



Altair

HyperWorks

HM-4320: Defining Abaqus Contacts for 3-D Models in HyperMesh

In this tutorial, you will learn how to:

- Load the Abaqus user profile and model
- Start the Abaqus Contact Manager
- Define surfaces for solid elements
- Define surfaces for shell elements
- Define surfaces by set
- Define surface interaction property
- Define contact pair


Model Files

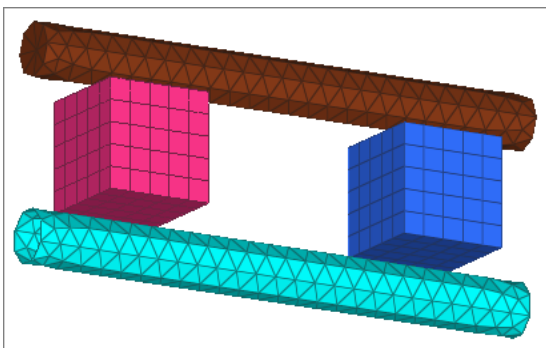
This exercise uses the `contactManager_3D_tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Load the Abaqus user profile and model

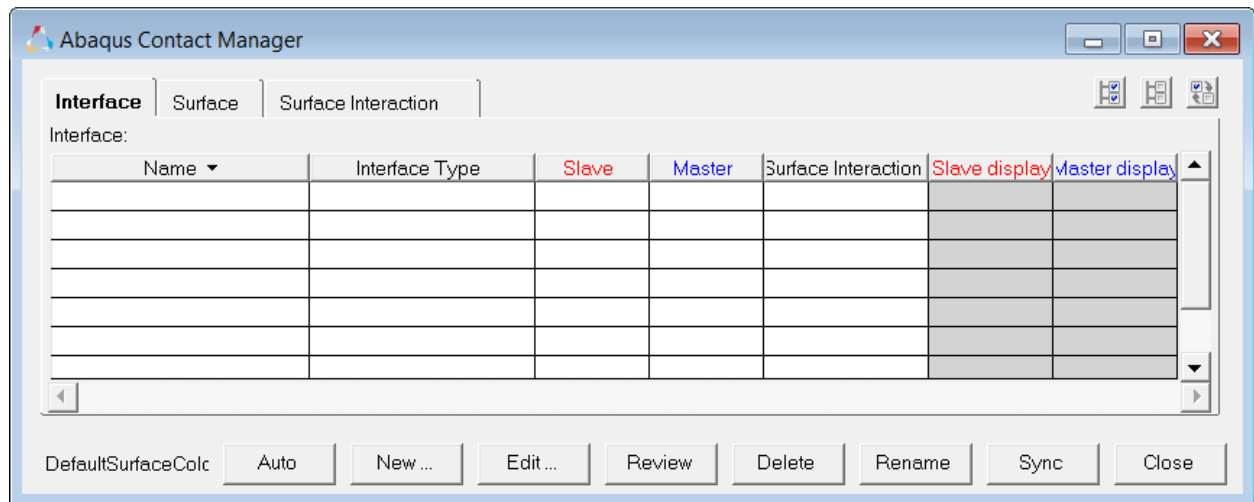
A set of standard user profiles is included in the HyperMesh installation. They include: OptiStruct, Abaqus, Actran, ANSYS, LS-DYNA, MADYMO, Nastran, PAM-CRASH, PERMAS, and RADIOSS. While the user profiles change the appearance of some panels, they do not affect the internal behavior of each function.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard 3D**.
3. Open a model file by clicking **File > Open > Model** from the menu bar, or clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus_contactManager_3D_tutorial.hm` file. The model appears in the graphics area.



Step 2: Start the Contact Manager


1. From the menu bar, click **Tools > Contact Manager**. The **Abaqus Contact Manager** opens.



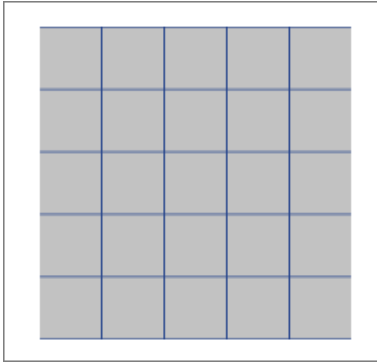
Steps 3 - 5: Defining Surfaces for Solid Elements

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers. In this exercise, you will create surfaces by defining individual element IDs and corresponding faces.

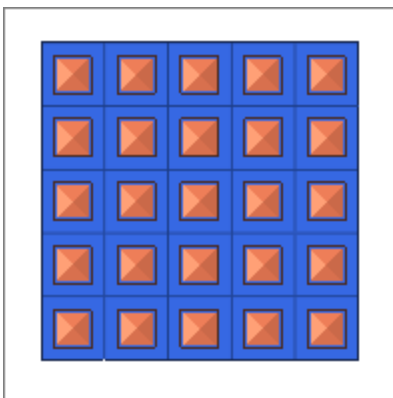
Step 3: Create the "box1-top" surface

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **New**. The **Create New Surface** dialog opens.
3. In the **Name** field, enter `box1-top`.
4. Set **Type** to **Element based**.
5. Click the box next to **Color** and select a color.
6. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
7. In the **Model** browser, **Component** folder, right-click on **BOX_1** and select **Isolate** from the context menu.
8. On the **Standard Views** toolbar, click  (**XY Top Plane View**).
9. In the **Element Based Surface** dialog, click the **Define** tab.
10. Set **Define surface for** to **3D solid, gasket**.
11. Click **Elements**.
12. In the panel area, click **elems >> by collector**.

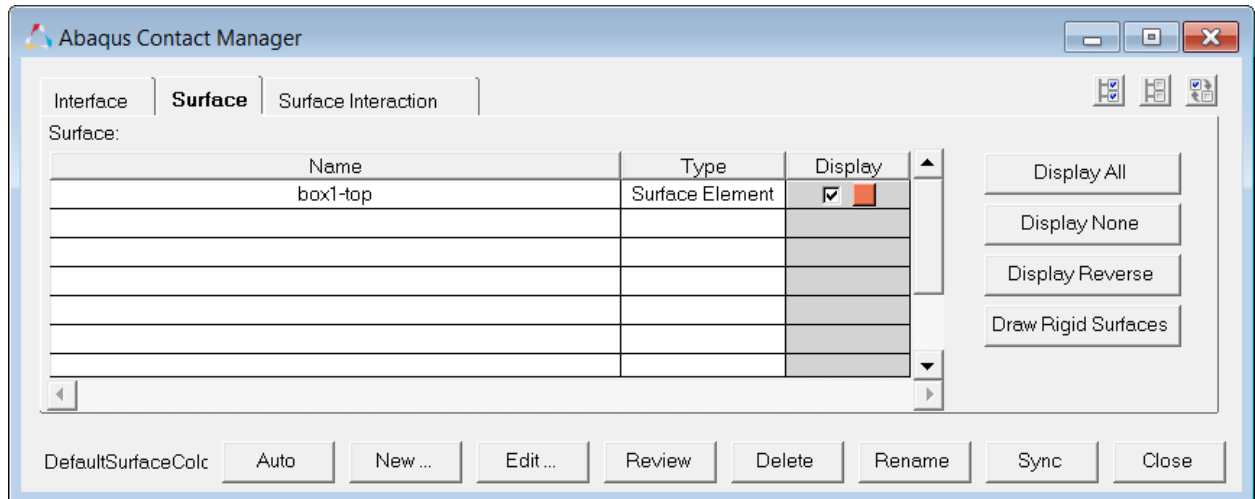
13. Select the component, **BOX_1**.
14. Click **select**. The elements in the **BOX_1** component highlight.




15. Click **proceed** to return to the **Element Based Surface** dialog.
16. Set **Select faces by** to **Solid skin**.
17. Select a color from the **Solid skin color** button.
18. Click **Faces**. HyperMesh creates a temporary skin of the selected elements.
19. Select an element from the top of the solid skin.
20. In the panel area, click **elems >> by face**. All faces at the top of the solid skin highlight.
21. Rotate the model to verify all desired faces are selected.
22. Optional. Deselect any element by right-clicking or add more if you like.
23. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.
24. Click **Add** to add these faces to the current surface. HyperMesh creates special face elements (rectangles with dot in the middle) for display.

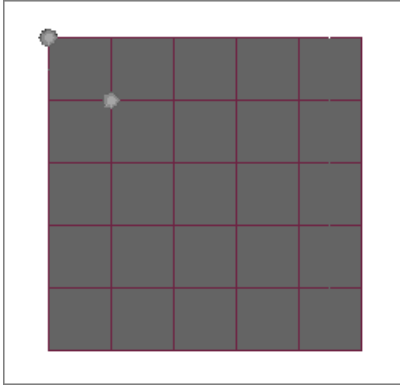


25. Optional. Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
26. When satisfied with the surface definition, click **Close** to return to the **Abaqus Contact Manager**.

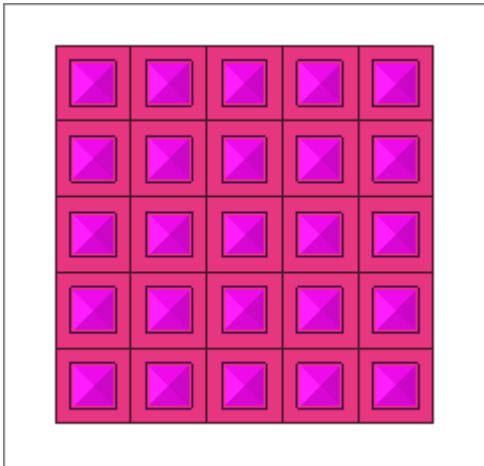


Step 4: Create the "box2-top" surface

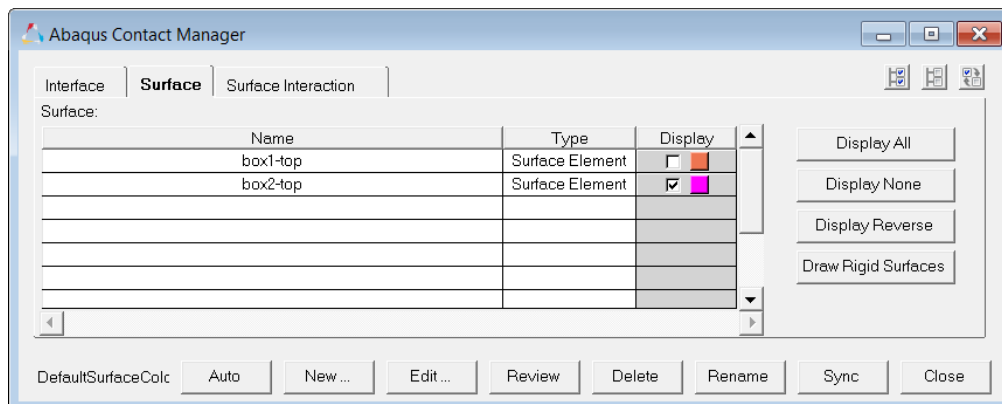
1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**. The **Create New Surface** dialog opens.
4. In the **Name** field, enter `box2-top`.
5. Set **Type** to **Element based**.
6. Click the box next to **Color** and select a color.
7. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. In the **Model** browser, **Component** folder, right-click on **BOX_2** and select **Isolate** from the context menu.
9. On the **Standard Views** toolbar, click  (**XY Top Plane View**).
10. In the **Element Based Surface** dialog, click the **Define** tab.
11. Set **Define surface for** to **3D solid, gasket**.
12. Click **Elements**.
13. In the panel area, click **elems >> by collector**.
14. Select the component, **BOX_2**.
15. Click **select**. The elements in the **BOX_2** component highlight.
16. Click **proceed** to return to the **Element Based Surface** dialog.
17. Set **Select faces by** to **Nodes on face**.
18. Click **Nodes**.
19. Select two corner nodes (or three nodes) from the top of the selected solids as shown below.



20. In the panel area, click **proceed** to return to the **Element Based Surface** dialog.
21. In the **Break Angle** field, enter 30.00.
22. Click **Add** to find all of the faces of the selected solids that fall within the break angle of the face defined by nodes. HyperMesh adds these faces to the current surface and creates special face elements (rectangles with dot at the middle) for display.

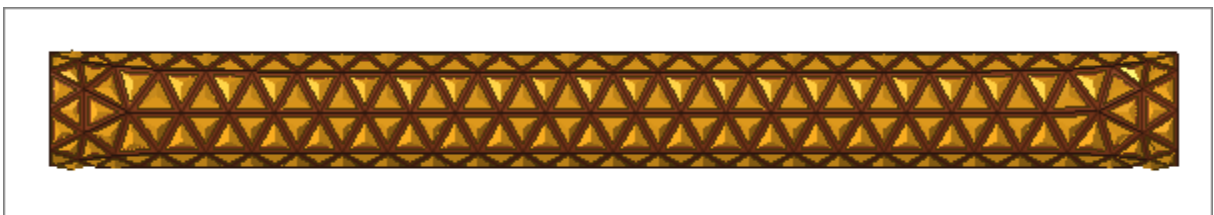


23. Optional. Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
24. When satisfied with the surface definition, click **Close** to return to the **Abaqus Contact Manager**.



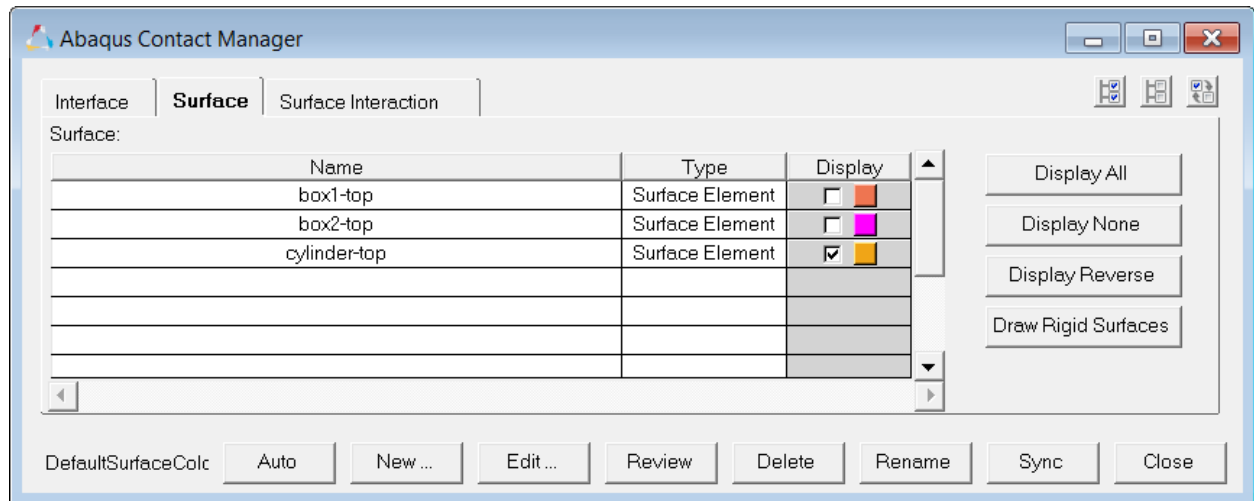
Step 5: Create the "cylinder-top" surface

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to undisplay all surfaces.
3. Click **New**. The **Create New Surface** dialog opens.
4. In the **Name** field, enter `cylinder-top`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. In the **Model** browser, **Component** folder, right-click on **TOP_CYLINDER** and select **Isolate** from the context menu.
9. In the **Element Based Surface** dialog, click the **Define** tab.
10. Set **Define surface for** to **3D solid, gasket**.
11. Click **Elements**.
12. In the panel area, click **elems >> by collector**.
13. Select the component, **TOP_CYLINDER**.
14. Click **select**. The elements in the **TOP_CYLINDER** component highlight.
15. Click **proceed** to return to the **Element Based Surface** dialog.
16. Set **Select faces by** to **Solid skin**.
17. Select a color from the **Solid skin color** button.
18. Click **Faces**. HyperMesh creates temporary skin of the selected elements.
19. Select an element from the solid skin.
20. In the panel area, click **elems >> by face**. The faces all around the solid skin highlight.
21. Rotate the model to verify all desired faces are selected.
22. Optional. Deselect any element by right-clicking, or add more if you like.
23. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.
24. Click **Add** to add these faces to the current surface. HyperMesh creates special face elements (rectangles with dot at the middle) for display.



25. Optional. Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.

26. When satisfied with the surface definition, click **Close** to return to the **Abaqus Contact Manager**.




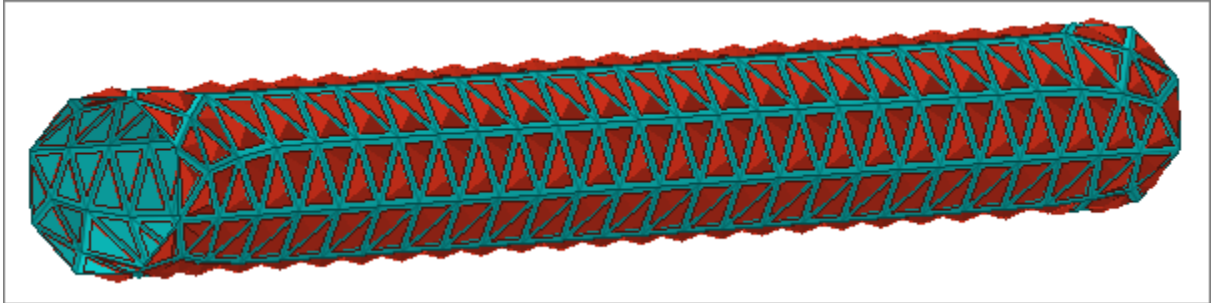
Step 6: Define surfaces for shell elements

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual shell element IDs or sets with corresponding SPOS/SNEG face identifiers. In this exercise, you will create surfaces by defining individual element IDs and corresponding normals to define the SPOS/SNEG faces.

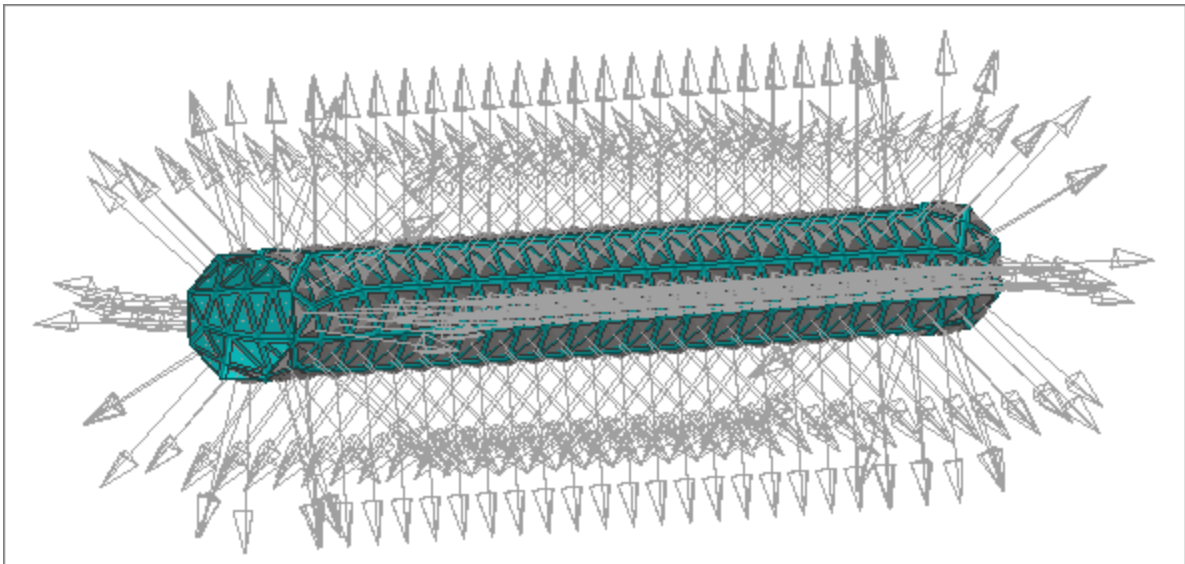
Complete the steps below to create the "cylinder-bot" surface:

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to undisplay all surfaces.
3. Click **New**. The **Create New Surface** dialog opens.
4. In the **Name** field, enter `cylinder-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. Click the **Define** tab.
9. Set **Define surface for** to **3D shell, membrane, rigid**.
10. In the **Model** browser, **Component** folder, right-click on **BOT_CYLINDER** and select **Isolate** from the context menu.
11. In the **Element Based Surface** dialog, click **Elements**.
12. In the panel area, click **elems >> by collector**.
13. Select the component, **BOT_CYLINDER**.
14. Click **select**. The elements in the **BOT_CYLINDER** component highlight.
15. Click **proceed**.

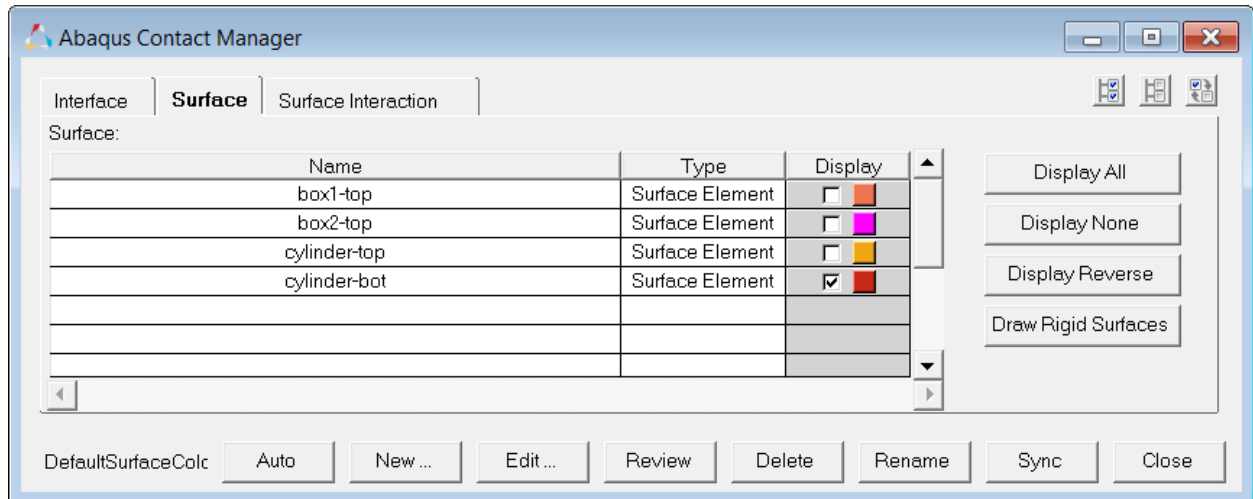
16. The normals of the selected elements will be displayed at this point. If the normals are too big, click  (**YZ Front Plane View**) on the **Standard Views** toolbar.
Notice that all normals are pointing inwards.
17. Check the **Reverse** option.
18. Click **Add** to add these faces to the current surface. HyperMesh creates special face elements (rectangles with dot at the middle) for display.



19. Optional. Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
20. Click the **Adjust Normal** tab.
21. Click **Display normals**. The normals of all the faces in the current surface display.
Notice that all normals are pointing outwards.




22. When satisfied with the surface definition, click **Close** to return to the **Abaqus Contact Manager**.



Steps 7 - 8: Define Surfaces by Set

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers. In this exercise, you will create surfaces by defining a set and corresponding face identifiers. HyperMesh allows only one set in a surface. It also does not support a combination of sets and individual elements in the same *SURFACE data line.

Step 7: Create the box1-bot surface

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to undisplay all surfaces.
3. Click **New**. The **Create New Surface** dialog opens.
4. In the **Name** field, enter `box1-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. Click the **Define** tab.
9. Set **Define surface for** to **Element set**.
10. In the **Model** browser, **Component** folder, right-click on **BOX_1** and select **Isolate** from the context menu.
11. On the **Standard Views** toolbar, click  (**YX Bottom Plane View**).
12. In the **Element Based Surface** dialog, set **Element set** to **box1-bot**.
13. Click **Review Set** to highlight all of the elements in the selected set.
14. Right-click on **Review Set** to reset the highlighting.
15. Click **Show Faces**. HyperMesh creates a temporary skin of the selected element set.

16. Select an element from the bottom of the solid skin.
17. In the panel area, click **elems >> by face**. All of the faces on the bottom of the solid skin highlight.
18. Optional. Deselect any element by right-clicking, or add more if you like.
19. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.

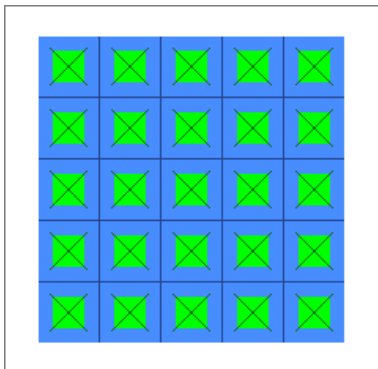
At this point, the face identifier tags (color coded) of the selected faces are displayed. In performance graphics, the solid mesh sometimes blocks these tags. You might have to rotate the model a little to make these tags visible or switch to Standard graphics.

20. Click the right arrow key to move the **box1-bot** set into the table.
21. In the table, click on the **Face** pull down menu and select **S3**.

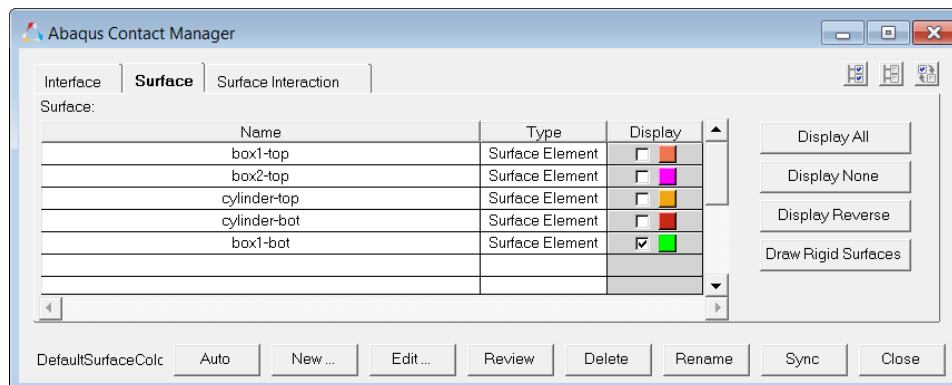
Note: Because all of the face identifier tags for the bottom side of the **box1-bot** set are **S3**, you can use the **S3** identifier for this set.

22. Select the **Display** checkbox, and then click **Update**. HyperMesh adds the selected set and face identifier to the current surface. In addition, it creates a special display for the surface.


By default, HyperMesh does not create a display for surfaces defined with sets. However, if you select the **Display** checkbox before clicking **Update**, it will create a special display using contactsurface elements. The special display does not have any link to the set in the HyperMesh database. Therefore, if you edit the set later on, the display will not reflect them automatically. In this case, you need to come to this page, select the **Display** checkbox and click **Update** again.

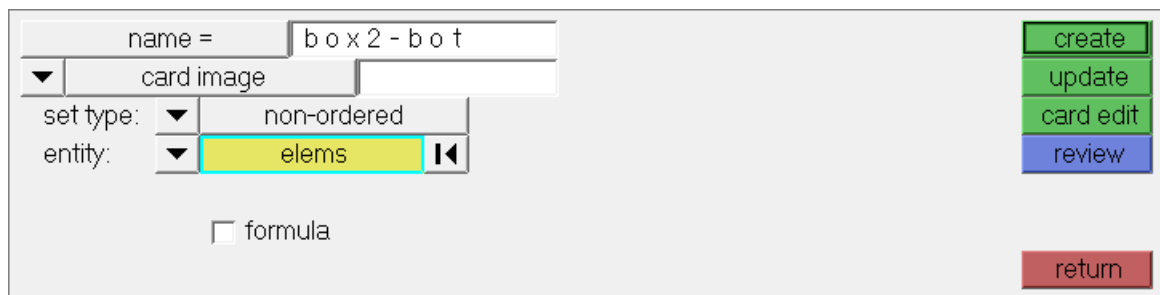


23. Click **Close** to return to the **Abaqus Contact Manager**.



Step 8: Create the box2-bot surface

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to undisplay all surfaces.
3. Click **New**. The **Create New Surface** dialog opens.
4. In the **Name** field, enter `box2-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**. The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. Click the **Define** tab.
9. Set **Define surface for** to **Element set**.
10. In the **Model** browser, **Component** folder, right-click on **BOX_2** and select **Isolate** from the context menu.
11. On the **Standard Views** toolbar, click  (**YX Bottom Plane View**).
12. In the **Element Based Surface** dialog, click **Create/Edit Sets**.
13. In the panel area, enter `box2-bot` in the **name** field.
14. Click **elems >> by collector**.
15. Select the component, **BOX_2**.
16. Click **select**.
17. Click **create**.



18. When you are done creating/editing the set, click **return**.
19. Set **Element set** to **box2-bot**.
20. Click **Review Set** to highlight all of the elements in the selected set.
21. Right-click on **Review Set** to reset the highlighting.
22. Click **Show Faces**. HyperMesh creates a temporary skin of the selected element set.
23. Select an element from the bottom of the solid skin.
24. In the panel area, click **elems >> by face**. All of the faces on the bottom of the solid skin highlight.
25. Optional. Deselect any element by right-clicking, or add more if you like.

26. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.

At this point, the face identifier tags (color coded) of the selected faces display. In performance graphics, the solid mesh sometimes blocks these tags. You might have to rotate the model a little to make these tags visible or switch to standard graphics.

27. Click the right arrow to move the **box2-bot** set into the table.
 28. In the table, click on the **Face** pull down menu and select **S3**.

Note: Because all of the face identifiers tags for the bottom side of the **box2-bot** set are **S3**, you can use the **S3** identifier for this set.

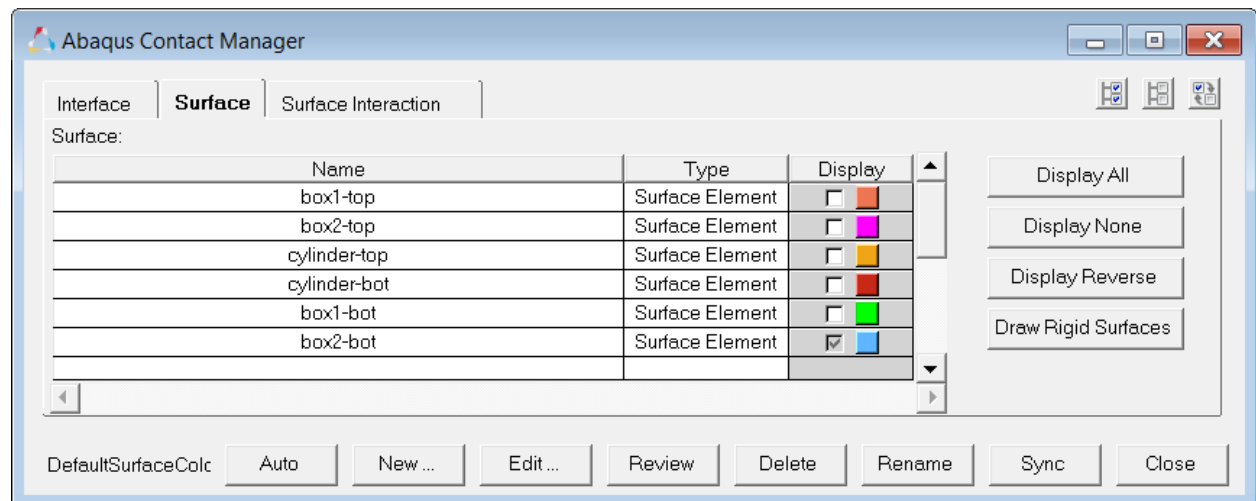
29. Clear the **Display** checkbox if it is selected, then click **Update** to add the selected set and face identifier to the current surface.


Note: By default, HyperMesh does not create any display for surfaces defined with sets.

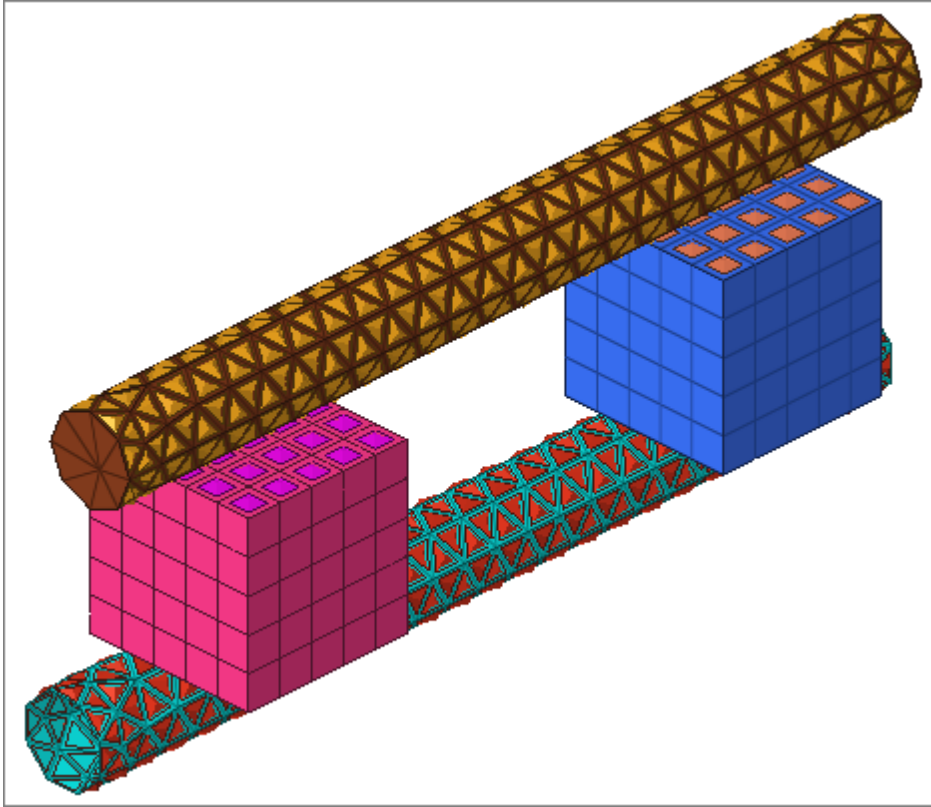
30. Click **Close** to return to the **Abaqus Contact Manager**.

Note: Notice in the **Surface** table that the **Display** option for the **box2-bot** surface is disabled.

At this point, you have created all of the required surfaces.



31. Click **Display All** to display all surfaces.
 32. In the **Model** browser, right-click on the **Components** folder and select **Show** from the context menu. All of the components display.
 33. On the **Standard Views** toolbar, click  (**Isometric**).



34. Review a surface by selecting it from the table and clicking **Review**. Selected surfaces will be highlighted in red, while the rest of the model displays in gray. If the surface is defined with sets (display option disabled), the underlying elements are highlighted. Right-click on **Review** to clear the highlighting.

Step 9: Define the surface interaction property

In this exercise, you will define the *SURFACE INTERACTION card with a corresponding *FRICTION card.

Complete the steps below to create the "friction1" surface interaction.

1. In the **Abaqus Contact Manager**, click the **Surface Interaction** tab.
2. Click **New**. The **Create New Surface Interaction** dialog opens.
3. In the **Name** field, enter `friction1`.
4. Click **Create**. The **Surface Interaction** dialog opens.
5. Click the **Define** tab.
6. Under **Select mechanical interaction properties**, select **Friction**. The **Friction** tab becomes active.
7. Click the **Friction** tab.
8. Set **Friction type** to **Default**.
9. In the second pane, select **Direct**.

Note: Selecting this option means that the exponential decay and Anisotropic parameters will not be written to the input file.

10. In the **No of data lines** field, enter 1. A single row appears in the **Direct** table.

11. In the **Friction Coeff** column, click the first cell and enter 0.05.

For **Direct** and **Anisotropic** tables:

- Change the number of columns in the table by specifying a value in the **No of Dependencics** field; change the number of rows in the table by specifying a value in the **No of data lines** field.
- Enter values in the table by clicking a cell to make it active and then typing in values. The table works like a regular spreadsheet.
- Read comma-delimited data from a text file by clicking **Read From a File**. In the file browser, select a file and click **Open** to export the comma-delimited data. The row number will be set to the number of data lines found in the file.
- Access copy, cut, and paste options by right-click in the table. Comma-separated data can be copied/cut into or pasted from clipboard with these options. Relevant hot keys (for example, Ctrl-C, Ctrl-X and Ctrl-V in Windows) will also work.
- Activate cells by left-clicking in a cell. Clicking into an already active cell moves the insertion cursor to the character nearest the mouse.
- Highlight cells by left-clicking while moving the mouse over a cell.
- Move the active cell using the left, right, up, and down arrows.
- Extend the selection in a specific direction using SHIFT-<arrow>.
- Move the insertion cursor within a cell using CTRL-left arrow and CTRL -right arrow.
- Selects all cells using CTRL -slash.
- Delete the character before the insertion cursor in the active cell using BACKSPACE. If multiple cells are selected, BACKSPACE deletes all selected cells.
- Remove the character after the insertion cursor in the active cell using DELETE. If multiple cells are selected, DELETE removes all selected cells.
- Move the insertion cursor to the beginning of the active cell using CTRL-A. Move the insertion cursor to the end of the active cell using CTRL-E.
- Decrease and increase the width of the column with the active cell in it using CTRL-minus (-) and CTRL-equal (=).
- Interactively resize a row or column by left-clicking or right-clicking on a border while moving the mouse.

11. Click **OK** to return to the **Abaqus Contact Manager**.

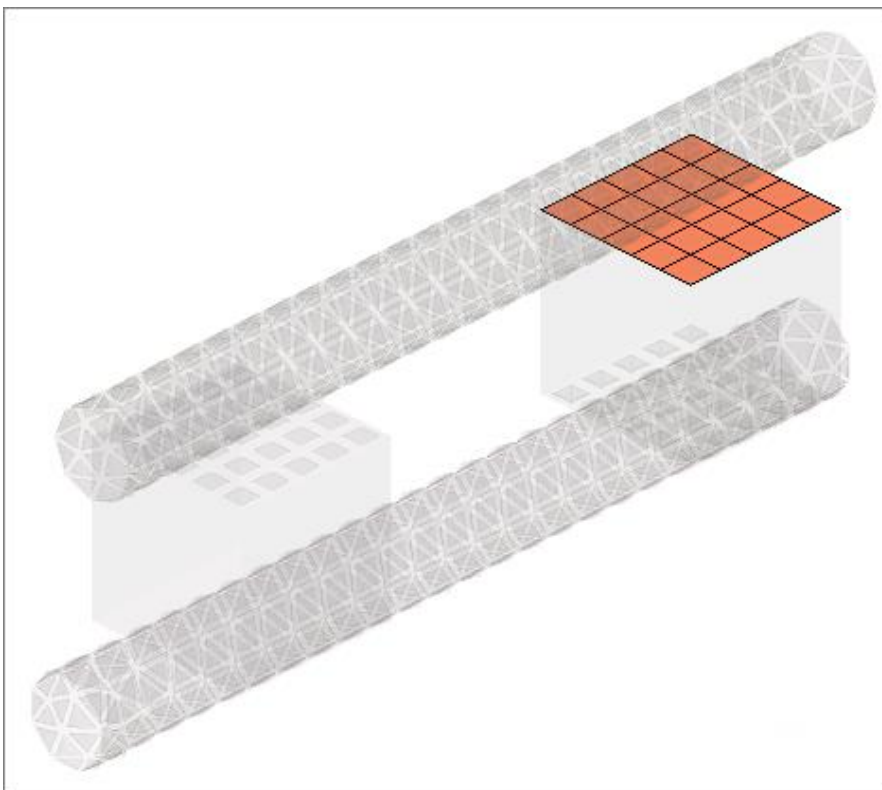
Steps 10-13: Define the Contact Pairs

In this exercise, you will define the *CONTACT PAIR card with corresponding surfaces and surface interaction.


Step 10: Create the top-cylinder-box1 contact pair

1. In the **Abaqus Contact Manager**, click the **Interface** tab.
2. Click **New**. The **Create New Interface** dialog opens.
3. In the **Name** field, enter `top-cylinder-box1`.
4. Set **Type** to **Contact pair**.
5. Click **Create**. The **Contact Pair** dialog opens.
6. Click the **Define** tab.
7. Set **Surface** to **box1-top**.
8. Click **Slave>>** to identify box1-top as the slave surface and move it into the table.
9. Click **Review**. The selected surface highlights red. If the surface is defined with sets (display option disabled), the underlying elements highlight. Right-click on **Review** to clear the highlighting.


Clicking **New** opens the **Create New Surface** dialog, from which you can create a new surface. When you are done creating and defining the surface, the **Contact Pair** dialog returns with the new surface selected as the slave surface.



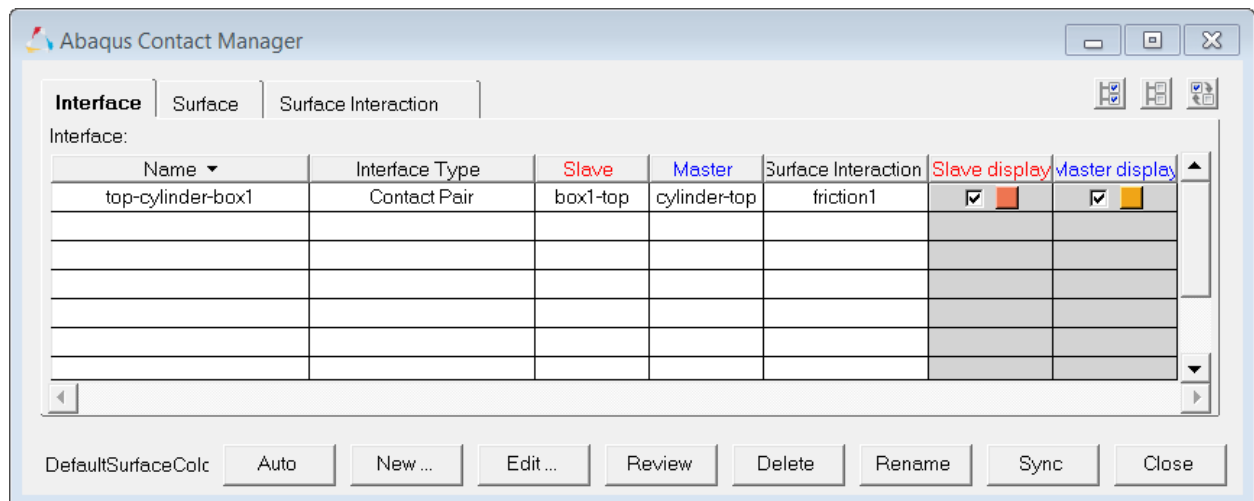
10. Repeat steps 10.7 and 10.8, selecting **cylinder-top** and clicking **Master>>** to identify it as the master surface.

Note: To more clearly see the surfaces available for selection, click . This opens an **enhanced** browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

11. Set **Interaction** to **friction1**.

Note: To more clearly see the interactions available for selection, click . This opens an **enhanced** browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

12. Click the **Parameter** tab.
 13. Select the **Small sliding** checkbox.
 14. Click **OK** to return to the **Abaqus Contact Manager**.



Step 11: Create the top-cylinder-box2 contact pair

Follow step 10.1 through 10.14 above to define the **top-cylinder-box2** contact pair with **box2-top** as slave surface, **cylinder-top** as master and **friction1** as the surface interaction.

Step 12: Create the bot-cylinder-box1 contact pair

Follow step 10.1 through 10.14 above to define the **bot-cylinder-box1** contact pair with **box1-bot** as slave surface, **cylinder-bot** as master and **friction1** as the surface interaction.

Step 13: Create the bot-cylinder-box2 contact pair

Follow steps 10.1 through 10.14 above to define the **bot-cylinder-box2** contact pair with **box2-bot** as slave surface, **cylinder-bot** as master and **friction1** as the surface interaction.

At this point, you have created all of the contact pairs required. Review any contact pair by selecting it from the table and clicking **Review**. Both the master and slave surface highlight in red while the rest of the model is grey. If a surface is defined with sets (display option disabled), the underlying elements highlight. Right-click on **Review** to clear the highlighting.

Click **Close** to close the **Abaqus Contact Manager**.

General comments:

- Click **Edit** to open the dialog for editing the selected interface, surface, or surface interaction
- Click **Delete** to remove the selected interfaces, surfaces, or surface interactions. Multiple selections can be removed from the **Interface** table at once.
- Click **Sync** to update the **Contact Manager** with the current HyperMesh database. If you create, update, or delete any components, groups, properties, or entity sets from HyperMesh panels while the **Contact Manager** is open, click **Sync** to update the **Contact Manager**.
- If you minimize the **Contact Manager** dialog or if it goes behind HyperMesh, click **Tools > Contact Manager** to restore it.
- Bubble help exists for important buttons. Place the mouse on the buttons for a few moments to view it.
- Double-click on interface, surface, and surface interaction names in the table to open the corresponding edit dialog. Right-click on these names to display a pull down menu with options.
- Left-click or right-click on a table border while moving the mouse can resize columns in a table.
- SHIFT and CTRL keys can be used while left-clicking to select multiple items in a table (useful for deleting multiple items).