Altair HyperWorks

Altair HyperMesh 2019 Tutorials

HM-2060: Creating and Editing Solid Geometry

HM-2060: Creating and Editing Solid Geometry

In this tutorial, you will learn what solid geometry and topology is, and what 3D topology looks like.

Solids are geometric entities that define a three-dimensional volume. Geometric entities are defined as follows:

- Point: 0 dimensional; has only x, y, and z coordinates
- Line: one-dimensional; has length, can be curved through three-dimensional space
- Surface: two-dimensional; has an area
- Solid: three-dimensional; has a volume

The use of solid geometry is helpful when dividing a part into multiple volumes, for example, to divide a part into simple, mappable regions for hex meshing.

Model Files

This exercise uses the `solid_geom.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.
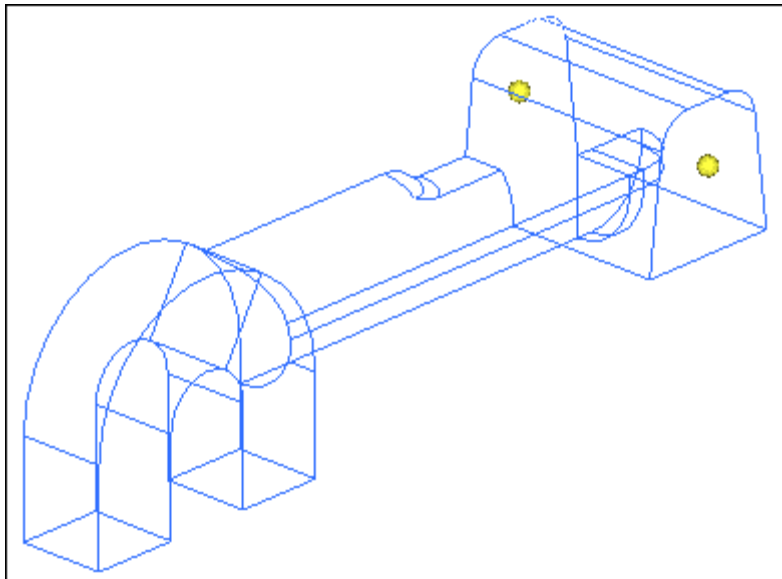
Exercise: Creating and editing solid geometry

Step 1: Retrieve model file, `solid_geom.hm`.

1. Start HyperMesh Desktop.

2. From the menu bar, click *File* > *Open* > *Model*.

3. In the **Open Model** dialog, open the `solid_geom.hm` model file.

### Step 2: Create solid geometry from the bounding surfaces.

1. To open the **Bounding Surfaces**, click *Geometry* > *Create* > *Solids* > *Bounding Surfaces* from the menu bar.

2. Verify that the **Auto select solid surfaces** check box is selected.

3. Select one surface on the part. HyperMesh automatically selects all of the surfaces.

4. Click *create*. HyperMesh creates the solid, and the **status bar** displays message that says one solid has been created.

    Note: The solids are identified by thicker lines than surfaces.

5. To exit the panel, click *return*.

### Step 3: Create a solid geometry cylinder using primitives.

1. To open the **Cylinder Full** panel, click *Geometry* > *Create* > *Solids* > *Cylinder Full* from the menu bar.

2. Click *bottom center* and then select one of the temporary nodes as illustrated in the following image. The cursor advances to the **normal vector** selector.

Select one of these nodes.
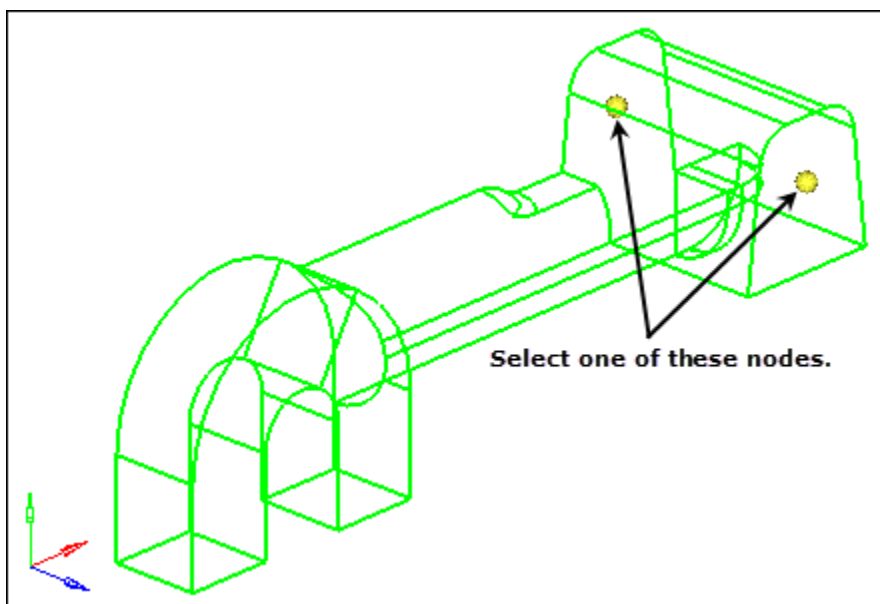
3. Select the remaining temporary node shown in the previous image.

4. In the **Base radius=**, enter `1.5`.

5. In the **Height=** field, enter `25`.

6. Click *create*. HyperMesh creates **solid cylinder** in the middle of the first solid that was created.

7. To exit the panel, click *return*.

## Step 4: Subtract the cylinder's volume from the rest of the part.

1. To open the **Boolean** subpanel, click *Geometry* > *Edit* > *Solids* > *Boolean* from the menu bar.

2. Verify that **operation type:** is set to *simple (combine all)*.

3. Set **operation:** to *A-B (remove B from A)*.

4. Verify that the **A: solids** selector is active, and then select the original solid.

5. Activate the **B: solids** selector, and then select the solid cylinder created in step 3.

6. Click *calculate*.

7. Click *return*.

8. To confirm the material has been removed, click 🖱 on the **Visualization** toolbar and rotate the model to inspect the part.
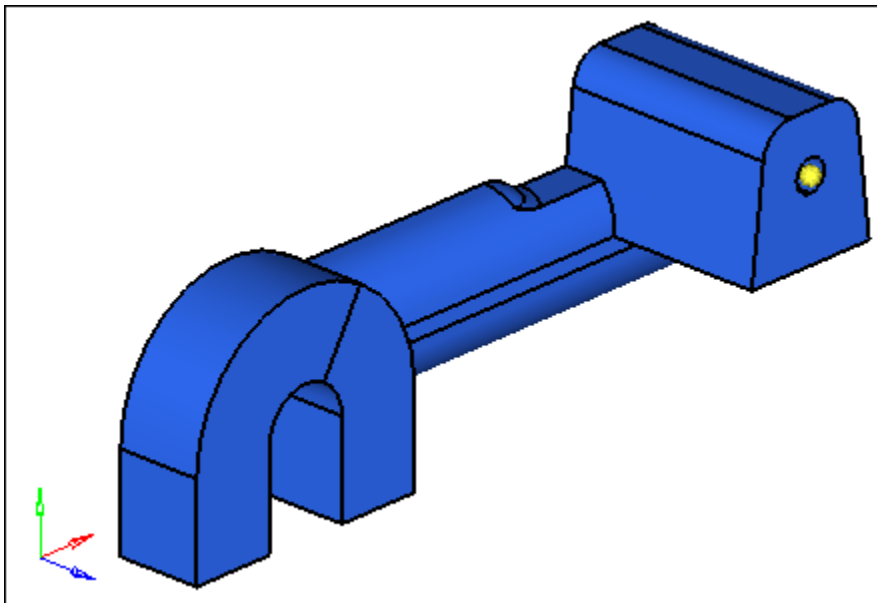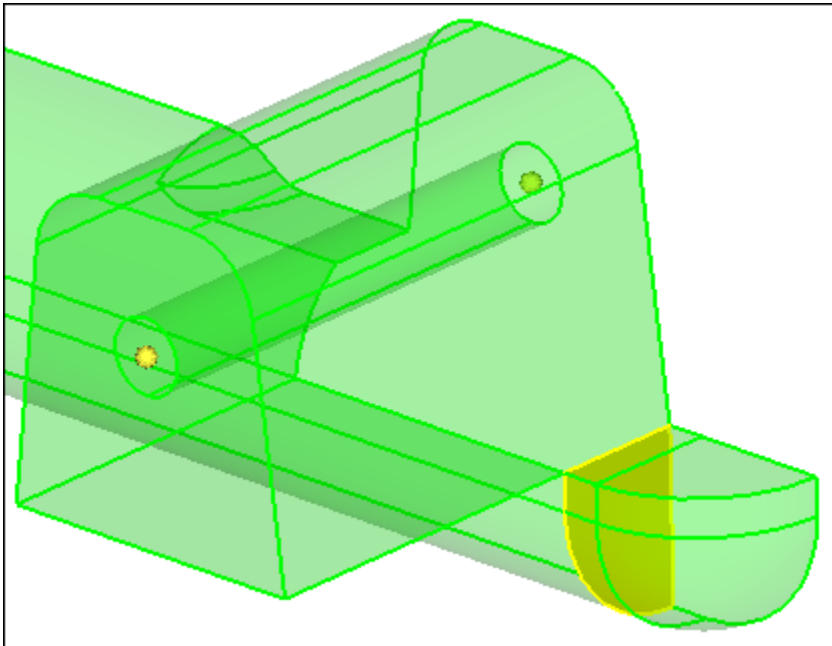

Step 5: Split the solid geometry using bounding lines.

1. To go to the **trim with lines** subpanel, click *Geometry* > *Edit* > *Solids* > *Trim with Lines* subpanel.

2. Activate the **with bounding lines** selector and set it to *solids*.

3. Click anywhere on the model to select it.

4. Under the **with bounding lines** selector, activate the *lines* selector.

5. Select the four lines indicated in the following image.
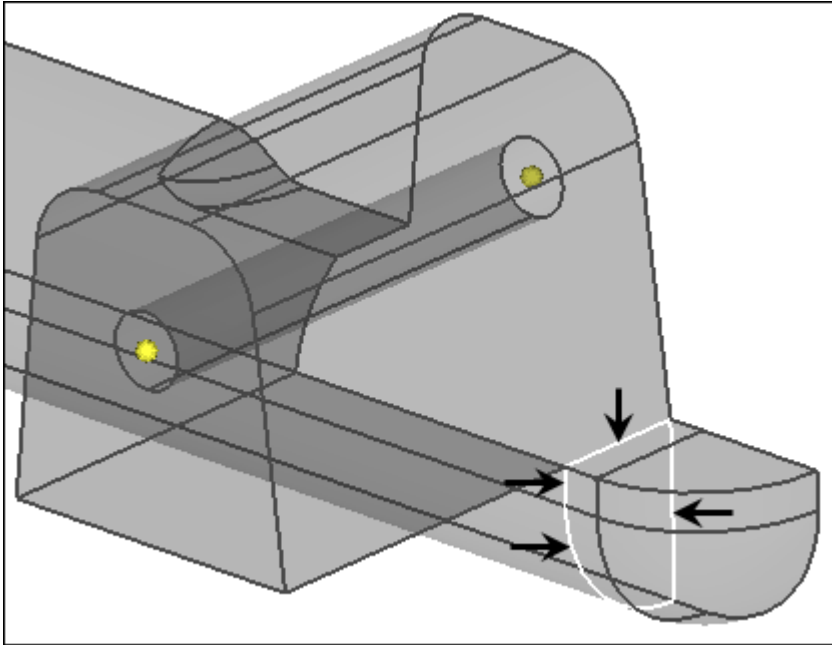
6. Click *trim*. HyperMesh trims a plane.

   Note: The two solids now intersect.

Step 6: Split the solid geometry using a cut line.

In this step, you should still be in the **Solid Edit** panel, **trim with lines** subpanel.

1.   Activate the **with cut lines** selector and set it to ***solids***.

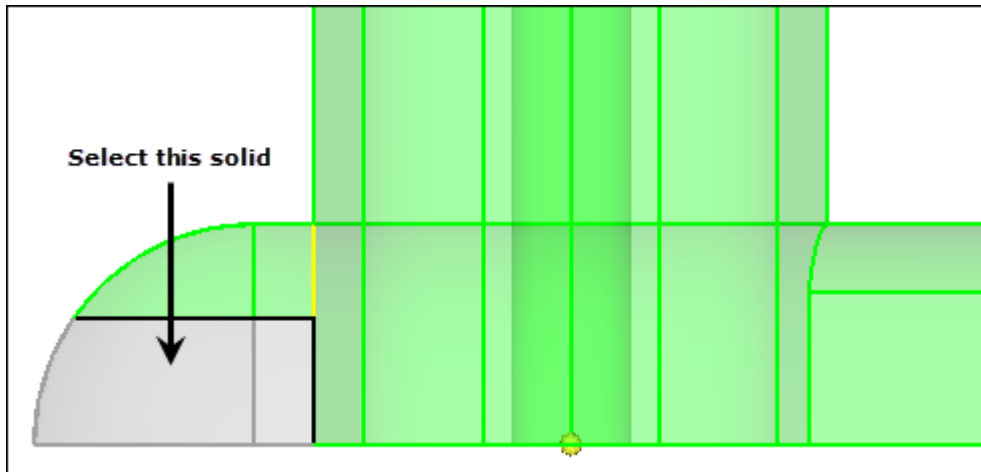2.   Select the small, tetrahedral shaped solid created in step 5.
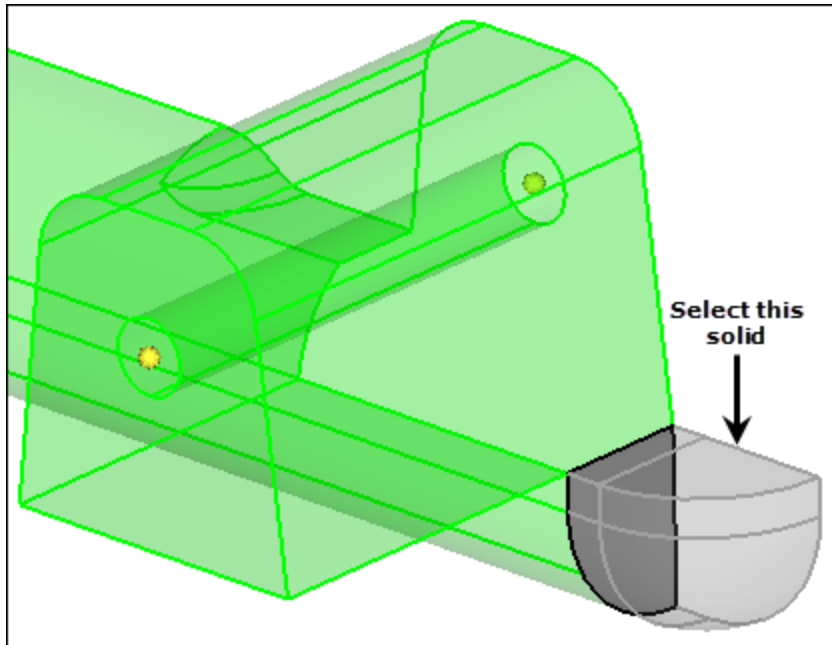
3.   In the **Model** browser, **View** folder, right-click on ***View1*** and select ***Show*** from the context menu.

4.   Click ***drag a cut line***.

5.   To define the end points of a line that roughly divides the tetrahedral solid in half, select the two locations indicated in the following image.
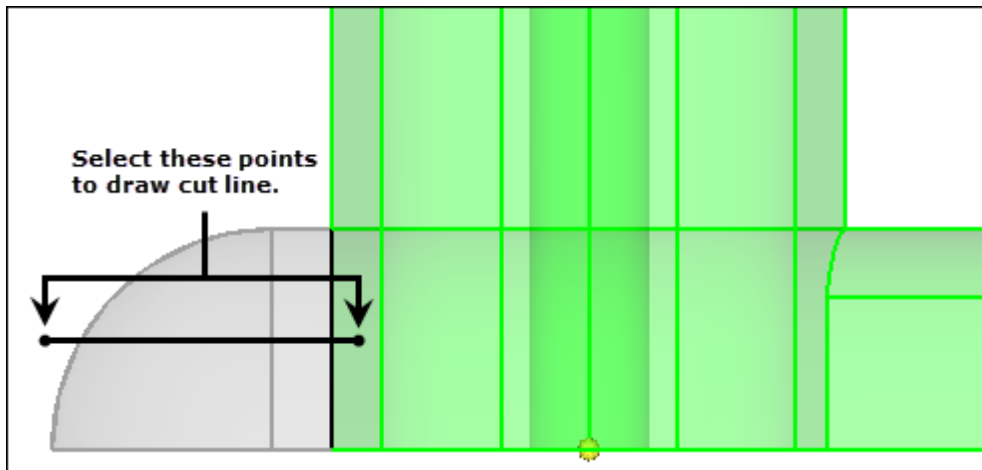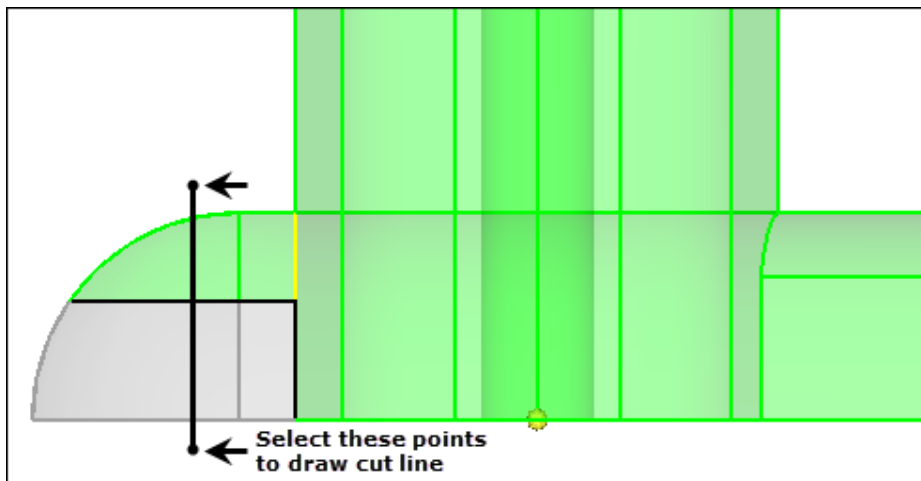

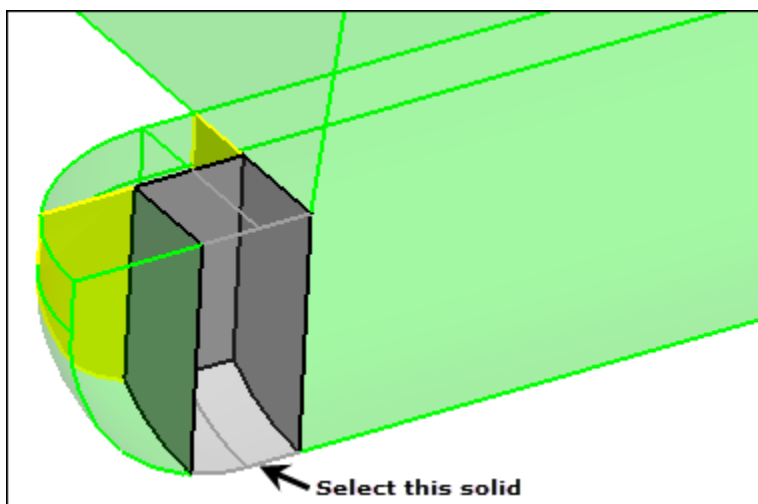6.   To split the solid, middle-mouse click.

7. Select the half of the original tetrahedral solid indicated in the following image.
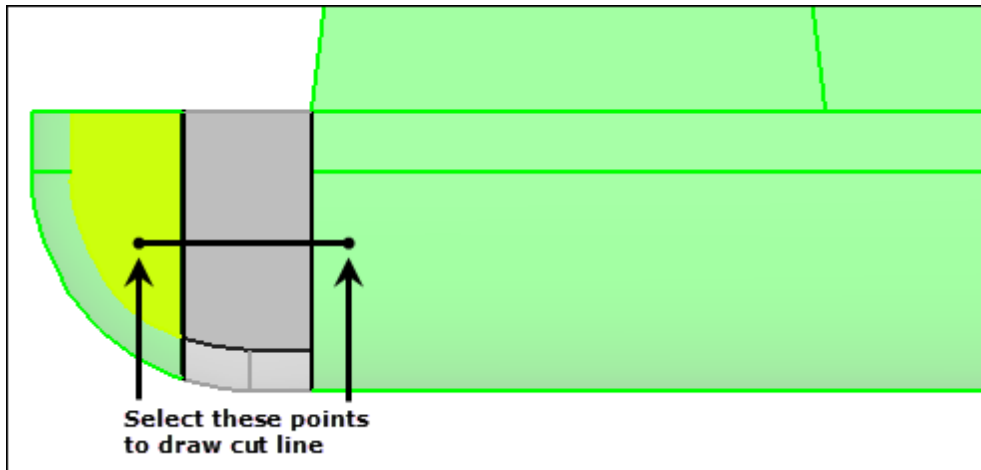

8. To split the solid indicated in the following image, repeat steps 6.4 through 6.6.

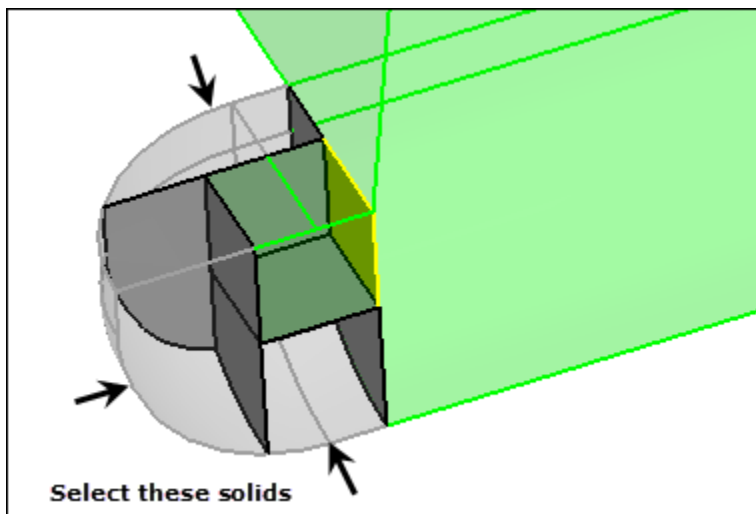9. Select the solid indicated in the following image.

△ Altair

10. In the **Model** browser, **View** folder, right-click on **View2** and select **Show** from the context menu.

11. To split the solid indicated in the following image, repeat steps 6.4 through 6.6.

Select these points
to draw cut line

Step 7: Merge solids together.

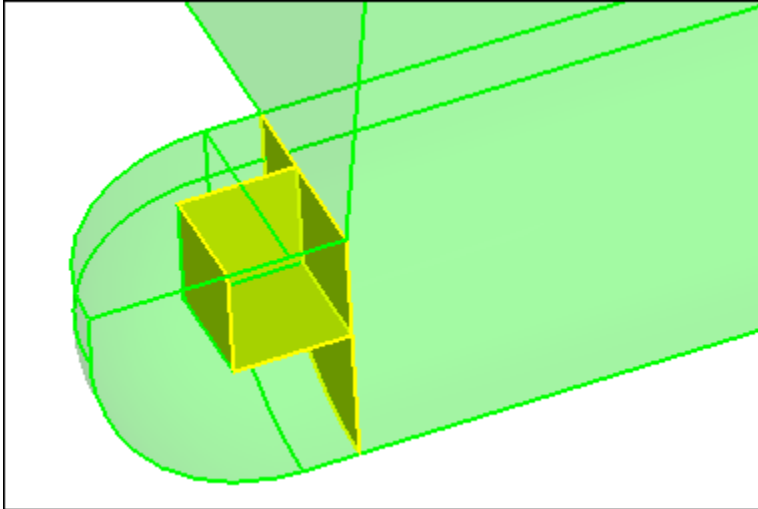In this step, you should still be in the **Solid Edit** panel.

1. Go to the **merge** subpanel.

2. Activate the **to be merged: solids** selector.

3. Select the three solids indicated in the following image.

Select these solids

4. Click **merge**. HyperMesh merges the solids.

   Note: The resulting solids in the tetrahedral area should resemble the following image. There should be two solid entities, with one of them being hexahedral in shape in the corner.
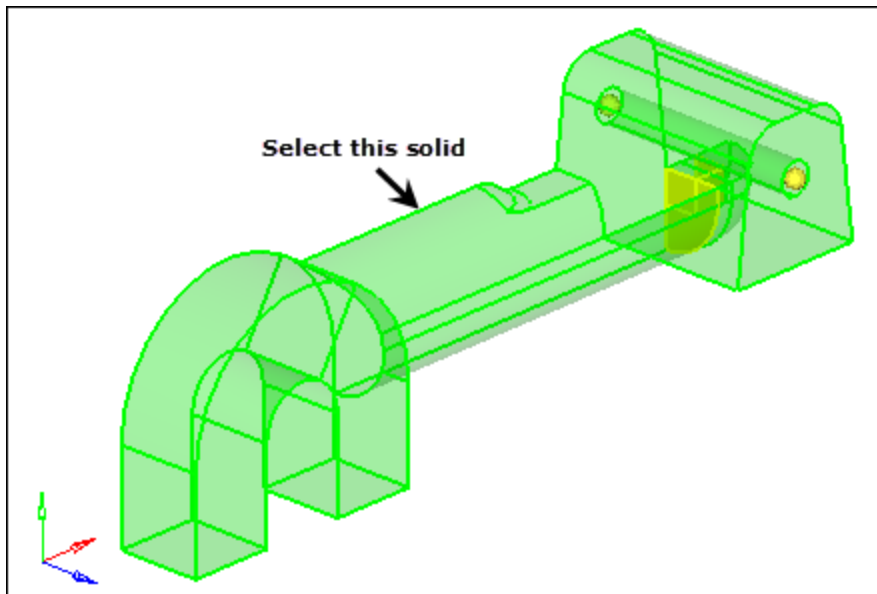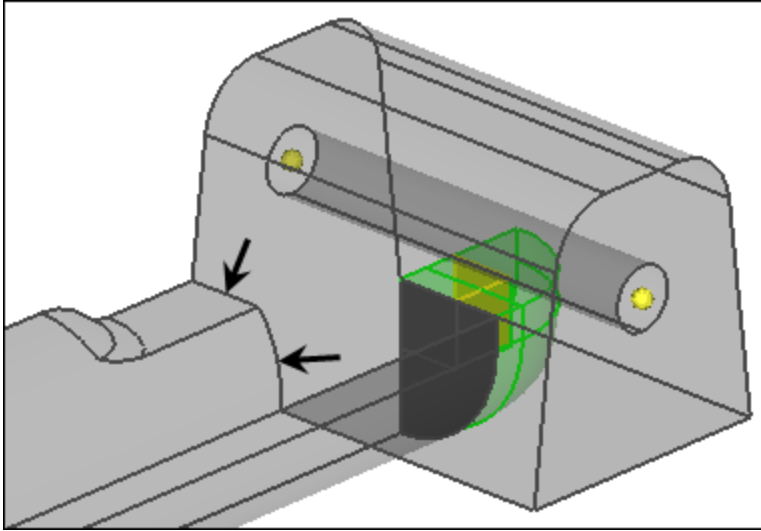
Step 8: Split the solid geometry with a user-defined plane.

In this step, you should still be in the **Solid Edit** panel.

1. Go to the **trim with plane/surf** subpanel.

2. From the **Model** browser, **View** folder, right-click on *View3* and select *Show* from the context menu.

3. Activate the **with plane** selector and set to **solids**.

4. Select the large solid representing the majority of the part.

5. Set the **orientation vector** to *N1*, *N2*, *N3*.

6. With **N1** active, press and hold your left mouse button, and move the mouse cursor over one of the edges indicated in the following image. HyperMesh highlights the edge.

7. Release the mouse button, and left-click in the middle of the edge. A green temp node appears at the location to indicate the selection for **N1**.
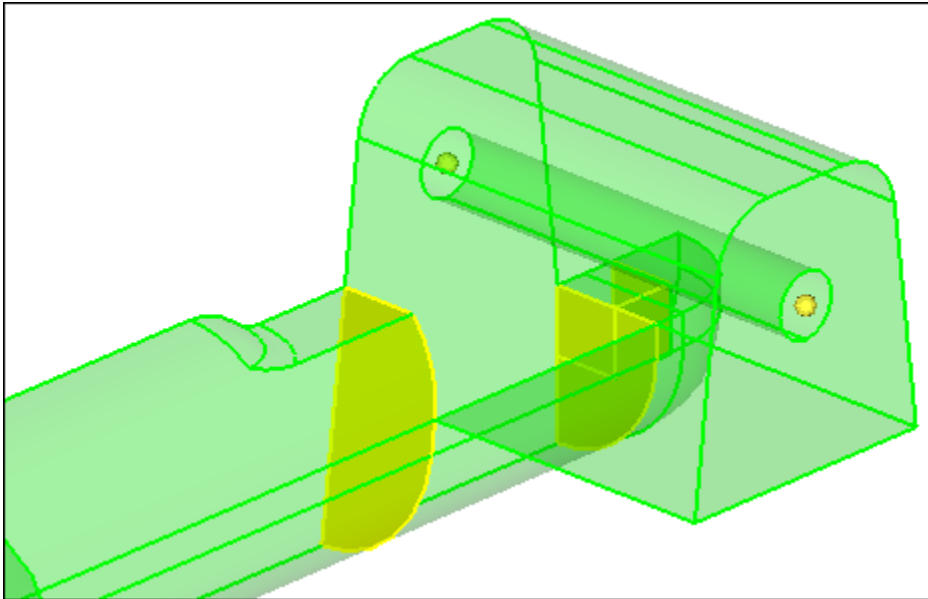
    Note: The plane selector advances to the **N2** selection.

8. In the same manner, highlight the other line shown in the previous image.

9. Release the mouse button, and select two nodes along its length.

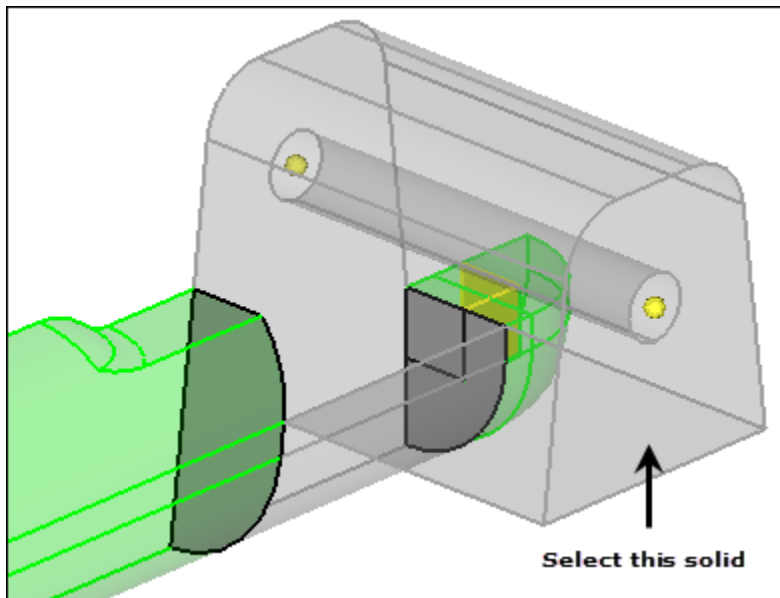   Note: Your selection should look similar to the following image.
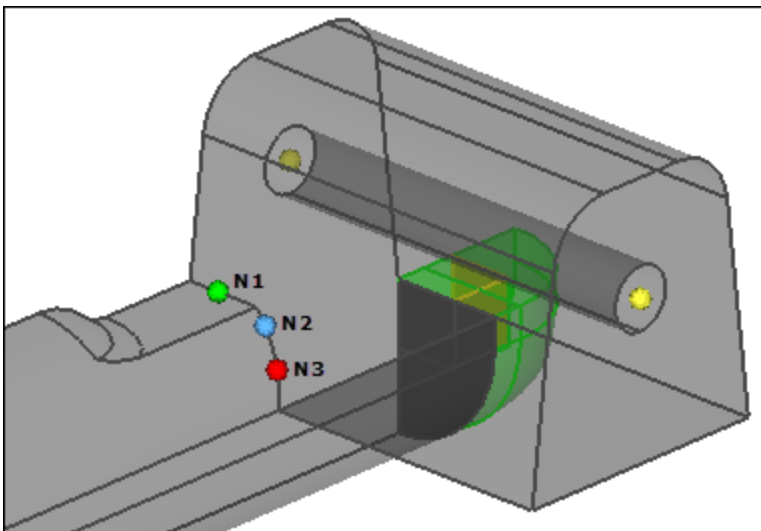
10. Click *trim*. HyperMesh trims the solid.

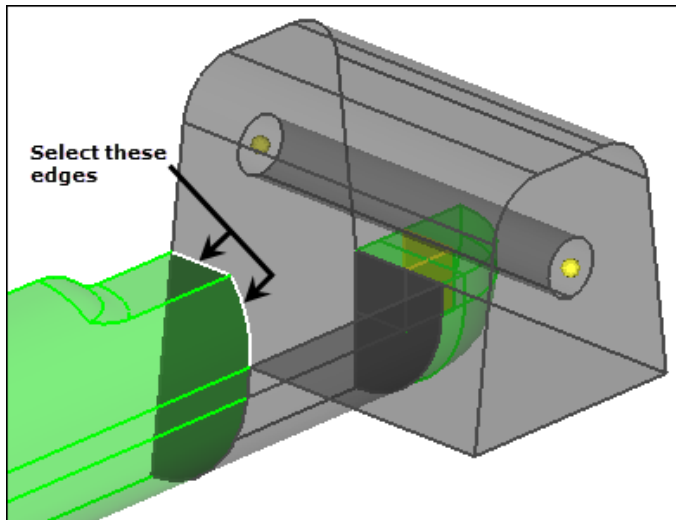## Step 9: Split the solid geometry with a swept line.

In this step, you should still be in the **Solid Edit** panel.

1. Go to the **trim with lines** subpanel.

2. Activate the **with sweep lines** selector and set it to *solids*.

3. Select the solid with the cylinder removed.

Select this solid

4. Activate the **lines** selector.

△ Altair

5. Select the edges indicated in the following image.

6. Under **sweep to:**, set the **orientation vector** to *x-axis*.
7. Under the **orientation vector**, verify that *sweep all* is selected.
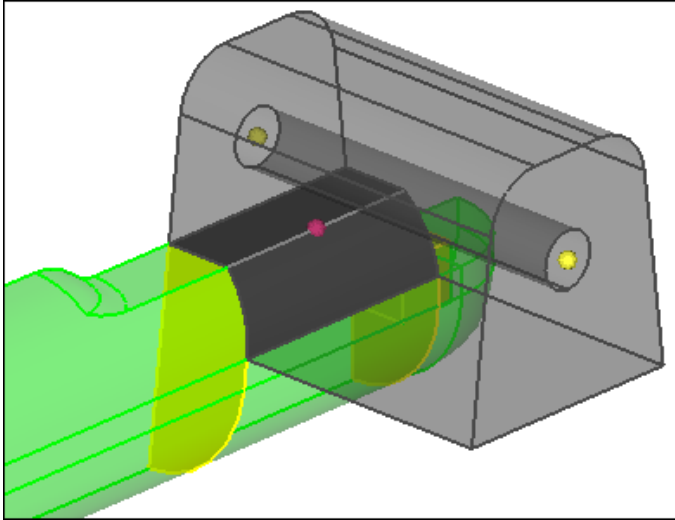6. Click *trim*. HyperMesh trims the solid.

Step 10: Split the solid geometry with a principal plane.

In this step, you should still be in the **Solid Edit** panel.

1. Go to the **trim with plane/surf** subpanel.
2. Activate the **with plane** selector and set it to **solids**.
3. Select the solid with the cylinder removed.
4. Set the **orientation vector** to *z-axis*.
5. Press and hold your left mouse button, and move the mouse cursor over the edge indicated in the following image. HyperMesh highlights the edge.
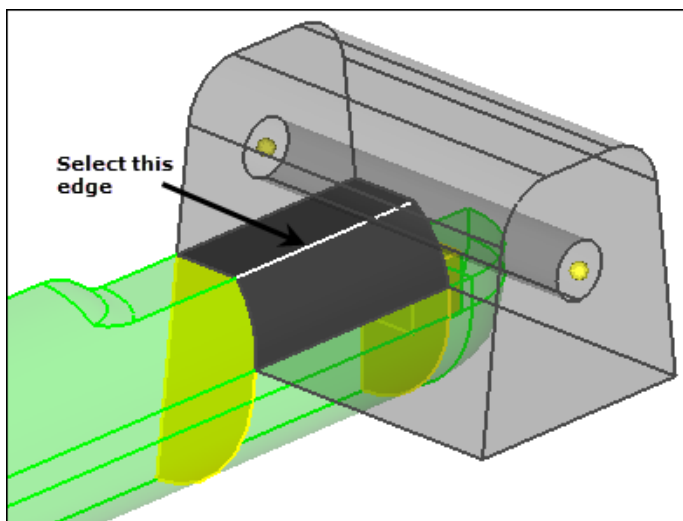
6. Release the mouse button, and left-click anywhere along the edge. A purple temp node appears at the location to indicate the selection for the base node.


7. Click *trim*. HyperMesh trims the solid.
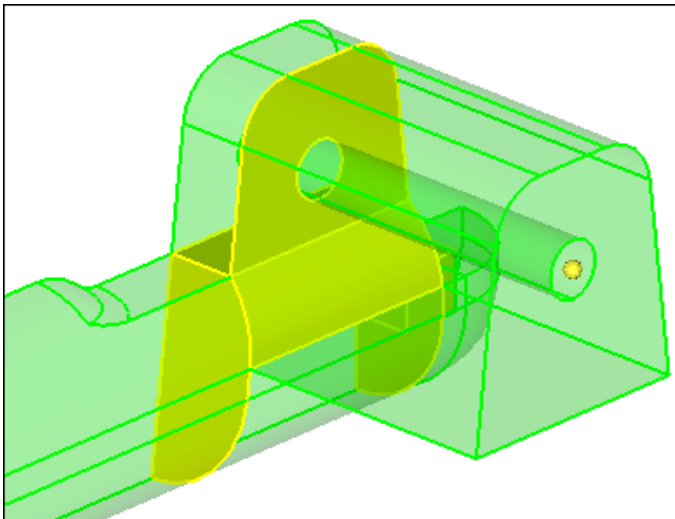
8. To exit the panel, click *return*.

Step 11: Split the solid geometry by creating surfaces inside the solids.

1. To open the **Spline/Filler** subpanel, click *Geometry* > *Create* > *Surfaces* > *Spline/Filler* from the menu bar.

2. Clear the *Auto create (free edges only)* and *keep tangency* check boxes.
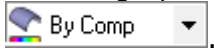
3. Select the five lines indicated in the following image.
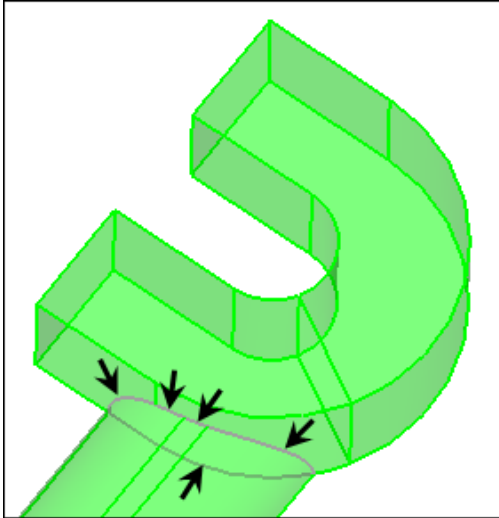
4. Click *create*. HyperMesh creates a surface.

5. To exit the panel, click *return*.

6. To go to the **Trim with Plane/Surface** subpanel, click *Geometry* > *Edit* > *Solids* > *Trim with Plane/Surfaces* from the menu bar.

7. Activate the **with surfs** selector and set to **solids**.

8. Select the solid with the cylinder removed.

9. Activate the **surfs** selector.

10. Select the surface that you created in step 11.4.

11. Click *trim*.

12. To exit the panel, click *return*.

13. Go to the **Spline/Filler** subpanel.

14. Set the entity selector to **lines.**

15. Select the four lines indicated in the following image.
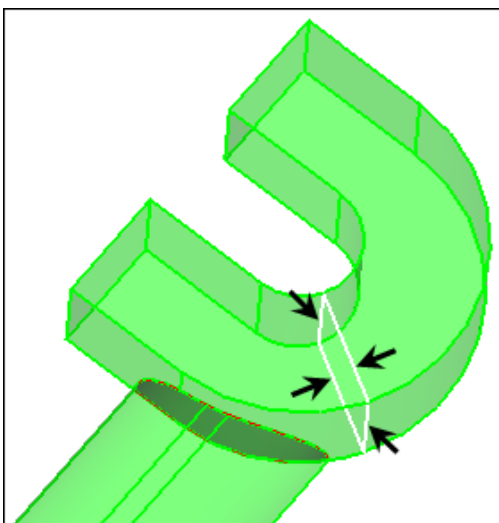
16. Click *create*. HyperMesh creates a surface.

17. To exit the panel, click *return*.

18. Go to the **Trim with Plane/Surface** subpanel.

19. Activate the **with surfs** selector and set it to **solids**.

20. Select the solid you created a surface for in step 11.16.

21. Activate the **surfs** selector.

22. Select the surface that you created in step 11.16.

23. Clear the *extend trimmer* check box.

24. Click *trim*.

25. To exit the panel, click *return*.

Step 12: Suppress extraneous edges on the part.

1. To open the **(Un)Suppress** subpanel, click *Geometry* > *Edit* > *Surface Edge* > *(Un)Suppress* from the menu bar.

2. Click *lines* >> *by geoms*.

3. Verify that the **solids** selector is active.

4. Select the three solids indicated in the following image.

   Tip: To view a more efficient graphical representation of the solids, set the surface
   display mode to ![By Comp] .
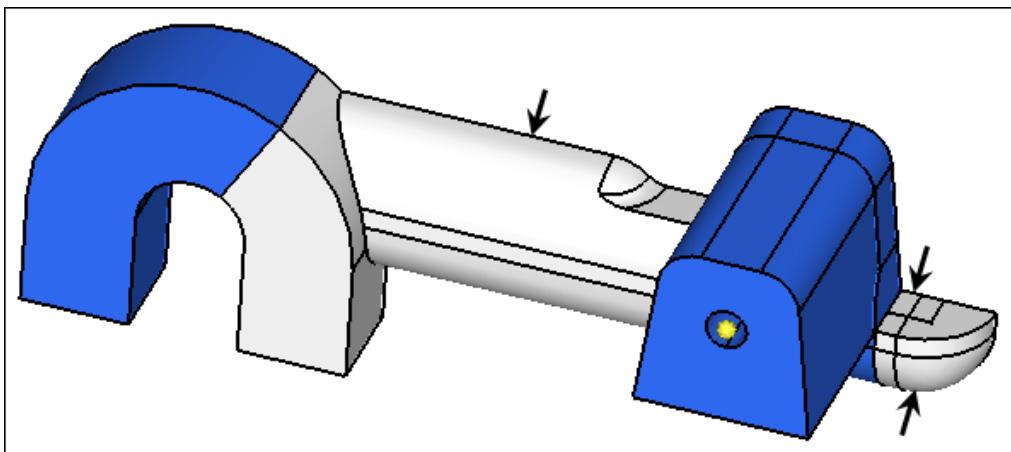
5. Click *add to selection*.

6. In the **breakangle =** field, enter 45.

7. Click *suppress*. HyperMesh suppresses the edges.

8. To exit the panel, click *return*.