



Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-1030: Organizing a Model

altairhyperworks.com

HM-1030: Organizing a Model

A large portion of HyperMesh functionality is organized into panels. Many panels have common attributes and controls, so once you become familiar with the features of one panel, it is much easier to understand other panels.

In this tutorial, you will learn how to:

- Create geometry and organize it into components
- Organize elements into the components
- Rename components
- Identify and delete empty components
- Delete all of the geometry lines
- Reorder the components in a specific order
- Renumber all of the components, starting with ID 1 and incrementing by 1
- Create an assembly
- Organize the constraint

Model Files

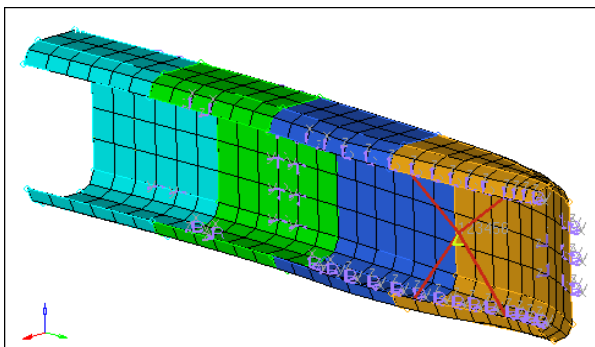
This exercise uses the `bumper.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

Exercise

It is recommend that you review the general overview before completing this tutorial.

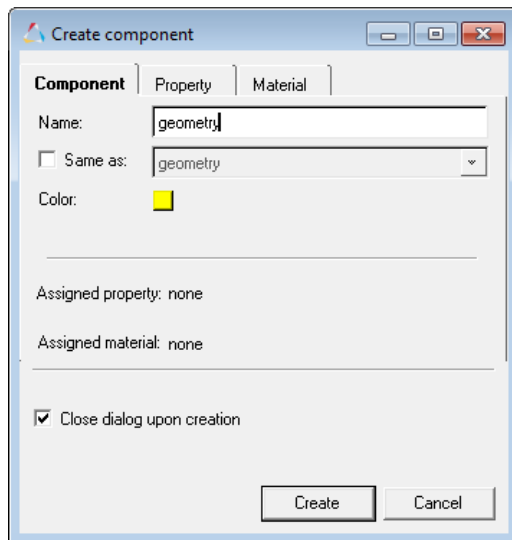
Step 1: Retrieve the model file, bumper.hm.

1. Start HyperMesh Desktop.
2. From the menu bar, click **File > Open > Model**.
3. In the **Open Model** dialog, navigate to your working directory and open the `bumper.hm` model file. A model appears in the graphics area.



Step 2: Create a component named *geometry* to hold the model's geometry.

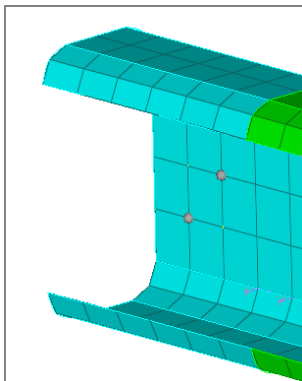
1. To create a new component, click **Collectors > Create > Components** from the menu bar, or right-click in the **Model** browser and select **Create > Components** from the context menu.
2. In the **Create component** dialog, enter *geometry* in the **Name** field.
3. Click the **Color** icon, and select yellow.



4. Click **Create**. The **Model** browser displays the current component collector **geometry** in bold.

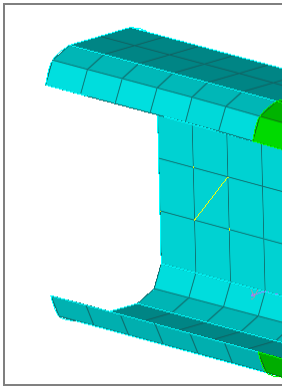
Step 3: Create two geometry lines and organize them into different components.

1. To open the **Lines** panel, click **Geometry > Create > Lines > Standard Nodes** from the menu bar.
2. Activate the **node list** selector.
3. In the graphics area, select two opposite and diagonal nodes of the same element as illustrated in the image below:

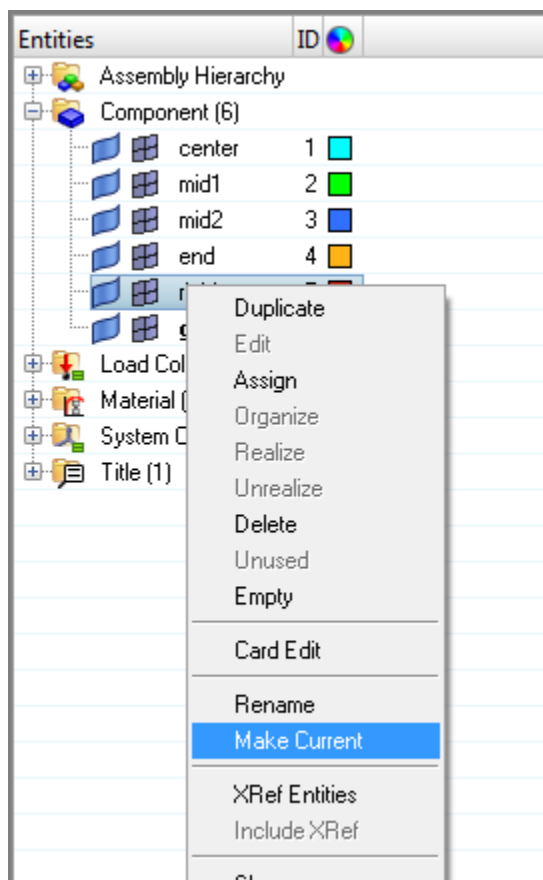


- Click **create**. HyperMesh creates a yellow line.

Note: This line is the same color as the **geometry** component, because it is organized into the current component, **geometry**.



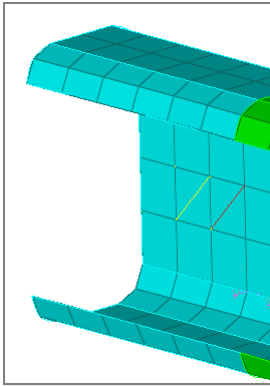
- In the **Model** browser, **Component** folder, right-click on **rigid** and select **Make Current** from the context menu. HyperMesh makes the **rigid** component the active component.



- In the graphics area, select two opposite and diagonal nodes of the same element, but different than the element selected above.


- Click **create**. HyperMesh creates a red line.

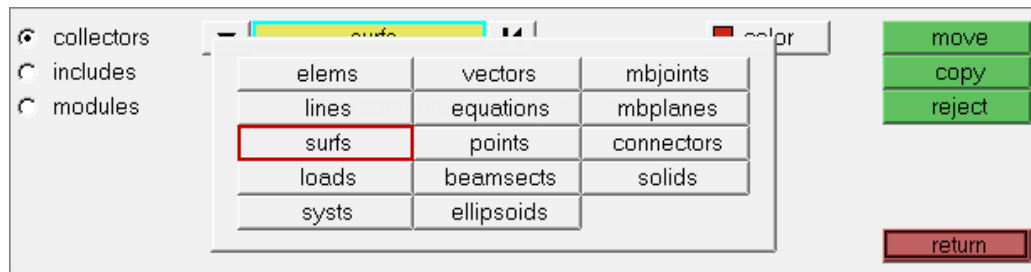
Note: This line is the same color the **rigid** component, because it is organized into the current component, **rigid**.



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

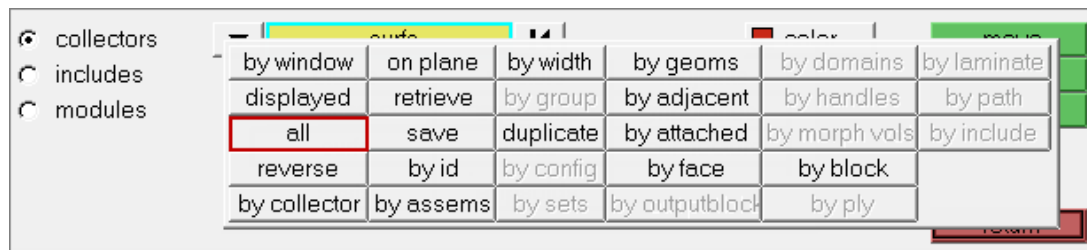
Step 4: Move all of the model's geometry surfaces into the component, **geometry**.

- To open the **Organize** panel, click **Geometry > Organize > Surfaces** from the menu bar.
- To go to the **Collectors** subpanel, click **collectors**.
- Click  on the entity selector, and select **surfs** from the list of entities that can be collected. The entity selector is now set to **surfs**.

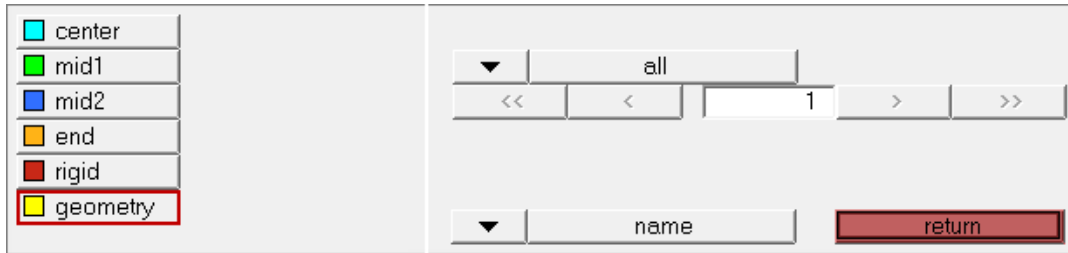


- Click **surfs >> all**. HyperMesh highlights all of the displayed surfaces in white, which indicates they are selected.

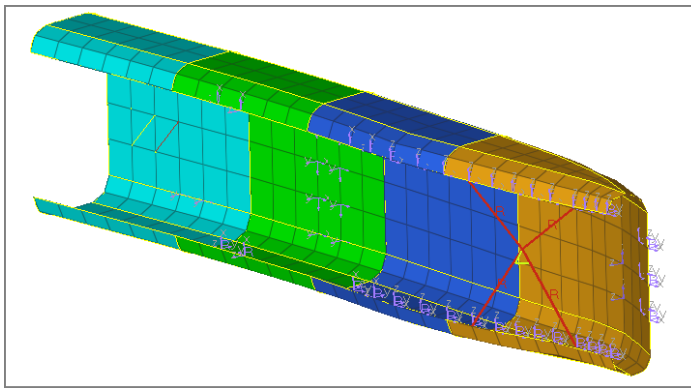
Note: If there are surfaces that are not displayed, HyperMesh still selects them because you selected **surfs >> all**.



- Click **dest component=**.
- From the list of components, select **geometry**.




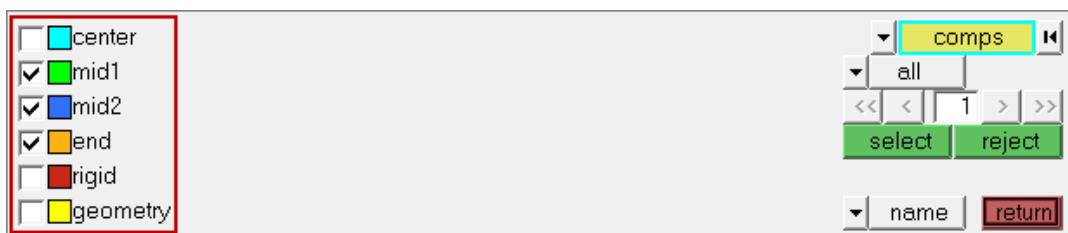
- Click **move**. Hypermesh moves the selected surfaces into the **geometry** component, and colors the geometric entities yellow to match the component color.



Step 5: Move all the model's shell elements (quads and trias) into the component, **center**.

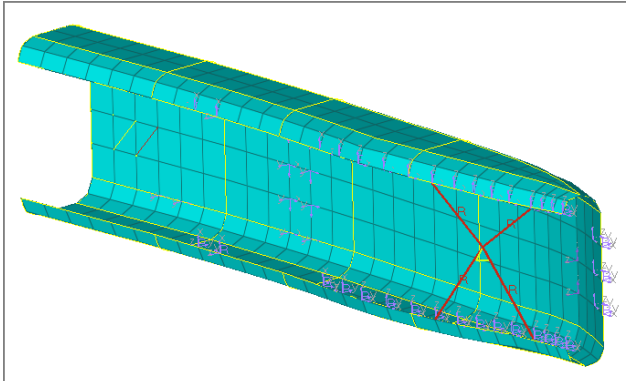
In this step, you should still be in the **Organize** panel, **collectors** subpanel.

- Click  on the entity selector, and select **elems** from the list of entities that can be collected. The entity selector is now set to **elems**.
- Click **elems >> by collector**.
- Select the components: **mid1**, **mid2**, and **end**.



- Optional: To select a component in the graphics area, left-click on it. A check mark appears in the check box of the component you selected in the panel area.
- Optional: To unselect a component, right-click its check box in the panel area, or right-click on it in the graphics area.

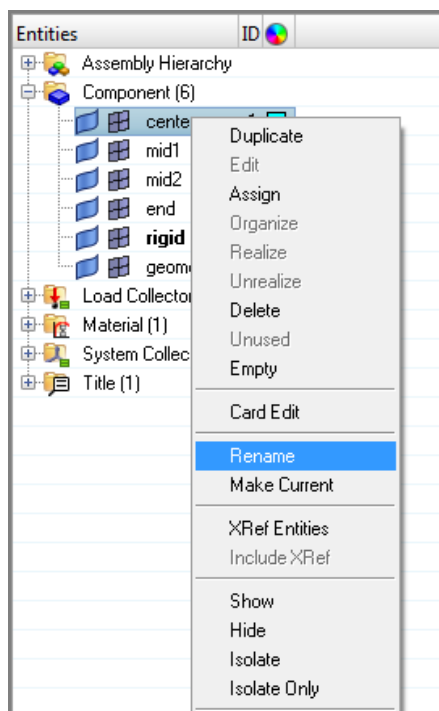
6. To complete your component selection, click **select**.
7. Click **dest component =**.
8. Select the component **center**.
9. Click **move**. HyperMesh moves the elements in the selected components into the **center** component, and colors all of the shell elements cyan blue to match the component color.



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

Step 6: Rename the component **center** to **shells**.

1. In the **Model** browser, **Component** folder, right-click on **center** and select **Rename** from the context menu.



2. In the editable field, rename the component `shells` and then press ENTER.

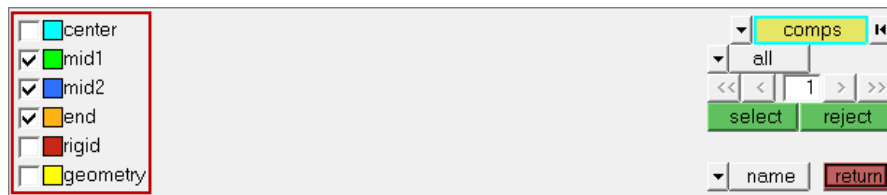
Step 7: Identify and delete all of the empty components.

1. To open the **Delete** panel, press **F2**.
2. Set the entity selector to **comps**.
3. Click **preview empty**. The **Status** bar displays a message that says, "3 entities are empty."

Note: The empty entities are the **mid1**, **mid2**, and **end** components that no longer have elements in them.

4. Click **comps**. HyperMesh displays a complete list of the model's components.

Note: The empty components are indicated with a selected check box.



5. To return to the **Delete** panel, click **return**.
6. Click **delete entity**. The **Status** bar displays a message that says, "Deleted 3 comps 336 elems 28 surfs"

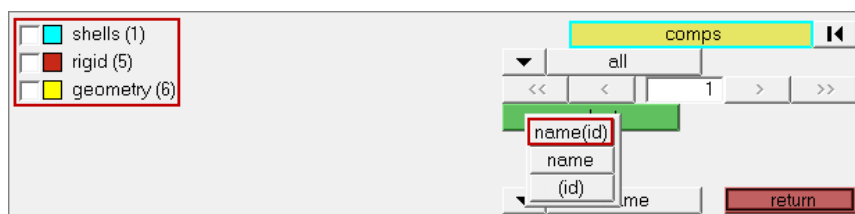
Step 8: Delete all the geometry lines in the model.

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

1. Set the entity selector to **lines**.
2. Click **lines >> all**.
3. Click **delete entity**. HyperMesh deletes the two lines you created earlier.
4. To exit the **Delete** panel, click **return**.

Step 9: Move the component, **geometry**, to the front in the components list.

1. To open the **Reorder** panel, click **Collectors > Reorder > Components** from the menu bar.
2. Click **comps**. HyperMesh displays a complete list of the model's components.
3. On the right side of the panel, click the bottom switch and select **name(id)**. HyperMesh displays the IDs for each component next to its name. The ID for shells is 1, the ID for rigid is 5, and the ID for geometry is 6.



4. Select the component, **geometry**.
5. To complete your selection, click **select**.
6. Under **move to**, select **front**.
7. Click **reorder**. HyperMesh applies the reorder function to the component, **geometry**.
8. To review the reordered list of components, click **comps**.

Note: The component, **geometry**, is at the top of the list. However, it still has the same ID, (6).



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

Step 10: Renumber the components to be the same as their position in the list.

1. To open the **Renumber** panel, click **Collectors** > **Renumber** > **Components** from the menu bar.
2. Go to the **single** subpanel.
3. Set the entity selector to **comps**.
4. Click **comps**. HyperMesh displays a complete list of the model's components.
5. On the right side of the panel, click **comps** >> **all**.
6. To complete your selection, click **select**.
7. In the **start with =** field, enter 1.
8. In the **increment by =** field, enter 1.
9. In the **offset =** field, enter 0.
10. Click **renumber**. HyperMesh rennumbers the components.
11. To review the renumbered list of components, click **comps**.

Note: The components are numbered according to their position in the list. Set the view to **name(id)** to see the numbers.

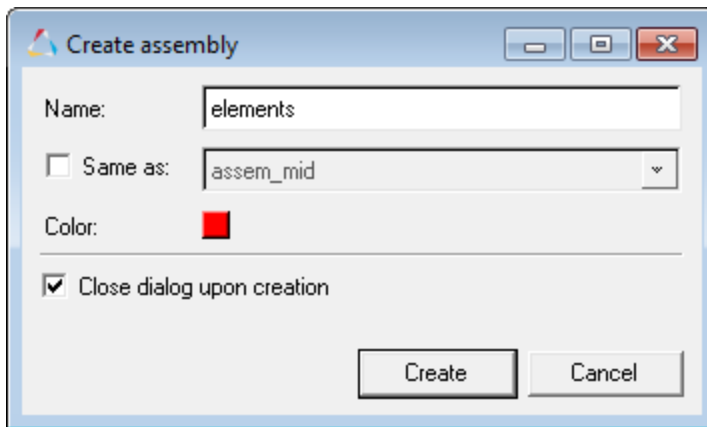


12. To exit the panel, click **return**.

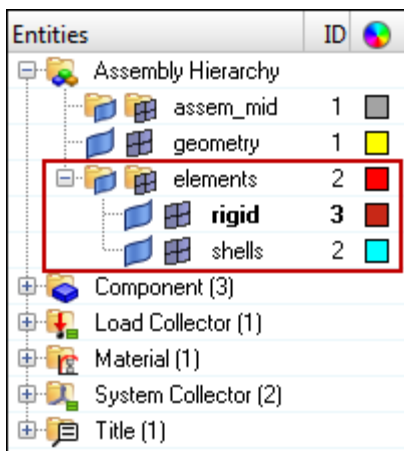
Note: Having components with IDs that do not reflect their position in the model's list of components will not result in errors. However, having components with IDs that do reflect their position in the model's list of components can be helpful for organizational purposes.

Step 11: Create an assembly containing the components, *shells* and *rigid*.

1. In the **Model** browser, right-click and select **Create > Assembly** from the context menu.
2. In the **Create Assembly** dialog, enter `elements` in the **Name** field.
3. Select a **Color** for the assembly.



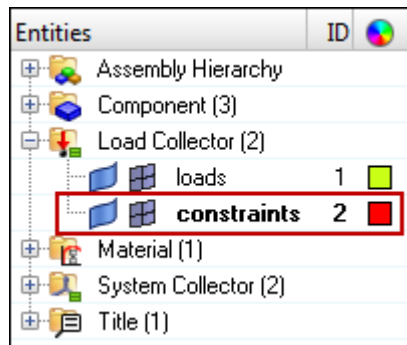
4. Click **Create**. HyperMesh creates the assembly.
5. In the **Model** browser, **Component** folder, select **rigid** and **shells**.
Tip: To select multiple components, press **Ctrl** + left-click.
6. To add the selected components to the **elements** assembly, drag the components, using the left mouse button, over the **elements** assembly until it highlights.



Step 12: Create a load collector named *constraints*.

1. To create a load collector, click **Collectors > Create > Load Collectors** from the menu bar, or right-click in the **Model** browser and select **Create > Load Collector** from the context menu.
2. In the **Create Load Collector** dialog, enter `constraints` in the **Name** field.
3. Click the **Color** icon, and select red.
4. Click **Create**. HyperMesh creates the load collector, and the **Status** bar displays a message that says, "Load collector created".

Note: The `constraints` load collector is displayed in bold in the **Model** browser, which indicates that it is the active load collector. Any loads that are created will be organized into this load collector.



Step 13: Move the model's one constraint into the load collector, *constraints*.

The existing load collector, `loads`, contains several forces and one constraint. In this step, you will use the **Organize** panel to move the one constraint in the load collector, `constraints`.

1. To open the **Organize** panel, click **Collectors > Organize > Load Collectors** from the menu bar.
2. Go to the **collectors** subpanel.
3. Set the entity selector to `loads`.
4. Select `loads >> by config`.
5. Click `config =`, and select `const`.
6. In the center of the panel, toggle from **displayed** to **all**.

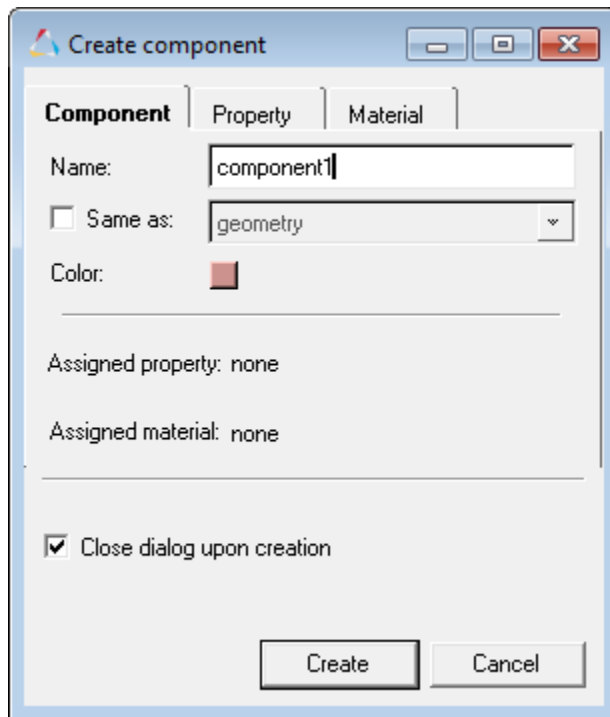


7. Click **select entities**.

8. Verify that **dest component=** is set to the load collector, **constraints**.
9. Click **move**. HyperMesh moves the selected constraints into the load collector, **constraints**.

Step 14: Create a component from the *Model Browser*.

1. In the **Model** browser, right-click and select **Create > Component** from the context menu.
2. In the **Create Component** dialog, enter `component1` in the **Name** field.
3. Click the **Color** icon, and select pink.



4. Click **Create**. HyperMesh creates the component, and appends it to the list.
5. In the **Model** browser, expand the **Components** folder to see that **component1** is boldfaced in the list, which indicates that it is the current component.

Step 15: Review the existing assembly elements from the *Model Browser*.

1. In the **Model** browser, expand the **Assembly Hierarchy** folder and then expand the **elements** assembly folder. It contains two components, **rigid** and **shells**.

Note: Use can use the **Assemblies** panel to add components from one assembly to another assembly. The **Model** browser does not allow you to do this, but you can create assemblies from it.

Step 16: Add the components, geometry and component1, to the assembly, assem_mid, using the Model browser.

1. In the **Model** browser, select the **geometry** and **component1** components.
Tip: To select multiple items in the **Model** browser one at a time, press and hold **Ctrl** and then left-click the items. If you wish to select multiple items in the **Model** browser at once, left-click the first item, press and hold **SHIFT**, and then left-click the last item in the list.
2. To add the selected components to the **assem_mid** assembly, drag the components, using the left mouse button, over the **assem_mid** assembly until it highlights.

Step 17: Rename assem_mid to assem_geom in the Model browser.

1. In the **Model** browser, **Assembly** folder, right-click on **assem_mid** and select **Rename** from the context menu.
2. In the editable field, rename the assembly `assem_geom` and then press **Enter**.

Step 18: Delete component1 from the Model Browser.

1. In the **Model** browser, right-click on **component1** and select **Delete** from the context menu.
2. In the **HyperMesh** dialog, click **Yes** to confirm that you wish to delete the component. HyperMesh deletes **component1**.

Note: In the **Model** browser, there is no longer a boldfaced component name. This indicates that there is no current component specified.

Step 19: Set the current component from the Model Browser.

1. In the **Model** browser, **Component** folder, right-click on **shells** and select **Make Current** from the context menu.

Note: The **shells** component name is boldfaced.