figure 3.20. Interior/Exterior Patch submenu.

Define Interior Patch

Assigns the patch as interior. Interior patches are shown in a different color compared to the default blue of the exterior patches.

Define Exterior Patch

Assigns the patch as exterior.

3.5 FEM Conversion Menu

This menu (Figure 3.21) provides options extracting a boundary surface representation from a 3D finite element model. FEM data is translated to a single common format (.conn file) and then stripped of all the interior element faces and edges. The resulting multi-faceted boundary surface mesh is then used to regenerate an approximate description of the original geometry. See Section 6 for additional explanations and an example.



Figure 3.21. FEM conversion menu.

Specific Commands on this menu:**Create Radial-Edge Database**

Create a radial edge database from the surface patches in the currently active window. The surface patches must form a closed manifold solid to create a valid database. If the database is not valid, a message is displayed and the problem area of the model is highlighted.

Highlight Geometry Points/Edges

Determine the key geometry points and edges in the FEM data. A dialog box is presented for entering the geometric angular tolerance (Figure 3.22). Each FEM surface node is examined to see if the local FEM face normals differ by more than the specified tolerance. If so, the point is retained as a key point. Key edges are found by traversing the FEM edges between key points. All final key points and edges are then highlighted in the modeling window.

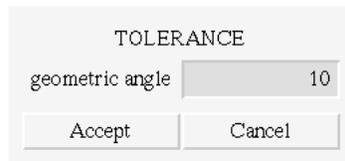


Figure 3.22. Geometric angle tolerance dialog.

Extract Geometry Features

The highlighted key points and edges of the previous command are used to find and create the boundary surface representation. FEM edges are merged to form larger edges and new surface patches are created for all valid closed loops of three or four merged edges. The new boundary surface representation is copied and pasted into a new modeling window.

Convert old .conn/.fstr Files

Reads an old style .conn neutral file along with an old .fstr file and converts these to a new MRP file format. The old style .conn files were created by OSM up to version 2.3.

Translate BLADE-GT

Reads a GEOM3D file from Stress Technology Inc. and converts it to a .conn file format. Select the desired GEOM3D file by double clicking on the file name with the left mouse button. A dialog box is presented to allow the user to work with a portion of the model as defined by the GEOM3D component number. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with

many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate SDRC-IDEAS

Reads a Universal file from IDEAS and converts it to a .conn file format. Select the desired Universal file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate ANSYS .cdb

Reads a .cdb neutral file from ANSYS and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to an .osm file automatically and then displayed in an OSM modeling window.

Translate NASTRAN

Reads a neutral file from NASTRAN and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in a OSM modeling window.

Translate PATRAN

Reads a neutral file from PATRAN and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate ABAQUS

Reads a .inp file from ABAQUS and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The

resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate MARC K6

Reads a neutral file from MARC and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate DIANA

Reads a neutral file from DIANA and converts it to a .conn file format. Select the desired neutral file by double clicking on the file name with the left mouse button. The interior FEM faces, edges, and nodes of the model are then stripped away. The process of stripping out the interior data can be time consuming for models with many elements; the progress is displayed on the terminal window. The resulting surface FEM data is saved to a .mod or .osm file automatically and then displayed in an OSM modeling window.

Translate ANSYS .igs

Reads an ANSYS generated IGES file and creates an OSM model. Select the desired file by double clicking on the file name with the left mouse button. The file is processed and the data is saved to a .mod file automatically and then displayed in an OSM modeling window.

4 OSM Model File Format

All lines beginning with the '#' character in the first column are interpreted as comments. Comments extend to the end of the line.

The syntax conventions used below are:

Items in square brackets “[]” are optional.

“< a | b >” means a or b.

Items in normal font should appear exactly as shown.

Items in *italic* font are placeholders, and substitutions should be made for them.

4.1 The Define Directive

The define directive is used to define new geometrical objects. There are six forms of this directive. These are described individually below.

define point [*name*] = { *x y z* }

Defines a new point with the specified x, y, and z coordinates.

define curve [*name*] = {{*point 1*} {*point 2*} ... {*point n*}}

Defines a spline curve. The *point x* values can be either x, y, and z coordinates, or the *name* of one of the points defined previously. This is also true for all the point values in all of the forms of the directive.

define line [*name*] = {{*point 1*} {*point 2*}}

Defines a straight line between the two specified points.

define arc [*name*] = *num_segs* {{*point1*} {*point 2*} {*center*}}

Defines a circular arc. The points specify the starting location, the ending location, and the center of the arc. All arcs are assumed to be $< 180^\circ$. The *num_segs* is an integer value that is required, but currently ignored.

define patch [*name*] = *u_points v_points* {{*point 1*} {*point 2*} ... {*point u*v*}}

Defines a quadrilateral spline patch. *u_points* and *v_points* specify the number of points in each of the two parametric directions. The definition for a simple patch is shown in Figure 4.1.

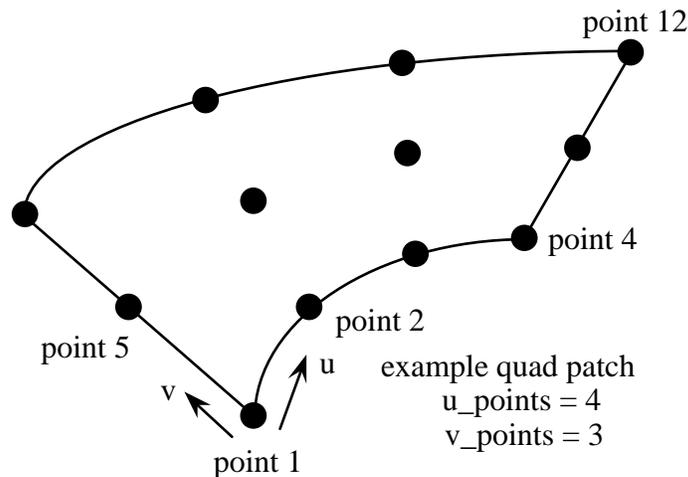


Figure 4.1. Four-sided surface patch.

define tri_patch [*name*] = {{*point 1*} {*point 2*} ... {*point 10*}}

Defines a triangular spline patch. All triangular patches are defined by ten interpolation points (Figure 4.2).

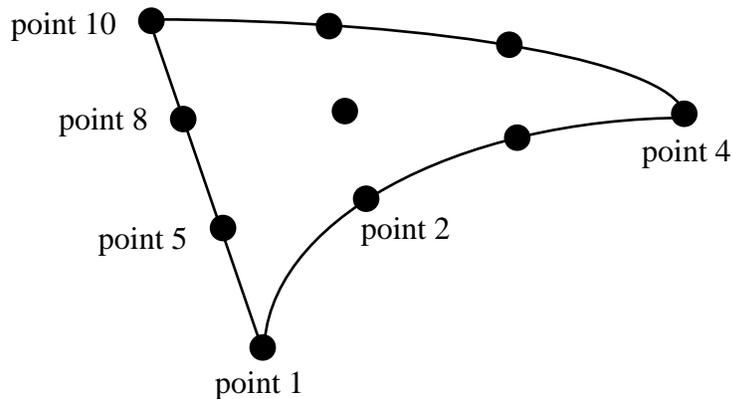


Figure 4.2. Three-sided surface patch.

```
define pln_patch [name] = num_corners {{point1} {point 2} {point3} {point4}}
```

Defines a planar patch, either a triangle or quadrilateral depending on the number of defined corners.

Note that patches can be differentiated as exterior or interior by specifying this after defining the points or corners. For example, the following is an exterior planar patch:

```
define pln_patch [name] = num_corners  
  {{point1} {point 2} {point3} {point4}}  
  {exterior}
```

4.2 The Configure Directive

The configure directive is used to affect automatically generated object names. There are three parts to an automatically generated name: a prefix, which is optional, a name, and a number:

[prefix_]name_number

For example, *brick_frontPoint_3*, and *backFace_5*, are both possible names. The configure command allows you to set the prefix, name, and next assigned number for all subsequently generated names for points, edges, and surfaces. The default is no prefix, the names *Point*, *Edge*, and *Surface*, respectively, and numbers beginning with zero.

There are four forms of this directive as shown below.

```
configure point <name name | number number>
```

configure curve <name *name* | number *number*>

configure surface <name *name* | number *number*>

configure prefix *prefix*

4.2 The Group Directive

The group directive is used to the beginning and ends of groups in the file. There are two forms of this directive. The first indicates the start of a group along with an optional name for the group. The second form indicates the end of a group. Group definitions can be nested.

group start [*name*]

group end

4.3 An Example .mod File

An example .mod file is shown below.

```
#
# name generation configuration
#
configure prefix examp
configure point name mypoint
configure curve name mycurve
configure surface name mysurface
configure point number 4
configure curve number 9
configure surface number 8

#
# object definitions
#
define point exampmypoint_0 = { 0 0 0 }
define point exampmypoint_1 = { 1 0 0 }
define point exampmypoint_2 = { 0.5 0.8 0 }
define point = { 0 0 -2 }
define point = { 1 0 -2 }
define point = { 0.5 0.8 -2 }
define point exampmypoint_3 = { 0.5 1 -1 }
define line exampmycurve_0 = { { 0 0 0 } { 1 0 0 } }
define line exampmycurve_1 = { { 1 0 0 } { 0.5 0.8 0 } }
define line exampmycurve_2 = { { 0.5 0.8 0 } { 0 0 0 } }
define tri_patch exampmysurface_1 = {
```

```

    { 1 0 0 }
    { 0.666667 0 0 }
    { 0.333333 0 0 }
    { 0 0 0 }
    { 0.833333 0.266667 0 }
    { 0.5 0.266667 0 }
    { 0.166667 0.266667 0 }
    { 0.666667 0.533333 0 }
    { 0.333333 0.533333 0 }
    { 0.5 0.8 0 }
}
define arc examp_mycurve_3 = 5 {
  { 1.86265e-09 -1.86265e-09 -2 } { 1 1.86265e-09 -2 }
  { 0.5 0.8 -2 } }
define curve examp_mycurve_3 = {
  { 1.86265e-09 -1.86265e-09 -2 }
  { 0.189698 -0.0909054 -2 }
  { 0.394823 -0.137517 -2 }
  { 0.605177 -0.137517 -2 }
  { 0.810302 -0.0909054 -2 }
  { 1 1.86265e-09 -2 }
}

define line examp_mycurve_4 = { { 1 0 -2 } { 0.5 0.8 -2 } }
define line examp_mycurve_5 = { { 0.5 0.8 -2 } { 0 0 -2 } }
define tri_patch examp_mysurface_3 = {
  { 1.86265e-09 -1.86265e-09 -2 }
  { 0.325157 -0.127477 -2 }
  { 0.674843 -0.127477 -2 }
  { 1 0 -2 }
  { 0.166667 0.266667 -2 }
  { 0.5 0.202928 -2 }
  { 0.833333 0.266667 -2 }
  { 0.333333 0.533333 -2 }
  { 0.666667 0.533333 -2 }
  { 0.5 0.8 -2 }
}

define curve examp_mycurve_6 = {
  { 0.5 0.8 -1.49012e-08 }
  { 0.5 1 -1 }
  { 0.5 0.8 -2 }
}

define line examp_mycurve_7 = { { 1 0 -2 } { 1 0 0 } }
define patch examp_mysurface_5 = 2 3 {
  { 0.5 0.8 -1.49012e-08 }
  { 1 -1.49012e-08 -1.49012e-08 }
  { 0.5 1 -1 }
  { 1 -1.49012e-08 -1 }
  { 0.5 0.8 -2 }
  { 1 -1.49012e-08 -2 }
}

define line examp_mycurve_8 = { { 0 0 -2 } { 0 0 0 } }
define patch examp_mysurface_7 = 6 2 {
  { 1 0 -2.98023e-08 }
}

```

```
{ 0.8 -1.86265e-09 -2.98023e-08 }  
{ 0.6 -3.72529e-09 -5.96046e-08 }  
{ 0.4 0 -2.98023e-08 }  
{ 0.2 -9.31323e-10 0 }  
{ -3.72529e-09 0 -2.98023e-08 }  
{ 1 -1.86265e-09 -2 }  
{ 0.810302 -0.0909054 -2 }  
{ 0.605177 -0.137517 -2 }  
{ 0.394823 -0.137517 -2 }  
{ 0.189698 -0.0909054 -2 }  
{ -2.79397e-09 1.86265e-09 -2 }
```

5 OSM Resource File

The 'Osm' file contains resources which can be modified to change the colors, fonts, default sizes, and title of the OSM modeling environment. A description of the resources that can be set is given in Figure 28. The 'Osm' file should be placed in the user's home or login directory. If the file is not found, reasonable default values are used instead.

Name	Type	Default	Description
Font			
*facFont	Font	*times-medium-r-* 120*	Font used for menus and dialogs
*msgFont	Font	*courier-medium-r-* 100*	Font used for the message box
*facTitleFont	Font	*helvetica-bold-r-* 120*	Font used for title box
Color			
*facInsensitiveColor	Color	FireBrick	Color of inactive menu options
*facRemBackground	Color	DarkSlateGray	Background color of the main screen
*TdGraphics.background	Color	Black	Color of modeling window background
*tdgVectColor	Color	Orange	Color of object in modeling window
*facWritingColor	Color	MidnightBlue	Color used for menu text, dialogs, and simple graphics
*msgBorderColor	Color	Black	Border color for the message box
*messageBox.backgroud	Color	Tan	Background color for the message box
Title			
*FacTitle.facTitle	Text	OSM	Main title in the title box

Figure 28. Contents of the Osm resource file.

6 OSM Conversion of FEM Data

It is common to use a solid modeler to build complex geometry models of 3D structures. Automated 3D finite element mesh (FEM) generators then are used to create a corresponding mesh model for analysis. For example, ANSYS (ANSYS,¹ 1996) can be used to create a geometry model, create a mesh, and apply boundary conditions. Recent releases of ANSYS will write all of this information to various ascii files. The solid model geometry information is written as an IGES file. The mesh, boundary conditions, and other information are written to a neutral .cdb file.

Converting 3D FEM data to a solid model or boundary surface representation is rarely, if ever, done. There are instances, however, when this capability is very beneficial. For example, it is common to receive 3D FEM model data and the corresponding stress results as the starting point for doing detailed 3D crack growth simulations. Either the original model geometry information is not available electronically or it is difficult to convert the data to a form that OSM/FRANC3D can understand. Thus, the routines to convert automatically the 3D FEM data to a boundary surface representation for FRANC3D have been implemented within OSM. The routines and the process of extracting surfaces from discretized models are described herein.

The process of extracting the geometric boundary surface information begins once the .cdb file has been translated to the common format. This process of extracting boundary surfaces is complicated. In general, the complete boundary surface will not be extracted automatically. The user will often be required to finish and/or modify the extracted model before writing the FRANC3D input file. Some of these operations will be discussed below, but the user needs to be familiar with OSM in order to understand these discussions.

The FEM data represents the discretized geometry. Depending on the level of refinement of the finite element mesh, this information might not convey enough accurate geometric information to reconstruct the true model geometry. ANSYS can export true geometric descriptions of the model using the IGES format files. OSM is able to import these files and convert the information into its own internal geometric format. This effort is ongoing and is not completely functional yet.

¹ ANSYS (1996) ANSYS Computer Program, Release 5.3, Swanson Analysis Systems Inc., Houston, PA.

² IGES (1994) Initial Graphics Exchange Specification, U.S. Dept. of Commerce, National Institute of Standards and Technology.

6.1 FEM File Formats

OSM contains a set of menu buttons (see Figure 3.21) for converting FEM data to an OSM model. Two of the buttons on this menu allow one to translate ANSYS neutral (.cdb) and ANSYS IGES (.iges) file formats. ANSYS conversion is the most complete; however, the other commercial FEM formats should also generate correct OSM geometry. For ANSYS however, if the user has a .dsp file containing the nodal displacements and a .str file containing the nodal stresses, OSM will read those two files along with the .cdb file and create .mrp files that can be used in FRANC3D (see the boundary condition section of the FRANC3D Concepts and Users Guide). If the .cdb file has DISP commands for applied displacements, or SFE commands for applied surface pressure, or BF commands for nodal temperatures, then OSM will create separate .mrp files for each of these load types. The .dsp file results in a *_res_dsp.mrp file and the .str file results in a *_res_str.mrp file.

The built-in translators convert the ANSYS .cdb data to a single common format. The currently supported element topologies and connectivity numbering for this common format are shown in Figure 6.1. Additional element types can be added if necessary; midside nodes, if they exist, are removed from the elements when converting to osm geometry. The midside nodes remain in the .mrp files. Translators convert the element numbering in the ANSYS .cdb file into this format that is used for both OSM and FRANC3D. A .conn file is written once the .cdb data has all been converted. From the .conn file format, OSM is able to extract the important geometric information to describe the boundary surface representation of the model. Additional files are also written during this process, including the material data and boundary condition or results data.

6.2 FEM Volume Mesh to Surface Mesh

The first step in the process of extracting the boundary representation is to remove all the internal FEM data, keeping only the element information corresponding to the model surface. This is done by examining the connectivity of the elements, forming faces for each 3D element, and then removing all features that are in the interior.

Based on the element connectivity and nodal numbers shown in Figure 6.1, all the faces of all the elements are found and stored; faces are defined by the node numbers at the corners. All the edges of all the elements are then stored in a separate location; the edges are defined by the node numbers at either end. These set of faces and edges are examined to remove all duplicate entities. Each face or edge must be compared with all others in the list, and both forward and reverse ordering of the nodes on the entity must be checked. If a face or edge is used twice, then this entity is discarded. This removes all internal faces and edges and leaves only those faces and edges that lie on the model surface. The status of this process is displayed on the terminal window.

The faces and edges that remain after all checks have been completed form the surface mesh. There may be additional faces left inside the model depending on the presence of internal features with different meshes on opposing sides; such is often the case where

contact surfaces or multi-point constraints are created. These internal features currently cause problems and must be removed as discussed later.

Note that the ordering of the nodes on the element surfaces is counter-clockwise when looking at the face from outside of the model. This produces an outward pointing surface normal in OSM, which is a requirement for FRANC3D. Outward pointing normals are also a requirement in OSM for the subsequent steps of boundary surface extraction.

The nodal numbering for the eight-noded brick element faces is:

{1, 2, 3, 4}, {5, 6, 2, 1}, {3, 2, 6, 7}, {4, 3, 7, 8}, {7, 6, 5, 8}, {8, 5, 1, 4}

The nodal numbering for the six-noded wedge/prism element faces is:

{1, 2, 3}, {5, 4, 6}, {2, 5, 6, 3}, {5, 2, 1, 4}, {4, 1, 3, 6}

The nodal numbering for the four-noded tetrahedral element faces is:

{1, 2, 4}, {2, 3, 4}, {3, 1, 4}, {1, 3, 2}

The nodal numbering for the five-noded pyramidal element faces is:

{1, 2, 3, 4}, {1, 5, 2}, {1, 4, 5}, {2, 5, 3}, {3, 5, 4}

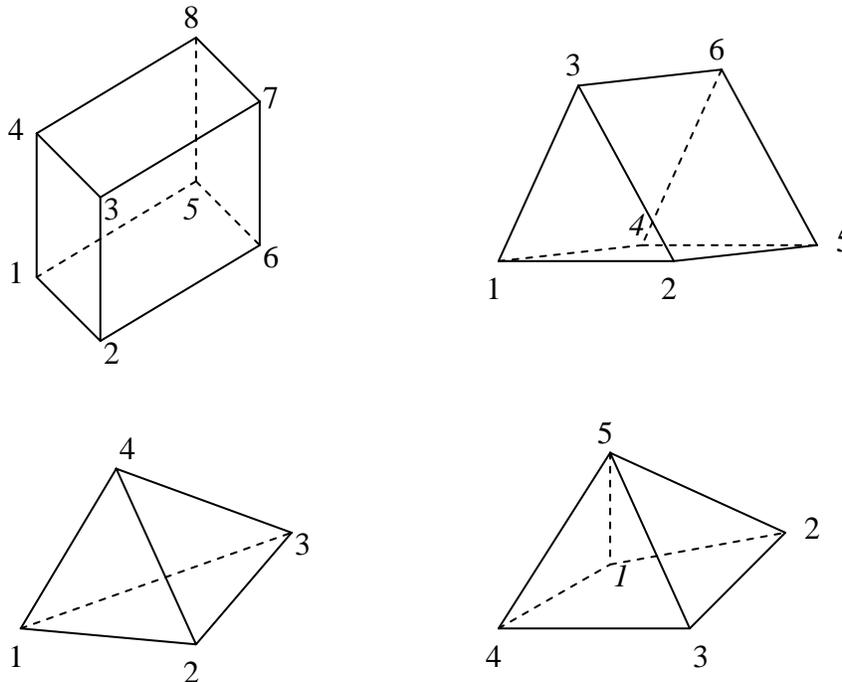


Figure 6.1. The nodal numbering in the .conn file for linear elements.

Note that the twenty-noded brick element is reduced to an eight-noded brick, the fifteen-noded wedge element is reduced to a six wedge prism, the ten noded tetrahedron is

reduced to a four-noded tetrahedron, and the thirteen-noded pyramid is reduced to a five-noded pyramid to eliminate the mid-side node only for the purpose of the OSM geometry extraction.

This process leaves us with many faces and edges on the model surface. A closed solid should exist, however, except for the possibility of internal or extra floating faces. The surface mesh model is automatically written to an OSM .mod file and then displayed in an OSM modeling window. Note that vertices are created for all the nodes on the surface.

The next step is to reduce the large number of mesh faces into a small number of faces to produce a reasonable approximation of the actual underlying geometry. A boundary surface representation composed of three- or four-sided surfaces is required by FRANC3D.

6.3 Extracting the Boundary Surface Representation

6.3.1 Creating the Radial-Edge Database

The first step in extracting the underlying boundary representation is to create a radial-edge database. This is the same database that is used in FRANC3D. It is an efficient data structure for querying and traversing the model entities, providing topological adjacency information that is extremely useful in trying to extract the geometry from the FEM surface data. The top menu button in Figure 3.21 is used to create the database.

The radial-edge database is created by adding the FEM surface faces one at a time to the database. The database is able to tie adjacent faces together and makes the corresponding face, edge, and node entities. Each of the topological entities also points back to its equivalent OSM geometric object. The code is able to detect when a region is completely closed and whether floating or non-manifold faces (usually internal) exist. Non-manifold conditions occur when an edge has more than two faces adjacent to it (Figure 6.2). The database also detects the presence of faces with a reverse normal, i.e., a surface normal that points into the model interior.

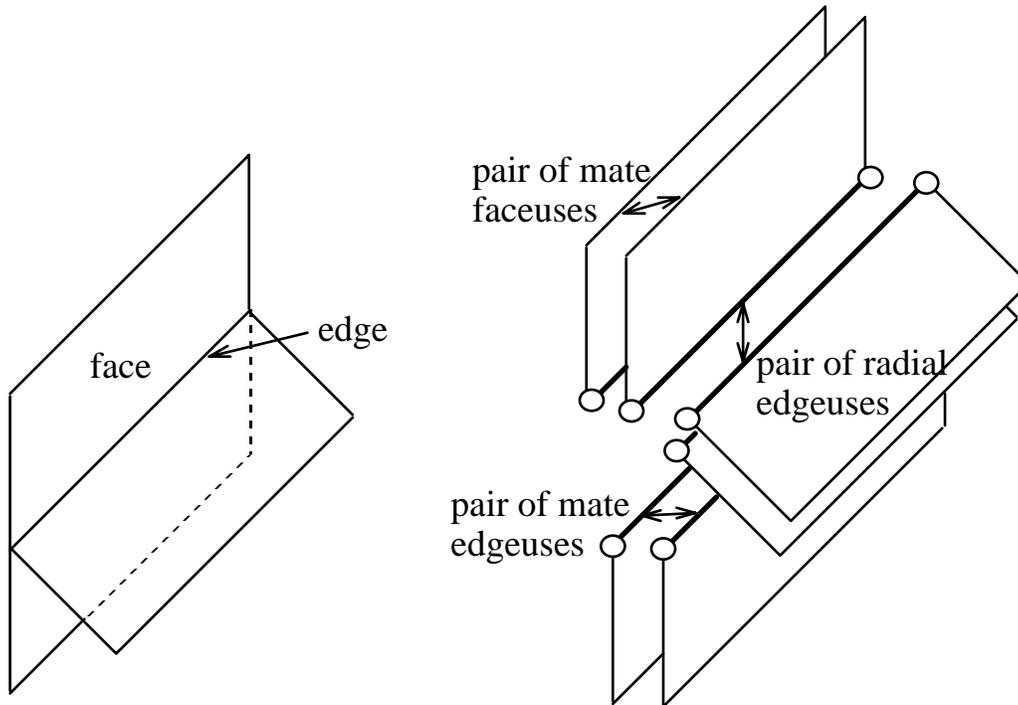


Figure 6.2. Non-manifold topology; three faces adjacent to one edge.

A closed manifold model with all surface normals pointing outward from the model is essential for the subsequent steps of boundary surface extraction. If a closed manifold model does not exist, a message dialog box appears on the screen informing the user that there is a problem. The user must modify the model before trying to create the radial-edge database again. The menu buttons **HilighGeometryPoints/Edges** and **ExtractGeometryFeatures** both remain inactive.

There are three common problems encountered in creating the radial-edge database: 1) missing faces, which leave holes in the model, 2) faces with reverse normals, and 3) non-manifold faces. The presence of the latter two conditions immediately causes the routine that adds the faces to the database to stop. In the case of non-manifold faces, the last two faces that were added to the database are highlighted. In addition, a search of the database is made for all edges that have more than two adjacent faces; these edges are also highlighted. In the case of reverse surface normals, only the last two faces that were added to the database are highlighted. A non-closed model can only be detected once all the faces have been added to the database. For the case of missing faces, all edges in the database that are used only once are highlighted.

In some cases, the highlighted entities may not pinpoint the exact problem. For example, in the case of internal free-floating faces, a non-manifold situation arises, but it might not be detected immediately. If under close scrutiny, the highlighted features appear to be okay, then search for floating internal faces.

After fixing the problems in the model, select the **CreateRadial-EdgeDatabase** menu button once again. If there still is a problem, repeat the procedure. If the model is okay, no message dialog box appears, and the menu buttons **HilightGeometryPoints/Edges** and **ExtractGeometryFeatures** both become active. You should save the model at this stage under a different name, especially if you have removed, added, or modified faces, in case you need to return to this point later; the following steps in the procedure greatly affect this model.

6.3.2 Defining the Key Geometry Features

The next step is to define the key geometry features in the model using the radial-edge database for topological adjacency information. This step consists of finding and highlighting the key geometry points and edges in the model. These features will be used to define the corners and bounding edges of boundary surface patches that encompass many FEM faces.

The process of finding the key features starts by selecting the **HilightGeometryPoints/Edges** menu button. A dialog box is displayed for entering the geometric angular tolerance. This tolerance is used to decide whether a node (topological point) lies on a geometric surface boundary. The faces adjacent to the node are found, and the local surface normals are computed. If the angle between any two local surface normals differs by more than the geometric angular tolerance, the node is stored as a key geometry node (Figure 6.3). If no key nodes are found all nodes are retained as key nodes; for example, on a suitably discretized sphere there is no abrupt change in the geometry surface normals, so all vertices are key vertices.

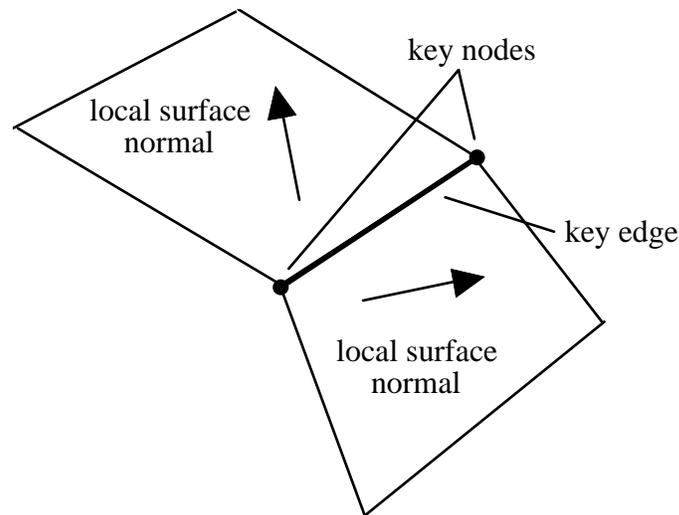


Figure 6.3. The angle between local surface normals is used to define key geometry points.

Once all the key nodes are found, the key edges and the end-key nodes are defined. A key edge is an edge that has a key node at both ends. An end-key node is a key node at the end of a chain of key nodes and key edges. A chain of key edges and key nodes is terminated by abrupt changes in geometry or adjacent edge direction. The end-key nodes define corners of the underlying boundary surface geometry and the chain of key edges and nodes between two end-key nodes defines a bounding edge of the underlying surface geometry. These relationships for a simple model are illustrated in Figure 6.4.

The next step is to form chains of bounding edges. A chain is created by starting at one end-key node and traversing the topologically adjacent key edges and nodes until another end-key node is encountered, (Figure 6.5). A chain terminates at an end-key node or if the next adjacent key edge changes direction by more than the angular tolerance.

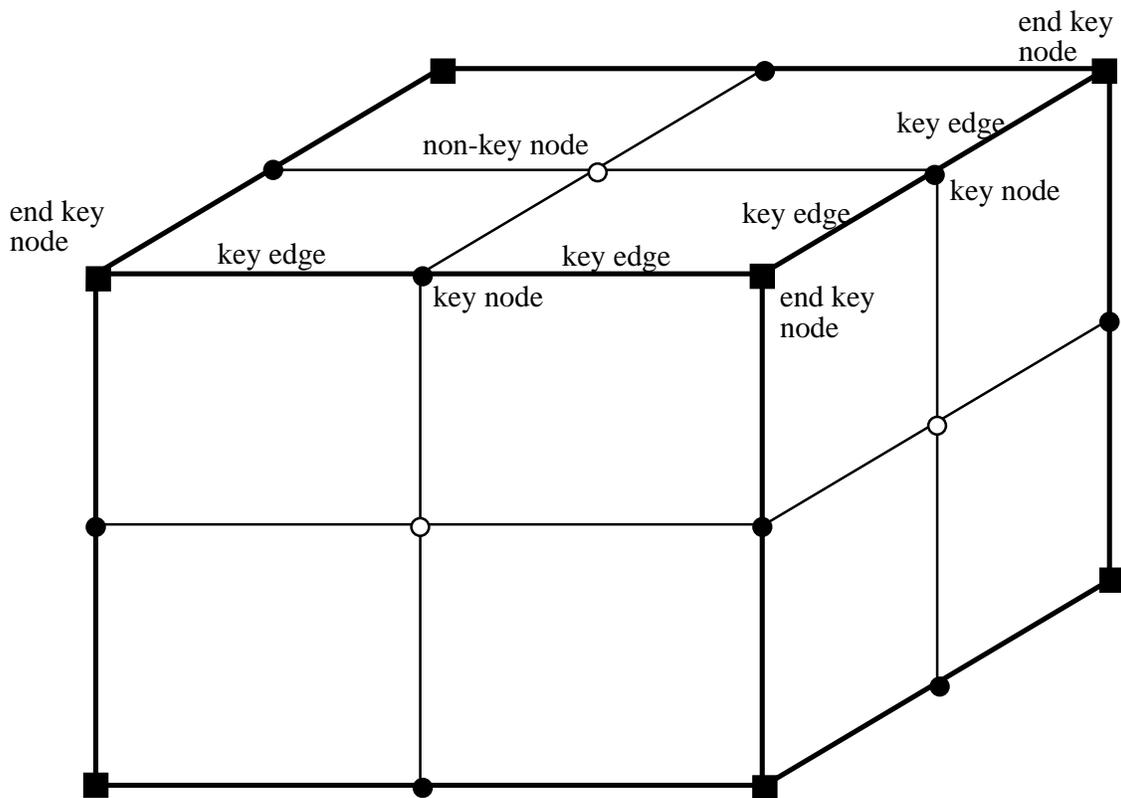


Figure 6.4. Key points, edges and end points are defined based on differences in local surface normal directions. End-key points terminate a chain of key points and edges.

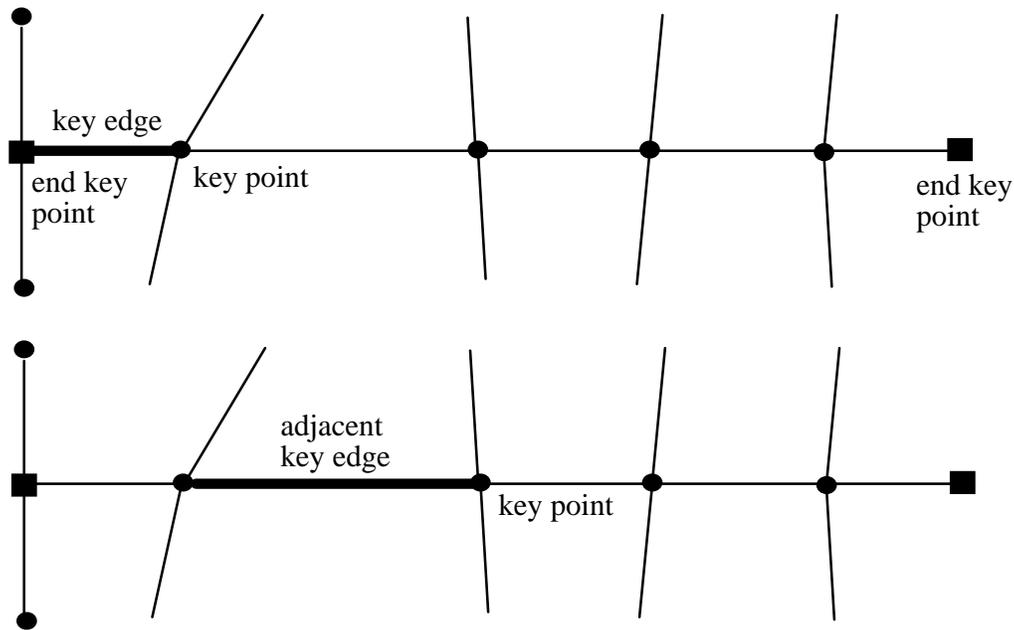


Figure 6.5. Traversing the key edges to create the bounding edge chains.

This completes the step of defining the key geometry features. Note that after the entities have been highlighted, a user can copy and paste these into another OSM modeling window and examine the features that potentially will be extracted.

6.3.3 Extracting the Geometry

The final step is to extract the features and form surface patches from the chains of bounding edges. This step starts by selecting the **ExtractGeometryFeatures** menu button. This is done by forming sets of closed bounding edge chains into closed bounding edge loops. While this is conceptually simple, computationally it is very difficult. An exhaustive search is used to find all the possible valid loops (a valid loop is closed circuit that does not cross itself except at the start/end node).

The following heuristics are used to aid the search for valid loops:

- Each key bounding edge can be used only twice (requires a manifold model).
- The geometrically shortest, closed, valid loop that is found for any given starting edge is retained.
- When going from one chain of bounding edges to another (i.e., at a end-key node), the individual edge segments of the bounding edge chain are examined. By traversing the edges that are adjacent to the same FEM face at the end-key node (Figure 6.6), loops are found more quickly.

As each valid loop is formed, it is stored. Each new valid loop is compared to existing saved loops to make sure there are no duplicates. The number of unique valid loops is printed to the terminal window before trying to form faces. For each loop, the number of chains of bounding edges is checked. If the number is greater than four, the loop cannot

be used to form a valid surface patch in OSM. Remember that OSM and FRANC3D require surface patches with three or four sides. Any loop with more than four chains will be left for the user to break into appropriate smaller loops and surfaces.

Each chain consists of one or more small edge segments. These small segments are merged to form a single curve. Merging is done by taking the end points of all constituent key edges and fitting a smooth b-spline curve through the points. When finished, the three or four chains become three or four bounding curves.

If the loop has three or four bounding curves, a valid face can be formed in OSM. The curves are passed to the OSM **CreatePatchByCurves** routine which blends the curves to form a bi-cubic b-spline or Bezier surface patch. All bounding curves and faces are stored so that they can be transferred later to a new OSM modeling window.

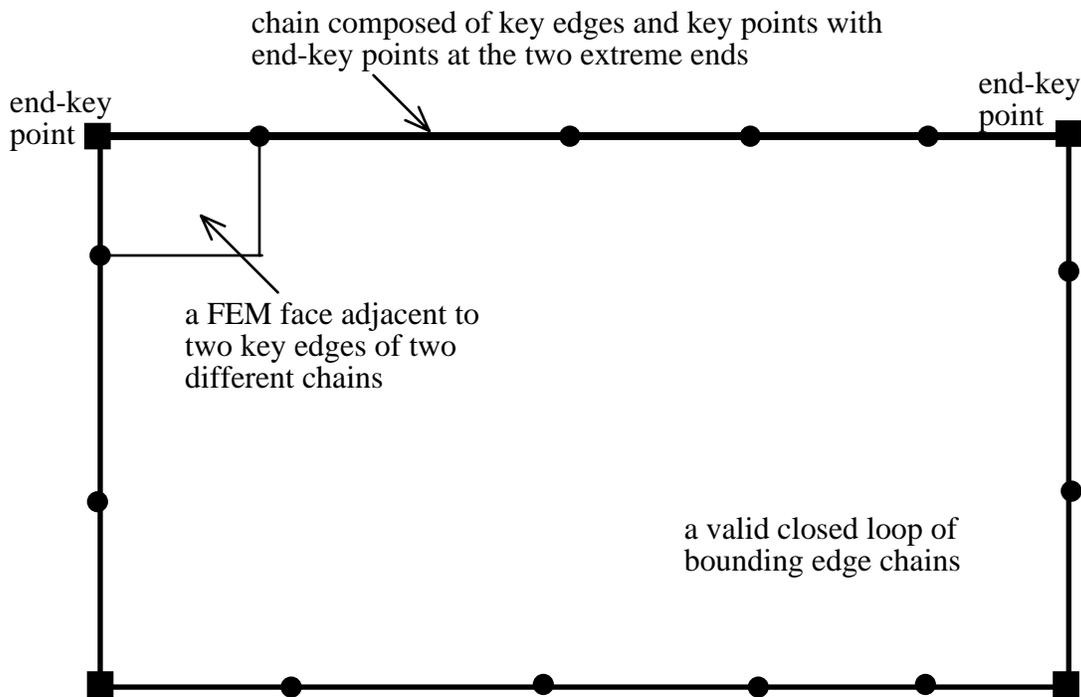


Figure 6.6. A loop of bounding edges.

For all cases where the three or four sided faces could not be created automatically, you must create faces manually using the regular OSM tools as described in the OSM documentation. The example problem described in the next section highlights many of the problems that are normally encountered in extracting the boundary surface geometry. Note that the **CreateRadial-EdgeDatabase** menu button can be used to determine if the new model forms a valid closed manifold model that FRANC3D will be capable of reading. This button actually can be used with any model.

The extracted model should be saved as an OSM .mod file under a different name. Note that the original model consists of both the original and extracted features, so one should not re-save this model.

6.4 Compressor Disk Example

The example is a piece of a compressor disk that was created using ANSYS. The model geometry was created and then meshed with the automatic tetrahedral meshing algorithm. An ANSYS .cdb file was written. The example has about 20000 tetrahedral elements. The steps to convert the FEM data to an OSM model are described below.

Step 1: Select the **Translate ANSYS cdb** menu button. Select the file name (disk.cdb) from the dialog box by double clicking (or single click followed by **Accept**). The process of translating the ANSYS format to the .conn format and extracting only the surface faces and edges begins automatically. The current status (number of faces processed) is printed to the terminal window. When finished, the files "disk.conn" and "disk.conn.mod" are saved automatically and the model is displayed in a new modeling window (Figure 6.7).

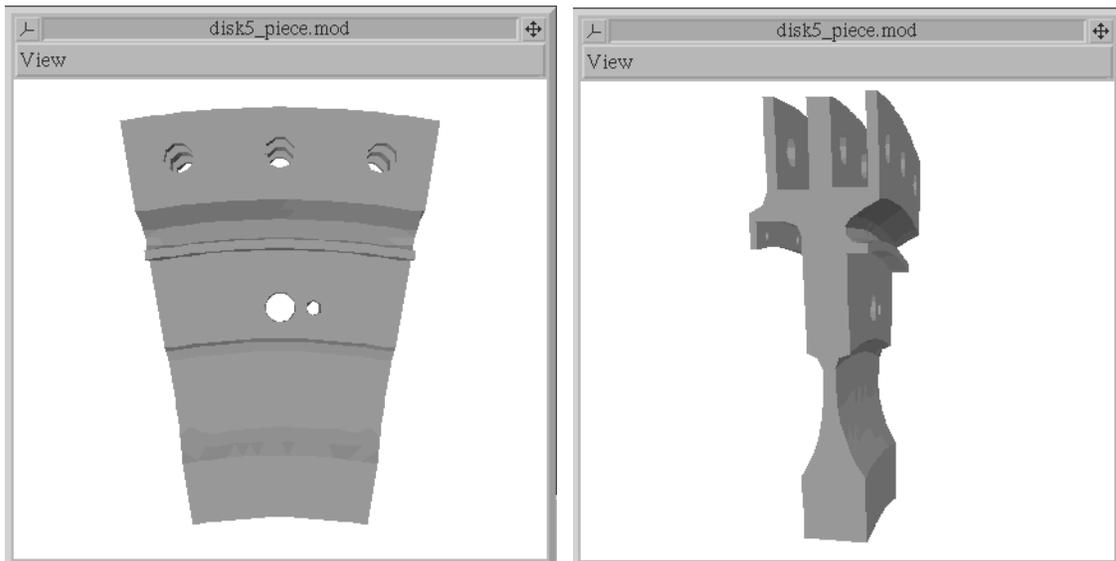
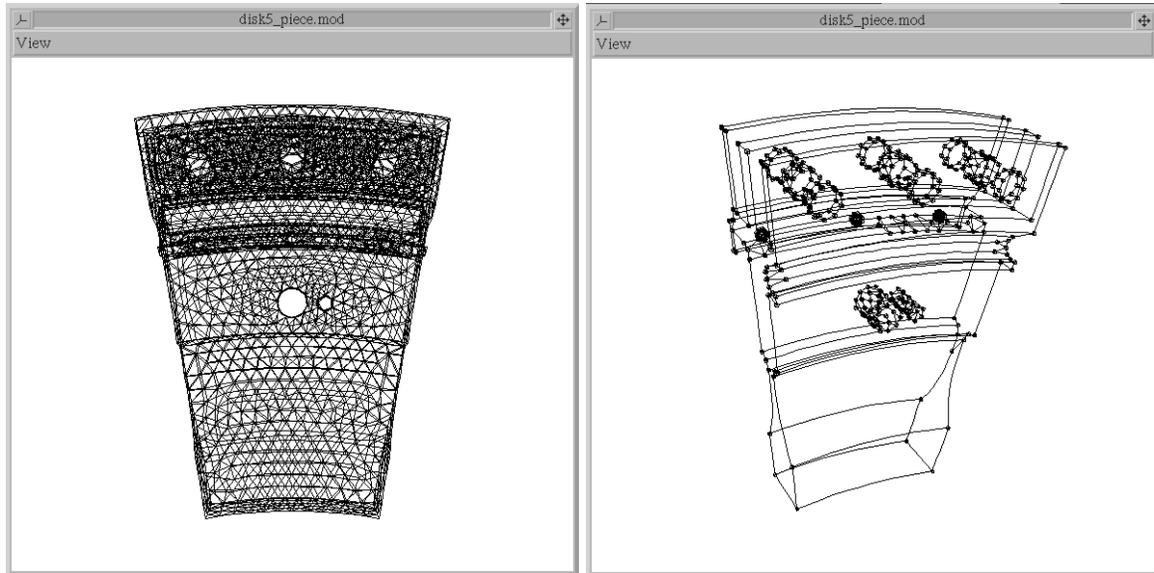


Figure 6.7. Two views of the initial OSM model of the portion of the NASTRAN disk; the solid model is shown with points and curves turned off.

Step 2: Select the **CreateRadial-EdgeDatabase** menu button. The process of adding the many mesh surface faces to the radial-edge database starts. The process finishes without displaying any messages, and the **HighlightGeometryPoints/Edges** menu command becomes active. If a valid radial-edge database had not been created, a message dialog box would have been displayed, problem areas of the model would have

been highlighted, and the **HighlightGeometryPoints/Edges** menu button would have remained inactive.

Step 3: Select the **HighlightGeometryPoints/Edges** menu button. In the geometric angular tolerance dialog box, make sure the number is 10.0 and select Accept. The process of finding the end key points and key bounding edges begins. When finished, the entities are highlighted in the OSM modeling window (Figures 6.8).



Figures 6.8. Original model edges and the highlighted end-key points and key bounding edges in the model.

Step 4: Select the **ExtractGeometryFeatures** menu button. The process of creating loops and faces from the key bounding edges starts. When finished the new entities are copied and pasted into a new OSM modeling window. Note that these same entities exist in the original model as well. The procedure manages to create a portion of the faces, but some loops contained more than four edges (Figures 6.9). The model needs to be finished manually using the regular OSM modeling tools.

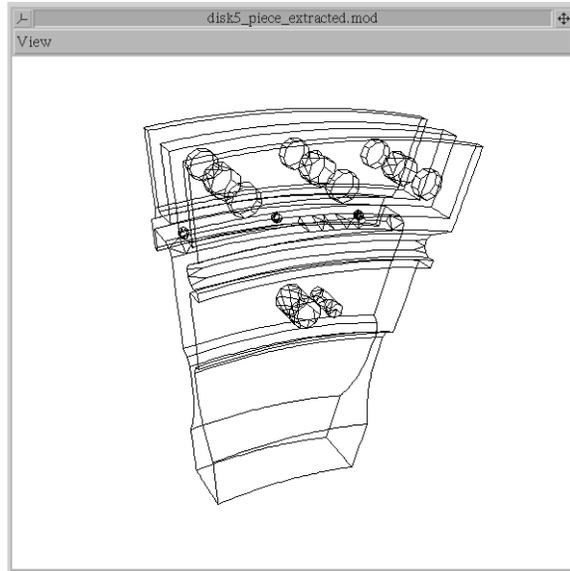


Figure 6.9. Extracted boundary representation of the model.

Step 5: Finish the model manually by merging edges and making new faces. Add curves where the curves are missing, merge curves together to form larger curves to reduce the number of surfaces, create surfaces to fill in the surface of the model. Note that OSM and FRANC3D require that surfaces be composed of three or four bounding edges. Also surfaces must be geometrically and topologically contiguous. In other words, adjacent faces must use the same edge to define the surface geometry.

For instance, the model shown in Figure 6.9 contains most of the key geometric points and edges for the boundary of the model. If some points or edges are missing, they can be copied from the original model and pasted into this model. It is best to position the extracted model, save the camera position, and then go back to the original model and reset the view based on the camera position that was just saved. Points that are needed to create the missing curves can be copied and pasted. By selecting sets of points and picking the **Create Curve** menu button, larger curves can be formed. For instance, the set of edges and points shown in Figure 6.10 were not merged automatically as they did not form part of a loop. These can be merged to form a single curve that will eliminate a large number of edges and points, and consequently surfaces.

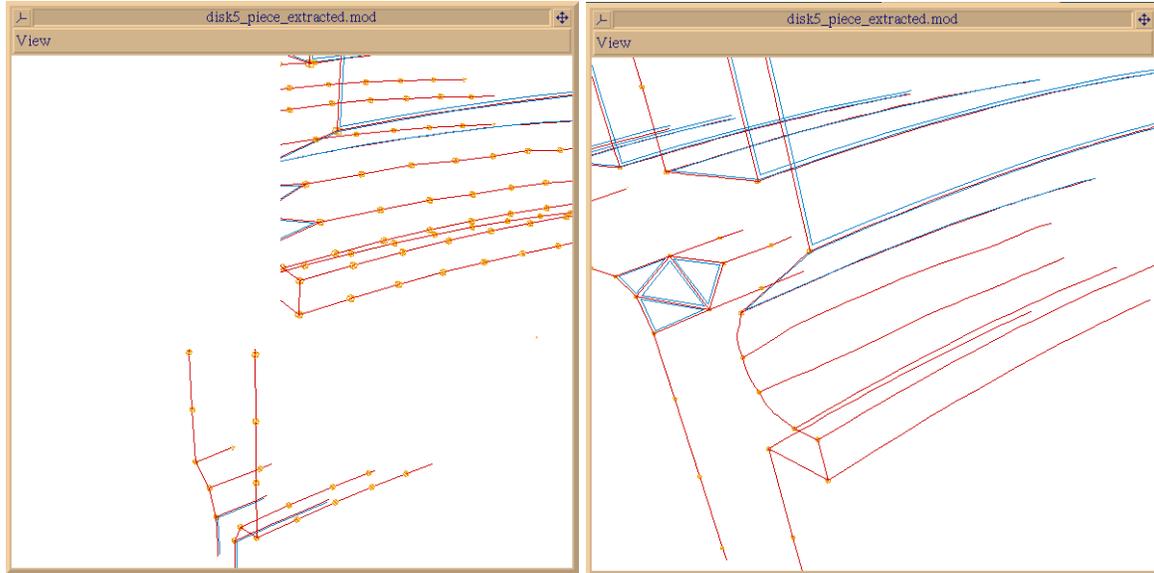


Figure 6.10. Edge segments can be merged to form a single curve.

Some surfaces of the model were not formed, the symmetry surfaces for example. This is because these surfaces are planar, but have a very irregular boundary made up of many edges. It is fairly easy to add straight-line edges between some end points to make faces on these surfaces. However, you must remember that faces can only have three or four bounding edges, so you need to break the surface up accordingly. For instance, the portion of the model shown in Figure 6.11 requires several faces to fill in the surface adequately.

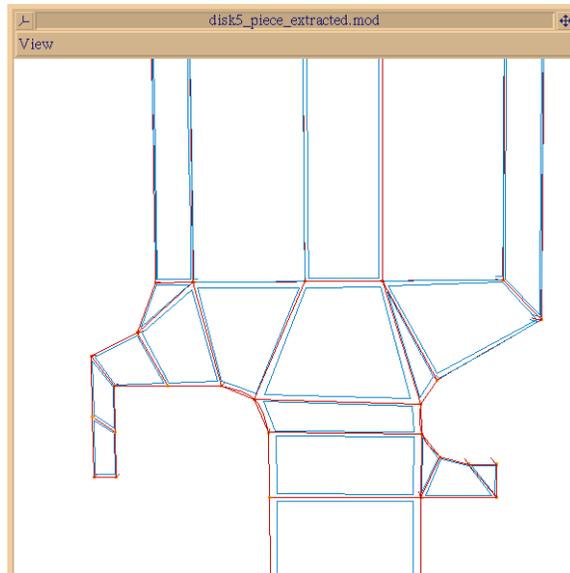


Figure 6.11. Add surfaces where needed to the model to make a closed solid.

Once the model has been completed, it can be saved both as an OSM restart file and as a FRANC3D geometry input file. However, before writing the FRANC3D file, you can ensure that the model will be read by FRANC3D by creating a radial edge database for this model in OSM. Select the **CreateRadial-EdgeDatabase** menu button. The process of adding the surface patches to the radial-edge database starts. If the process finishes without displaying any messages, then a valid radial-edge database had not been created and the model is ready for FRANC3D. If there are problems with the model such as reversed surface normals or missing surface patches, then a message is displayed and the problem areas highlighted. Save the FRANC3D restart file by selecting **Write FRANC3D File** and then **3D Solid ...** and enter the desired file name.