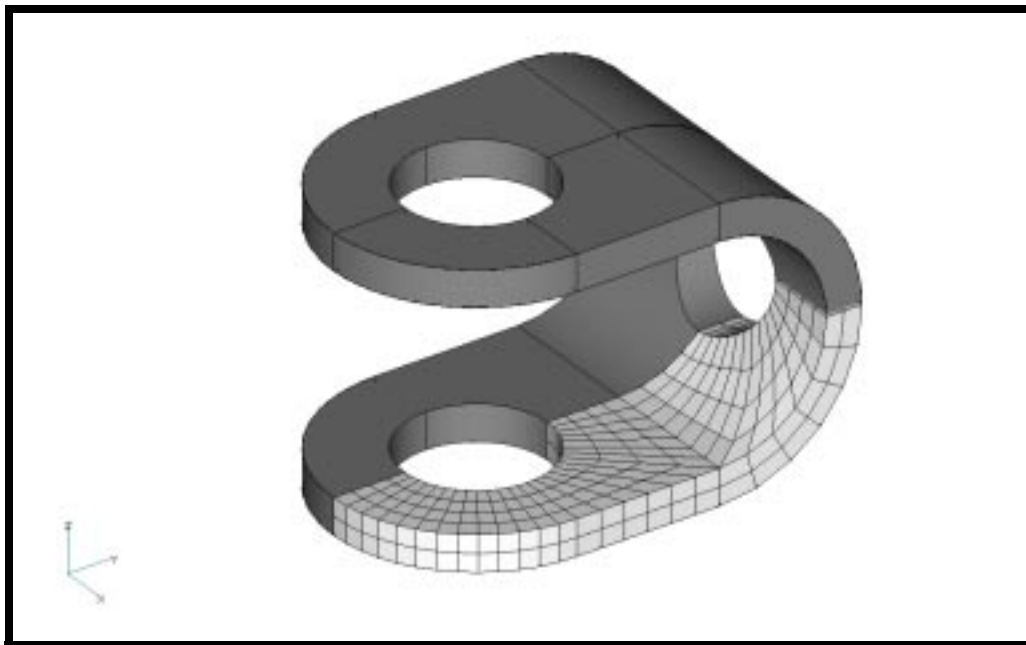

APPENDIX C

HexMesh a Solid



Objectives:

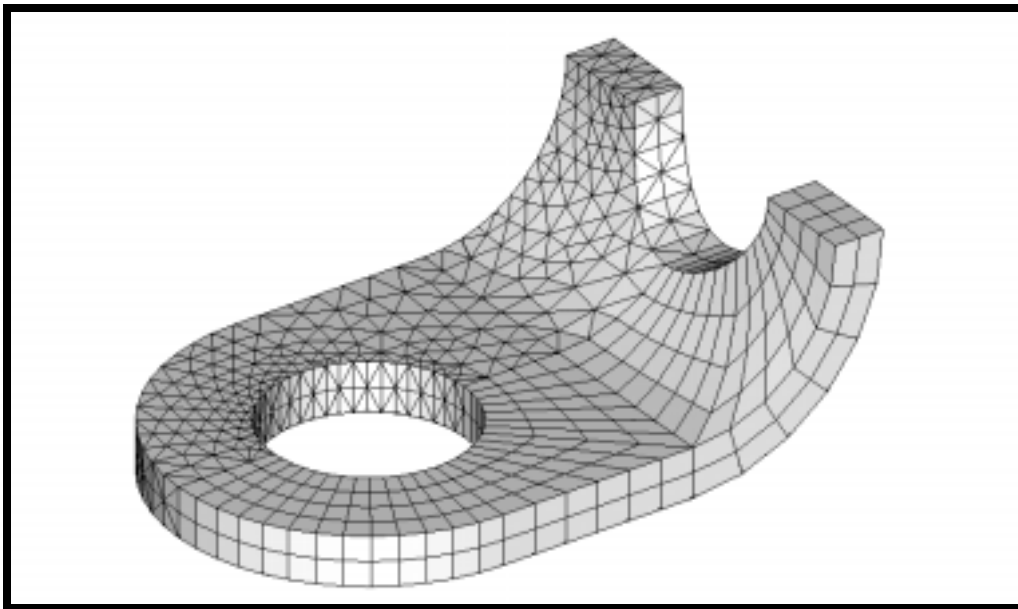
- Import a parasolid geometry file.
- Subdivide the part for hex-meshing.
- Parameterize the surfaces by specifying vertices for meshing.
- Merge coincident nodes.

Model Description:

Although it is possible to get fast reasonable results with a tetrahedral mesh, more accurate results can be obtained using a hex mesh. A hex mesh is also needed when modeling 3D-orthotropic materials such as circuit boards for the thermal analysis, fiberglass in a structural analysis, or any structure whose geometry cannot easily be mapped with a coordinate system.

In this workshop, we are going to demonstrate how the supplied solid geometry can be hexmeshed. Figure C.1 shows a quarter of the model temeshed, and we are going to build the other quarter of the model with hex elements.

Figure C.1 - Tet elements and hex elements side by side



Exercise Procedure:

1. Start up MSC.Nastran for Windows v4.0 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows v4.0.

On the *Open Model File* form, select **New Model**.

Open Model File:

New Model

2. Use the *Import Geometry icon* and select **hex.x_t** from the training directory and click the *Open* button.

From the pulldown menu, select **File/Import/Geometry**.

File/Import/Geometry...

File name:

hex.x_t

Open

In the *Solid Model Read Options* window select **OK** to accept default values.

OK

3. Rotate the model to get a better view.

Choose *View/Rotate* and select a *Dimetric* view.

View/Rotate...<F8>

Dimetric

OK

4. Turn off the workplane.

Choose *Tools/Workplane* and uncheck the *Draw Workplane* box.

Tools/Workplane...

Draw Workplane

Done

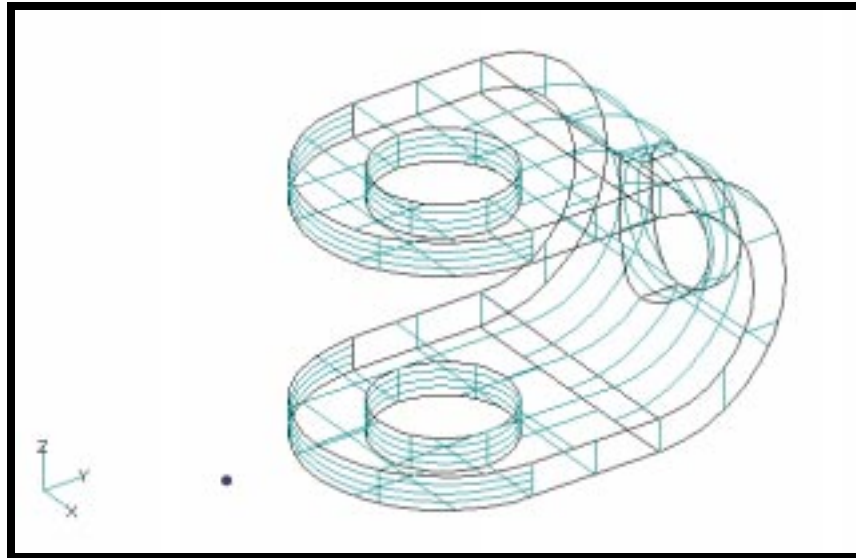
5. Choose *View/Redraw* to redraw the graphics.

View/Redraw

<Ctrl+D>

Your display should look like the figure below.

Figure C.2 - Imported solid geometry



6. Define material properties using the Library.

MSC.Nastran for Windows 4 has a material library containing commonly used materials. These materials are available only as a convenience and should be verified before use.

From the pulldown menu, select **Model/Material**.

Model/Material...

Load...

Library Entry:

AISI 4340 Steel

OK

OK

Cancel

7. Define a solid property.

From the pulldown menu, select **Model/Property**.

Model/Property...

Title:

prop_1

Assign the previously created material; click on the list icon next to the databox and select **AISI 4340 Steel**.

Material:

1..AISI 4340 Steel

Change the element *Property Type* to solid volume.

Elem/Property Type...

Under the *Element / Property Type* window, make the following selection:

Volume Elements:

Solid

OK

OK

Cancel

8. Rotate the model to get a better view.

Choose *View/Rotate* and select a *Isometric* view.

View/Rotate...<F8>

Isometric

OK

9. Subdivide the solid along planes of symmetry. Only 1/4 of this model will be meshed for demonstration purposes.

Slice the geometry symmetrically in the YZ plane.

Geometry/Solid/Slice...

Select All

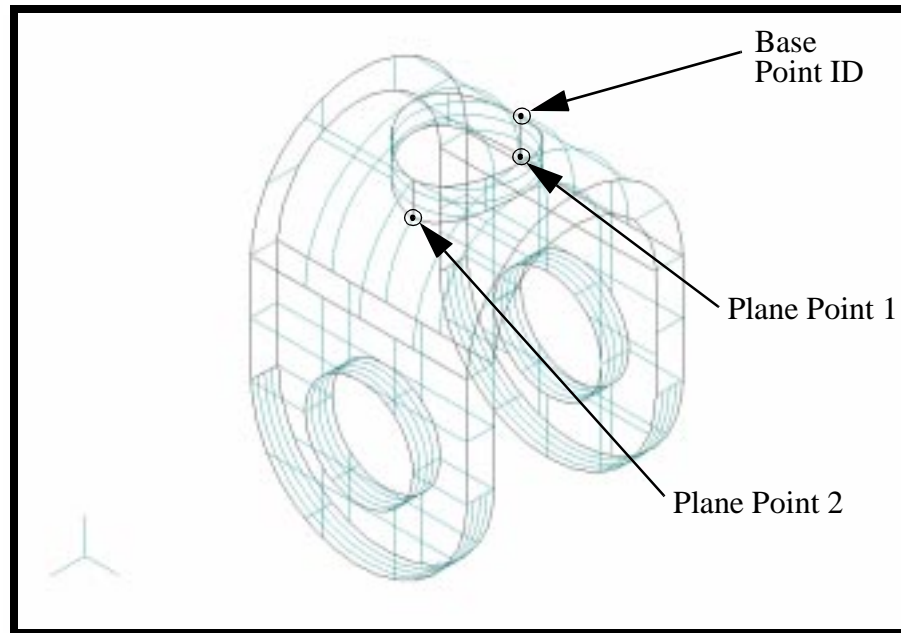
OK

Use points to define a cutting plane.

Methods^

Points

Figure C.3 - Selected points to define cutting plane parallel to the YZ plane.



Select three points for the cutting plane. (MSC.Nastran for Windows allows users to enter these points into the menu by picking them on the screen)

Base Point ID: (see Figure C.3 for location)

Plane Point 1: (see Figure C.3 for location)

Plane Point 2: (see Figure C.3 for location)

To visually inspect the cutting plane choose *Preview* from the menu box, then click *OK* when ready.

Preview

OK

10. Rotate the model to back to a *Dimetric* view.

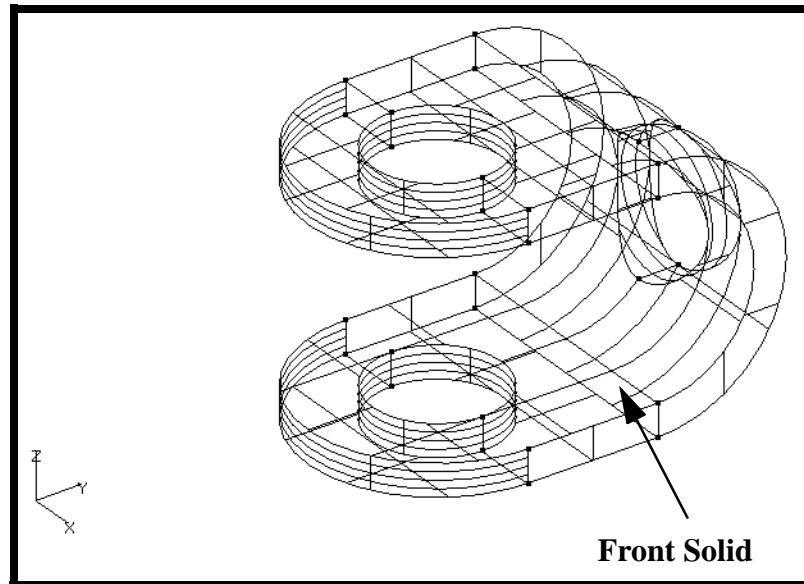
Choose *View/Rotate* and select a *Dimetric* view.

View/Rotate...<F8>

Dimetric

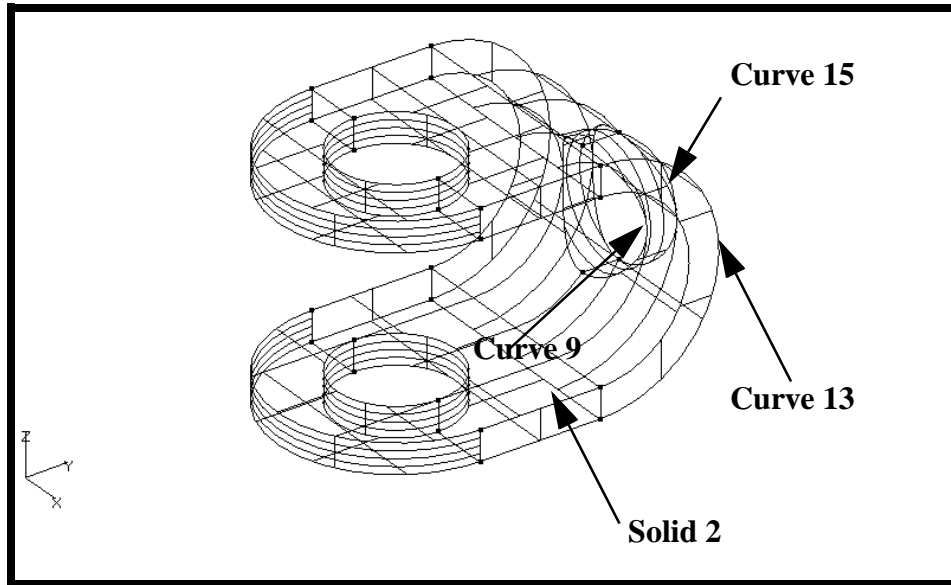
OK

Figure C.4 - Cutting the geometry along symmetry planes



11. The solid is also symmetric about XY. To exploit this, 3 points will need to be defined at the midpoint of curves on the solid.

Figure C.5 - Curves where the midpoint will be extracted for a cutting plane.



Create three points to define the cutting plane by selecting the midpoints of three curves:

Geometry/Point...

Methods ^

Midpoint

Curve ID:

9

OK

Curve ID:

15

OK

Curve ID:

13

OK

Cancel

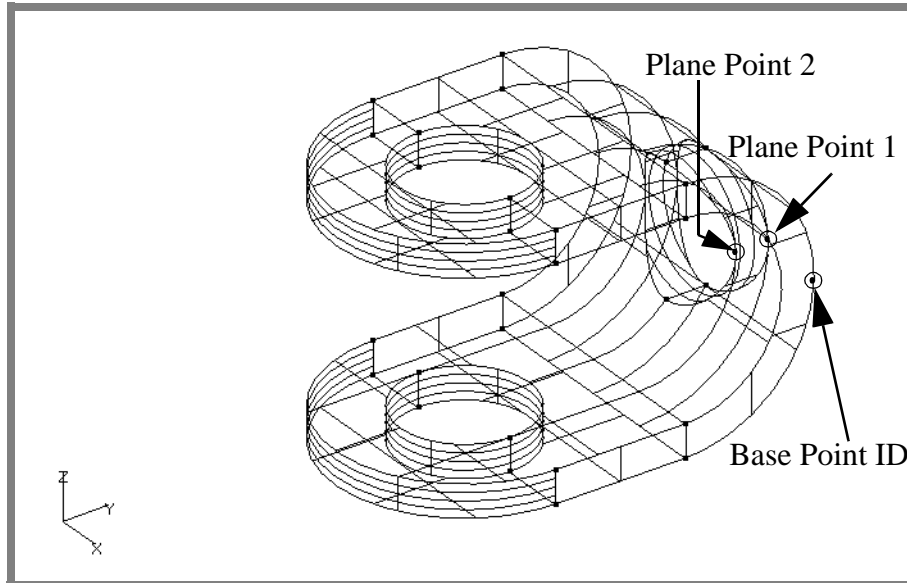
Slice the geometry in the XY plane.

Geometry/Solid/Slice...

Select Front Solid
(see figure C.4)

OK

Figure C.6 - Points to define a cutting plane in the XY plane.



Use the 3 midpoints to define a cutting plane.

Base Point ID: (see Figure C.6 for location)

Plane Point 1: (see Figure C.6 for location)

Plane Point 2: (see Figure C.6 for location)

OK

- To more easily facilitate the meshing, we will only mesh this quarter of the model. Place this solid and its associated surfaces, curves, and points into a group.

Group/Set...

Title:

quarter_model

OK

Group/Solid/ID...

View/Select

<F5>

Model Style: **Draw Model****Model Data***Group:* **Select***(from drop down menu)***Group/Surface/on Solid...****Group/Curve/on Surface...****Group/Point/on Curve...****Group/Operations/Automatic Add...** **Select**

13. The quarter model displayed in Figure C.7 is the part we want to mesh. A new hexmeshing tool has been added to MSC.Nastran for Windows V4.0, but to use this tool, we have to further subdivide this solid.

It may not be apparent how this geometry has to be subdivided, and there are different, viable options. This exercise will demonstrate one option that will generate hex elements on this geometry.

Start by subdividing the geometry into sections that can utilize the map mesher. Follow the exercise, and you will start to become familiar with geometry that can utilize the map mesher..

Geometry/Solid/Slice...

Select Solid

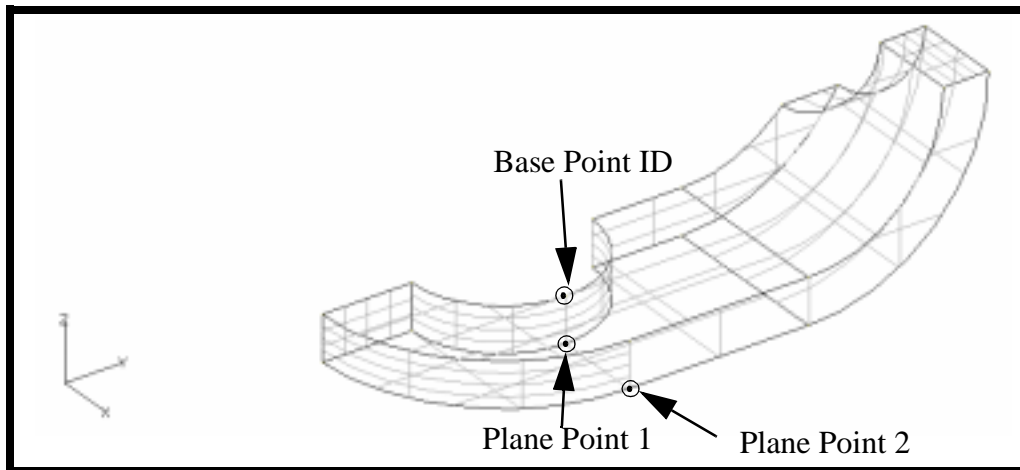
OK

Make sure the methods box is still set to points.

Methods ^

Points

Figure C.7 - Selected points to define cutting plane for first meshing solid.



Use three points for the cutting plane.

Base Point ID: (refer to figure C.7)

Plane Point 1: (refer to figure C.7)

Plane Point 2: (refer to figure C.7)

OK

14. The solid will need one more cut for mapped meshing. The points for the cutting plane are shown.

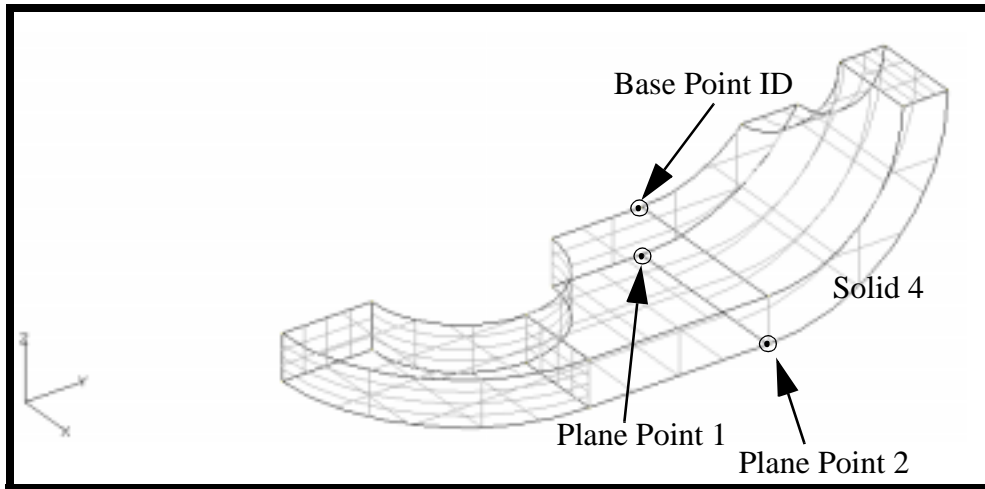
Geometry/Solid/Slice...

ID:

4

OK

Figure C.8 - Selected points to define the final cutting plane for meshing solids.



Use 3 points for the cutting plane.

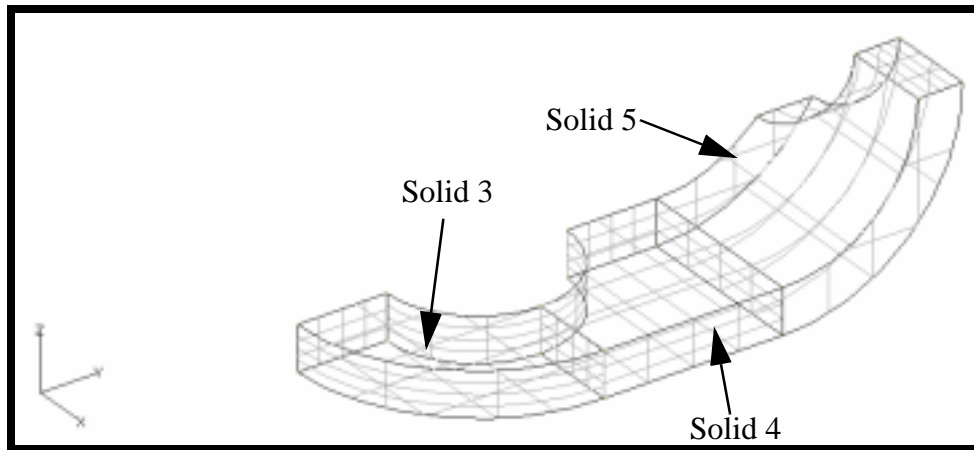
Base Point ID: (refer to figure C.8)

Plane Point 1: (refer to figure C.8)

Plane Point 2: (refer to figure C.8)

OK

Figure C.9 - Geometry cut into 3 parts.



15. Apply mesh control to these 3 solids.

Mesh/Mesh Control/Size On Solid...

Select solids 3, 4, 5

OK

Size For:

Hex Meshing

Element Size:

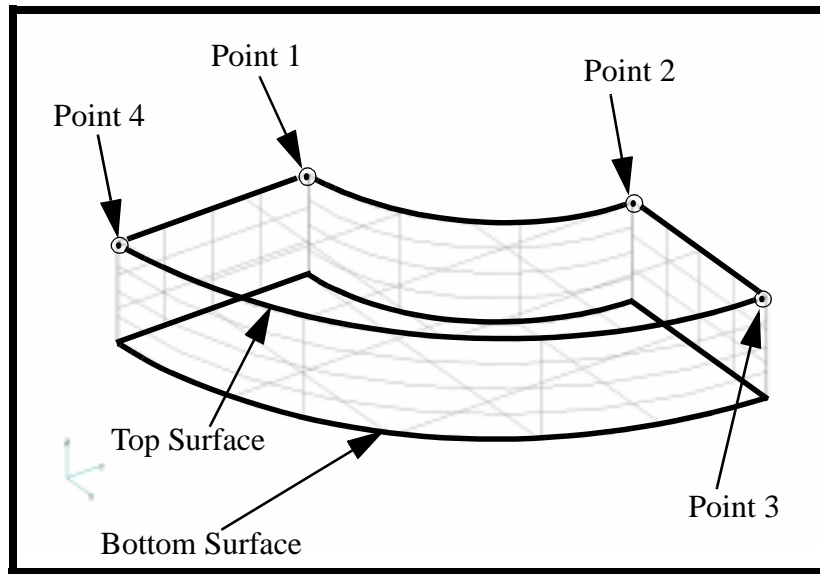
0.25

OK

16. Shown in Figure C.10 is the first solid to hex mesh. This first solid has 6 faces. The top surface has 4 edges and 4 vertices. This is the most obvious type of solid that can be hex meshed.

For the hexmesher to work, we have to identify a top surface with only 4 vertices, and then link the bottom surface to this top surface. After the opposing surfaces are determined, then we have to generate mesh controls of equal numbers on the opposing edges of these surfaces.

Follow these steps, and use Figure C.10 to help you make the proper selections. It's important that you select the surfaces and vertices identified in Figure C.10

Figure C.10 - Front solid with four vertices for hex meshing.**Mesh/Mesh Control/Approach On Surface...**

Select the top surface
(See figure C.10)

OK

Mesher:

● **Mapped - Four Corner**

Point 1:

(see figure C.10 for location)

Point 2:

(see figure C.10 for location)

Point 3:

(see figure C.10 for location)

Point 4:

(see figure C.10 for location)

OK

Slave the opposite surface of the solid.

Select the bottom surface
(See figure C.10)

OK

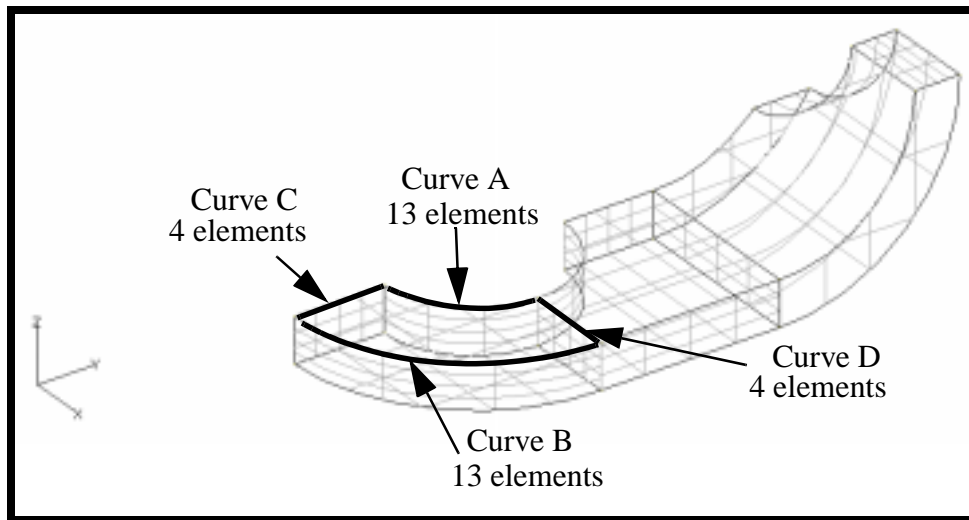
In the *Surface Mesh Approach* window input the following:

Mesher: **● Matched - Link to Surface**
Master Surface: (select top surface from figure C.10)

OK
Cancel

17. Before meshing, verify that the number of elements on opposing edges of the surfaces are the same. Use the dynamic query tool to verify the number of mesh elements on each edge. (See Figure C.11)

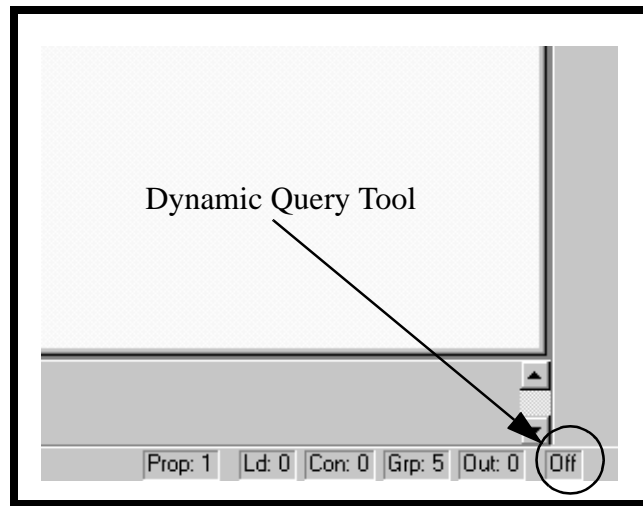
Figure C.11 - Selected edges for the first solid .



Make sure curves A and B both have 13 elements, and curve C and D both have 4 elements.

Set the dynamic query tool to *curve*.

The dynamic query tool is activated by left clicking on the item at the most lower right corner of the view port. This box is set to “Off” by default. Left clicking on this allows the user to inspect different types of entities. To check the number of elements on a curve, select “Curve” from the dynamic query box and highlight a curve. Leaving the cursor over an entity for two seconds causes a label to “pop”, which will tell the user the number of elements on that curve, as well as any other pertinent data. When finished, left click on the dynamic query box again, and set the menu back to “OFF”.

Figure C.12 - Activate the dynamic query tool

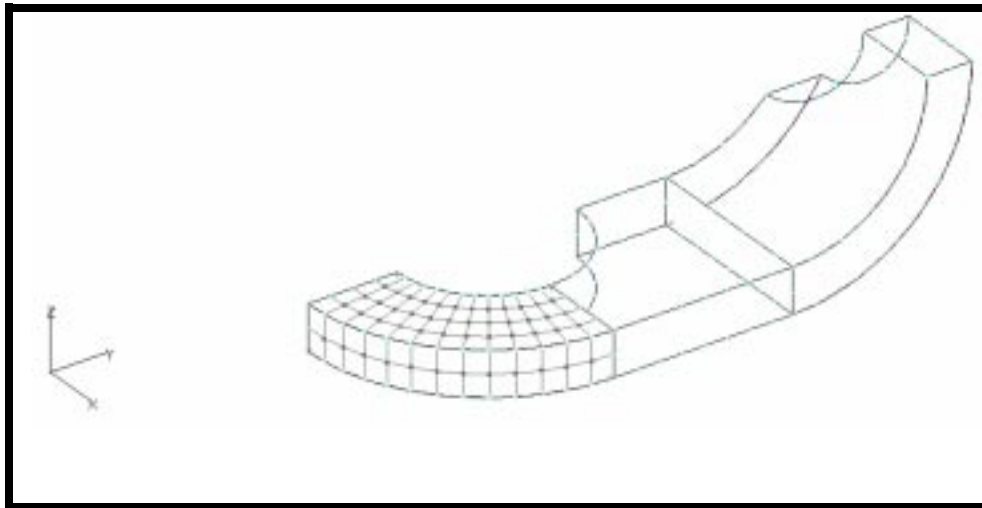
18. Once you have confirmed that the oppsing edges of the surfaces have equal numbers of mesh elements, we are ready to hex mesh the solid.

Mesh/Geometry/HexMesh Solids...*Select Solid:***3***(the front solid)***OK****Property:****prop_1****OK****View/Select...**

<F5>

Model Style:● **Quick Hidden Line** **Render****OK**

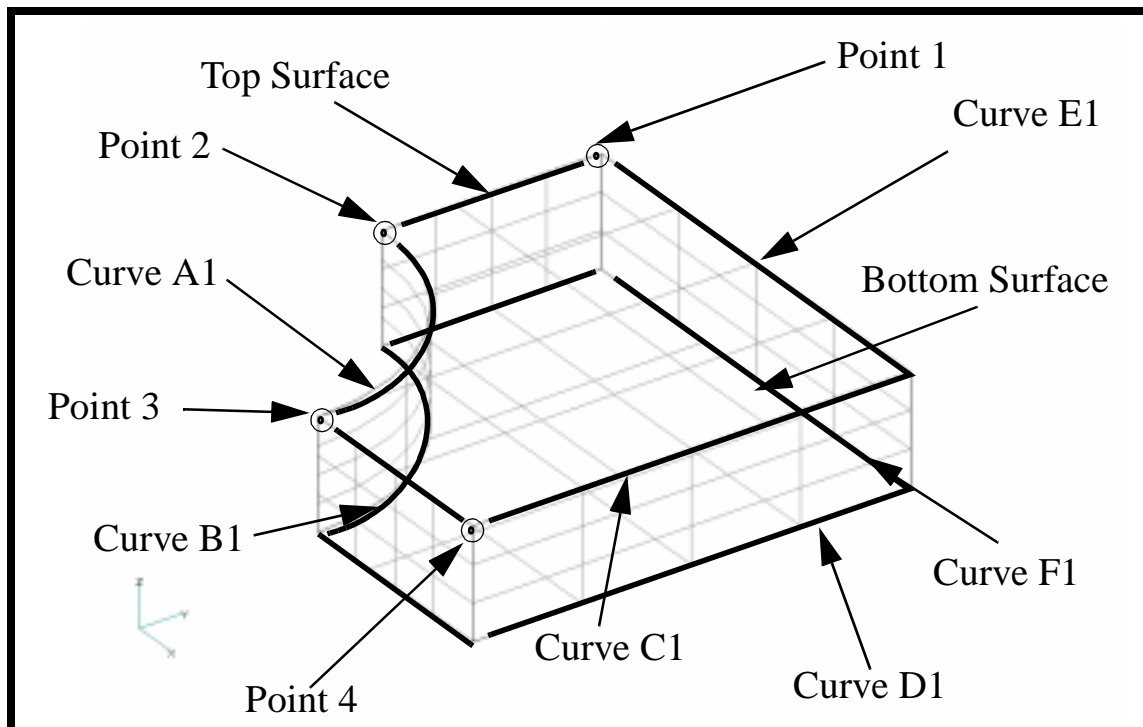
Figure C.13 - Mesh on first solid.



19. Figure C.14 shows the second solid. Notice that the top surface has 5 edges. To make the mapped mesher work, 4 vertices have to be chosen, and the total number of mesh elements between the vertices have to equal that of the opposing “edge.”

In this solid, the edge A1 will map to “edge” C1 and E1 combined, so the total number of elements on A1 has to equal the sum of the elements $C1 + E1$.

Figure C.14 - Middle solid with curves for mesh control and four vertices for hex meshing



Mesh/Mesh Control/Size Along Curve...*select A1 and B1*

OK

● Number of Elements

13

OK

select C1 and D1

OK

● Number of Elements

6

OK

select E1 and F1 twice

OK

● Number of Elements

7

OK

Cancel

NOTE: We have to select E1 and F1 twice because there are 2 curves in both positions. The coincident curves are from the different solids.

20. Now we are ready to prepare the surfaces for meshing.

Mesh/Mesh Control/Approach On Surface...*select top surface*

OK

Mesher:

● Mapped - Four Corner

*Point 1:**(see figure C.14 for location)**Point 2:**(see figure C.14 for location)**Point 3:**(see figure C.14 for location)*

Point 4:

(see figure C.14 for location)

OK

Slave the oppsite surface of the solid.

select bottom surface

OK

Mesher:

● Match - Link to Surface

Master Surface:

select top surface

OK

Cancel

21. Now the solid is ready to hex mesh.

Mesh/Geometry/HexMesh Solids...

Select Solid:

4

(the middle solid)

OK

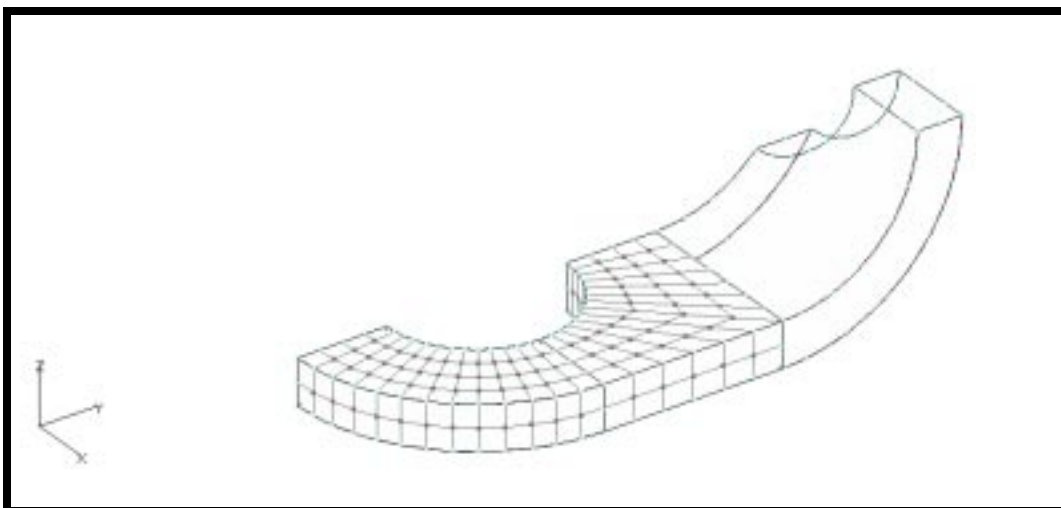
OK

22. Choose *View/Redraw* to redraw the graphics.

View/Redraw

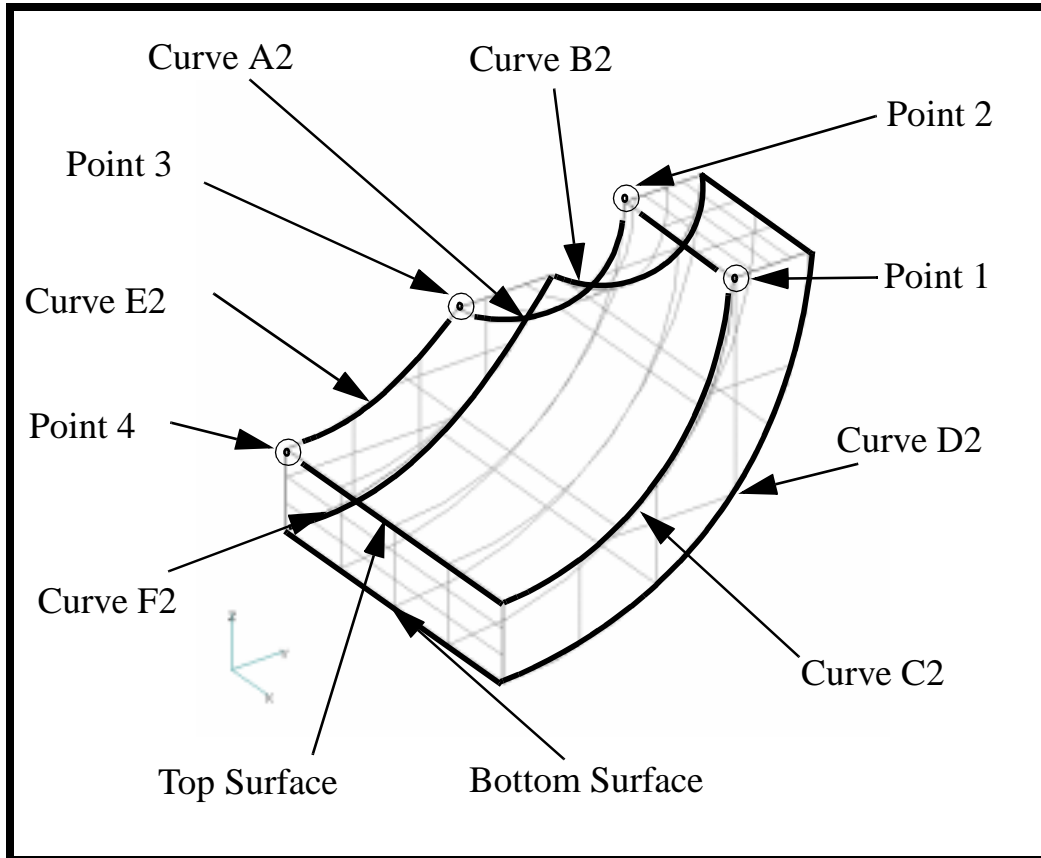
<Ctrl+D>

Figure C.15 - Mesh on middle solid



23. Now we will prepare and mesh the final solid. This solid is similar to the middle solid. Use Figure C.16 for direction in selecting vertices and surfaces.

Figure C.16 - Final solid with curves for mesh control and four vertices for hex meshing.



Mesh/Mesh Control/Size Along Curve...

select A2 + B2

OK

● **Number of Elements**

13

OK

select C2 + D2

OK

● Number of Elements

6

OK

select E2 + F2

OK

● Number of Elements

4

OK

Cancel

24. Prepare the surfaces.

Mesh/Mesh Control/Approach On Surface...

select top surface

OK

Mesher:

● **Mapped - Four Corner**

Point 1:

(see figure C.16 for location)

Point 2:

(see figure C.16 for location)

Point 3:

(see figure C.16 for location)

Point 4:

(see figure C.16 for location)

OK

Slave the oppsite surface of the solid.

select bottom surface

OK

Mesher:

● **Match - Link to Surface**

Master Surface:

select top surface

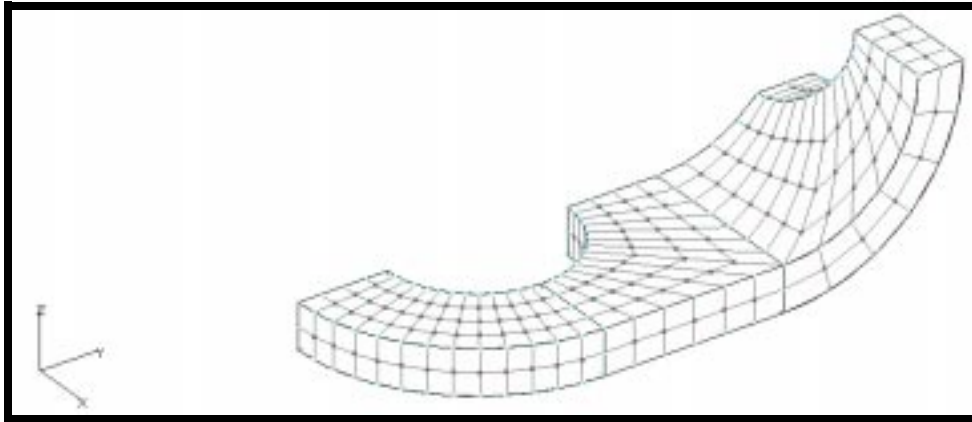
OK

Cancel

25. Now hex mesh the final solid.

Mesh/Geometry/HexMesh Solids...*Select Solid:**(the back solid)*

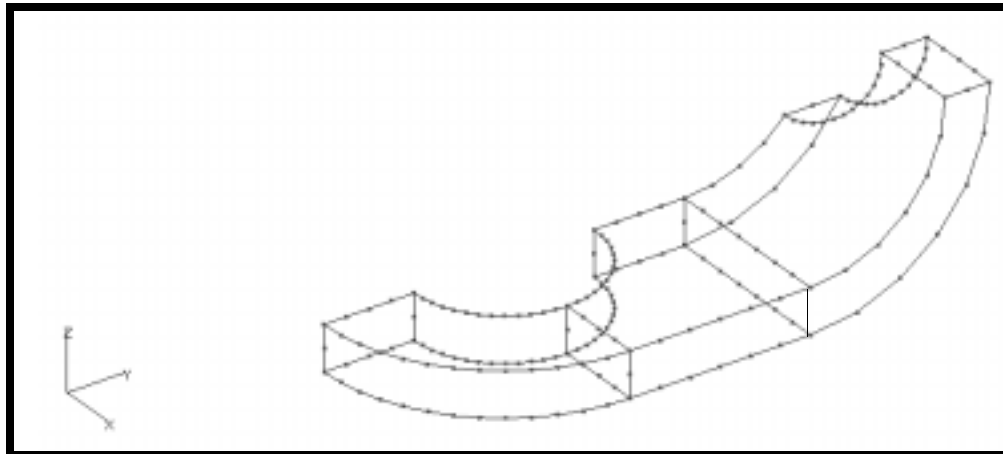
26. Choose *View/Redraw* to redraw the graphics.

View/Redraw**<Ctrl+D>****Figure C.17 - Fully hex meshed Quarter Model**

27. Check for free edges.

View/Select**<F5>***Model Style:***● Free Edge**

Figure C.18 - Cracks in hex mesh



28. Check coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

When asked, "OK to Specify Additional Range of Nodes to Merge?," select No.

No

Options:

OK

Merge Coincident Entities

Figure C.19 - Final mesh mpart with out cracks

