



Altair

HyperWorks

HM-4420: Defining ANSYS Contacts for 2-D Models in HyperMesh

Before you begin this tutorial, it is recommended that you complete [HM-1000: Getting Started with HyperMesh](#) and [Exploring the ANSYS Interface](#).

In this tutorial you will learn how to set up edge to edge contacts. In HyperMesh, you can create contact pairs manually or with the Autocontact tool. In this exercise you will learn how to set up contacts manually. You will create edge to edge contacts between circular and rectangular parts of the model. The edge of the circular body acts as a Target (slave) surface, and the top edge of the rectangular part acts as a contact (master) surface.

The steps mentioned in this tutorial are only applicable for HyperMesh 14.0.130 and on.

Model Files


This exercise uses the `hm-ansys_contact_manager_2-d_tutorial.hm` file, which can be found in `<hm.zip>/interfaces/ansys/`. Copy the file(s) from this directory to your working directory.

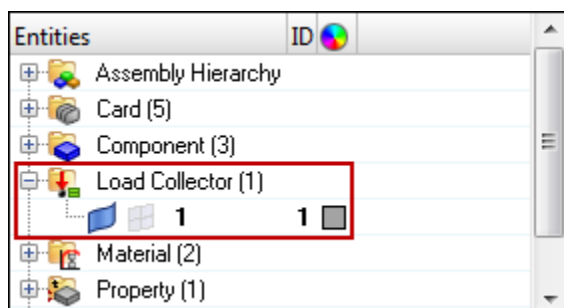
Exercise

Step 1: Load the ANSYS User Profile

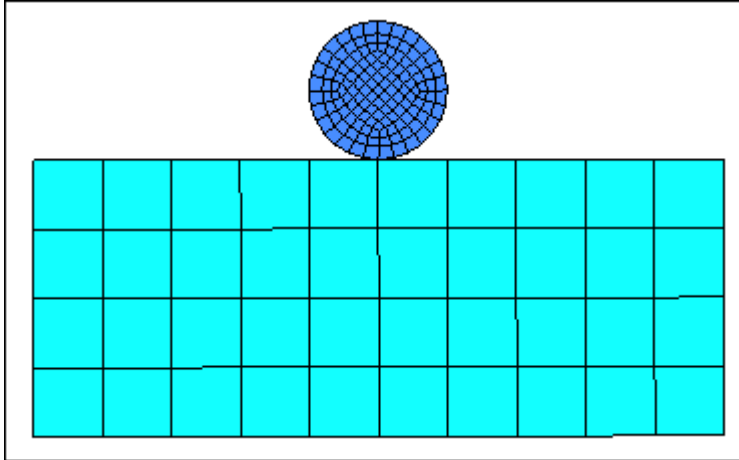
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

Step 2: Retrieve the Model File

1. From the menu bar, click **File > Open > Model**.
2. In the **Open Model** dialog, open the `hm-ansys_contact_manager_2-d_tutorial.hm` file. A model loads in the graphics area.
3. If the load collector is displayed, in the **Model** browser, click  next to the load collector to turn off the display of its element.



4. In the graphics area, click once to make it the current window for keyboard commands.
5. Fit the model to the graphics area by pressing **f**.

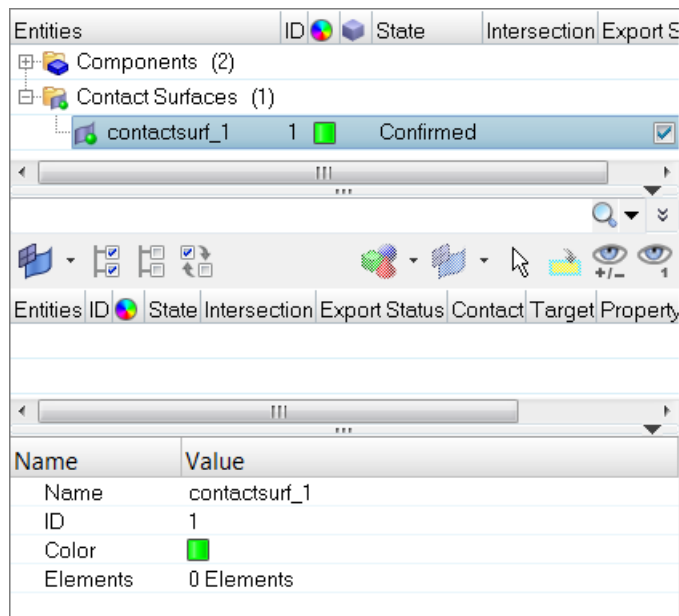


Step 3: Open the Contact Browser



1. From the menu bar, click **View > Browsers > HyperMesh > Contact**.

Step 4: Create Target (Slave) Surface

1. In the first pane of the **Contact** browser, right-click and select **Create > Contact Surfaces** from the context menu. HyperMesh creates and opens a contact surface in the **Entity Editor**.

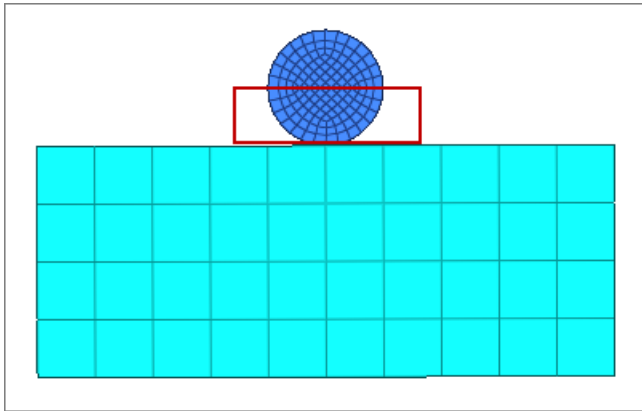


2. In the **Entity Editor**, enter a name and ID, and select a color for the contact surface.
3. For **Elements**, click **0 Elements >> Elements**.

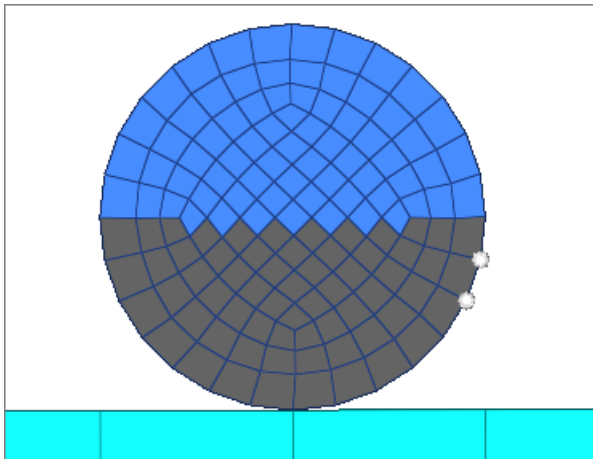
Name	Value
Name	contactsurf_1
ID	1
Color	
Elements	Elements 

4. In the panel area, set the first selector to **add shell edges**.
5. Select **reverse normals**.
6. For **face angle**, enter 30.0.
7. Set the entity selector to **elems**, then select the free edges indicated in the image below.

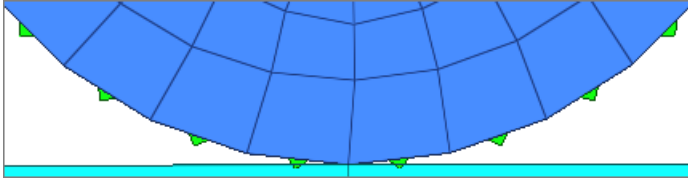
Tip: Quickly select elements with window selection by pressing **SHIFT** while clicking and dragging your mouse.



8. Using the **nodes** selector, select two nodes along the free edges you have selected to identify the edge on which contact surface are to be created.



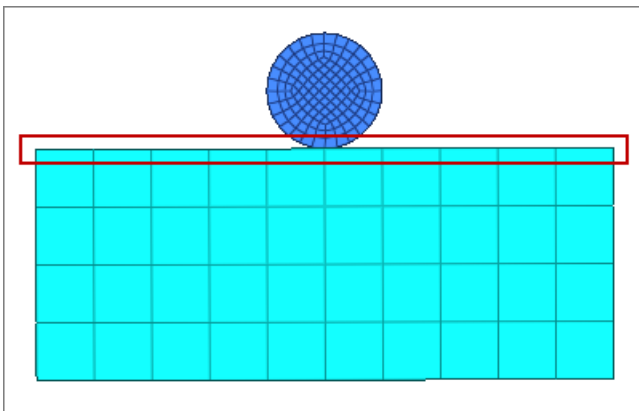
9. Click **Add**. All of the edges of the selected elements are added to the surface.



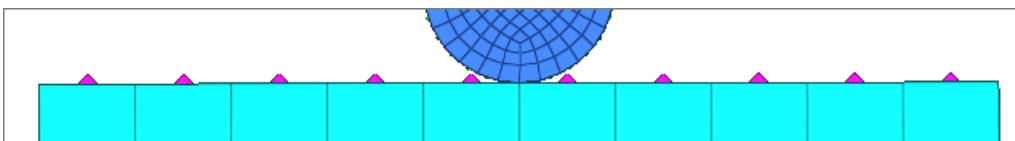
10. Click **return** to exit the panel.
11. Review the contact surface you just created.
 - a. In the **Contact** browser, right-click on **contactsurf_1** and select **Review** from the context menu.
 - b. Isolate the contact surface for a better visual display. The surface created here will be considered the target (slave) surface during analysis.

Step 5: Create Contact (Master) Surface

1. In the first pane of the **Contact** browser, right-click and select **Create > Contact Surfaces** from the context menu. HyperMesh creates and opens a contact surface in the **Entity Editor**.
2. In the **Entity Editor**, enter a name and ID, and select a color for the contact surface.
3. For **Elements**, click **0 Elements >> Elements**.
4. In the panel area, set the first selector to **add shell edges**.
5. For **face angle**, enter 30.0.
6. Select **reverse normals**.
7. Set the entity selector to **free edges**, then select the top free edges of the rectangular box as indicates in the image below.



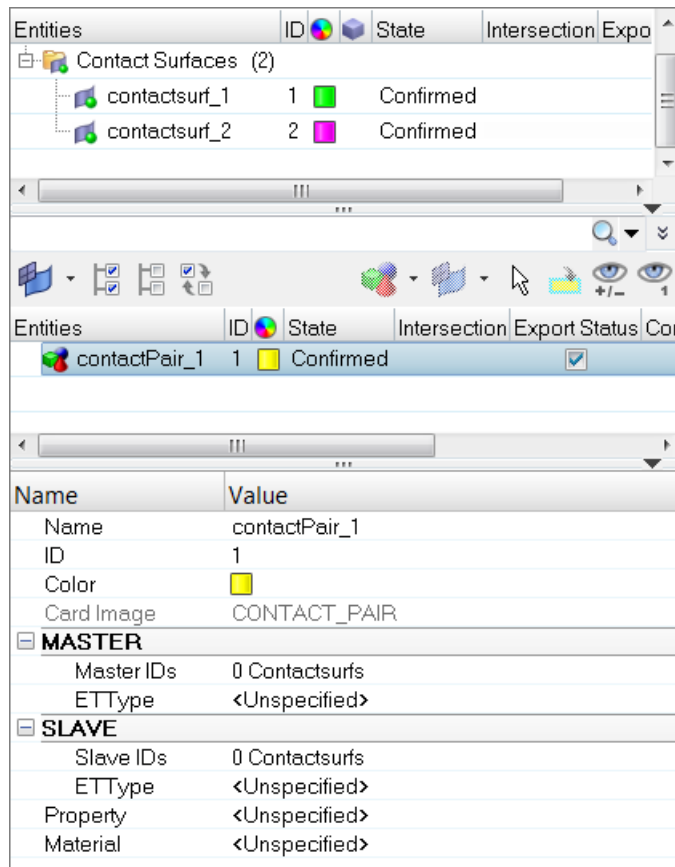
8. Click **add**. Surfaces are created on the top edge of the rectangular box.



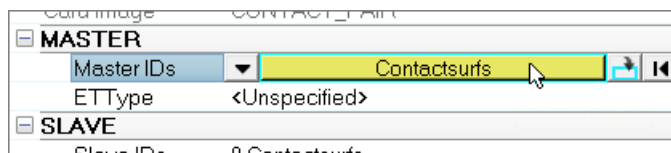
- Click **return** to exit the panel.

Step 6: Create a Contact Pair

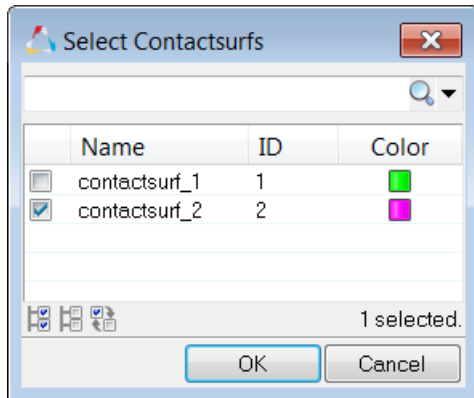
- In the second pane of the **Contact** browser, right-click and select **Create > Contact Pair** from the context menu. HyperMesh creates and opens a contact pair in the **Entity Editor**.



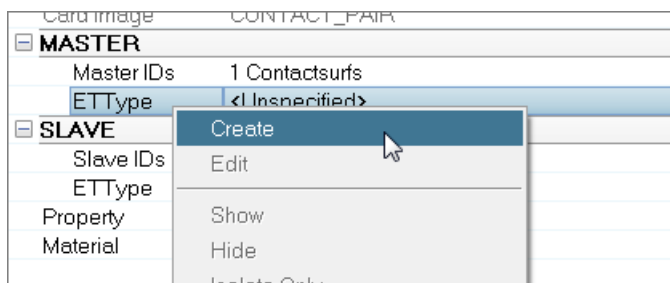
- In the **Entity Editor**, enter a name and ID, and select a color for the contact pair.
- Attach the contact (master) surface.
 - For **Master IDs**, click **0 Contactsurfs >> Contactsurfs**.



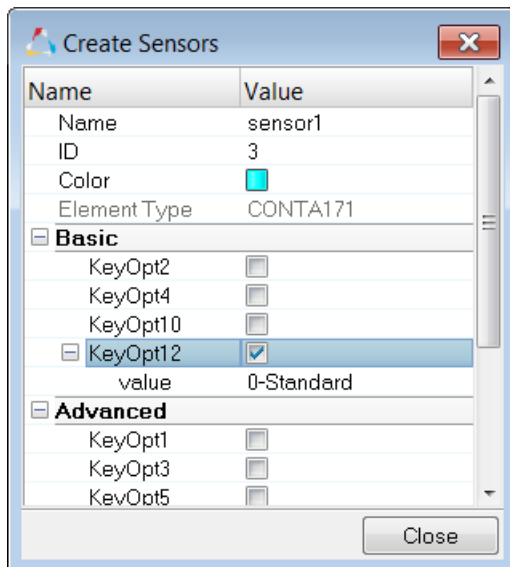
- In the **Select Contactsurfs** dialog, select **contactsurf_2** and click **OK**.



- c. Under **MASTER**, right-click on **ETType**, and select **Create** from the context menu.



- d. In the **Create Sensors** dialog, define the sensor and click **Close**.
- By default, the **Element Type** is set to **CONTA171**.
 - Enable **KeyOpt12**, then select **0-Standard** from the list. Keyopts determine the behavior of contacts during analysis. KeyOpt12 defines the type of contact.



4. Attach the target (slave) surface.
 - a. For **Slave IDs**, click **0 Contactsurfs >> Contactsurfs**.
 - b. In the **Select Contactsurfs** dialog, select **contactsurf_1** and click **OK**.


In this contact pair, contactsurf_1 acts as the target (slave) surface. This surface will be exported with the element type Targe169.
 - c. Under **SLAVE**, right-click on **ETType** and select **Create** from the context menu.
 - d. In the **Create Sensors** dialog, define the sensor and click **Close**.
 - By default, the **Element Type** is set to **Targe169**.
 - Enable **KeyOpt4**, then select **111-ROTZ, UY, UX** from the list.

The target (slave) surface will be exported with the elements type TARGE169.
5. Create and attach the contact property.

A contact property for a pair is always required, as the solver recognizes the pair only by the property ID associated with the master and slave surfaces. Same property IDs are shared by the master and slave surfaces. If a property ID is missing for a contact surface, then it will not be considered as part of the contact pair and it will not be recognized.

- a. In the **Entity Editor**, right-click on **Property** and select **Create** from the context menu.
 - b. In the **Create Properties** dialog, define the property and click **Close**.
 - For **FKN**, enter 1.0.
 - For **FTOLN**, enter 0.1.
 - For **TAUMAX**, enter 1e20.
 - For **FKOP**, enter 1.0.
 - For **FKT**, enter 1.0.
 - For **FHTG**, enter 1.0.
 - For **RDVF**, enter 1.0.
 - For **FWGT**, enter 0.5.
 - For **FACT**, enter 1.0.
6. Create and attach the contact material.

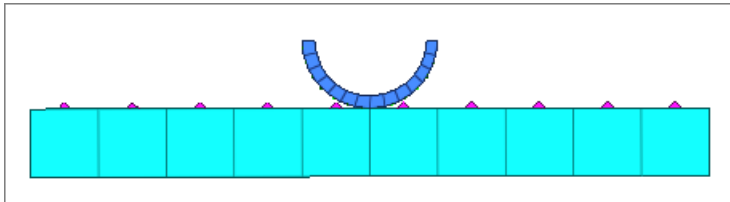
The properties MU (coefficient of friction) and EMIS (Thermal Emissivity) are available to define. If you are creating a structural contact, select MU; if you are creating a thermal contact, select EMIS. In this exercise you will be creating a structural contact.

- a. In the **Entity Editor**, right-click on **Material** and select **Create** from the context menu.
- b. In the **Create Materials** dialog, define the material and click **Close**.
 - Select the **MU** (coefficient of friction) checkbox.
 - For **MP_MU_LEN**, enter 1.
 - Under **MP_MU_LEN=**, next to **Data: C**, click .

- In the **MP_MU_LEN=** dialog, enter 0.45 and click **Close**.

Step 7: Review the Contact Pair

1. In the second pane of the **Contact** browser, select the contact pair to display its entity details in the **Entity Editor**.
2. Right-click on the contact pair and select **Isolate Attached > Elements** to review the contact area.



Step 8 (Optional): Add /SOLU, ANTYPE, and SOLVE in the Control Cards

If your model needs to be solved in ANSYS, add the following control cards.

1. Open the **Control Cards** panel by clicking **Setup > Create > Control Cards** from the menu bar.
2. Click **/SOLU** to exit the **PREP7** preprocessor and enter the **SOLU** preprocessor.

AUTOTS	ETABLE	OUTRES	MODOPT	LUMPM
/BATCH	KBC	/POST1	MXPAND	ACEL
BFUNIF	LNSRCH	PRESOL	EQSLV	CGLOC
/COM	MODE	RSYS	ALPHAD	CGOMGA
CNVTOL	NEQIT	/SOLU	BETAD	CMDOMEGA
DELTIM	NLGEOM	SOLU	PSTRES	CMOMEGA
DOF	NSUBST	ANTYPE	EXPASS	DCGOMG

```
! Exit PREP7 processor
FINISH
! Enter SOLU processor
/SOLU
```

3. Click **return**.
4. Since you are solving the model for static analysis, click **ANTYPE**.

AUTOTS	ETABLE	OUTRES	MODOPT	LUMPM
/BATCH	KBC	/POST1	MXPAND	ACEL
BFUNIF	LNSRCH	PRESOL	EQSLV	CGLOC
/COM	MODE	RSYS	ALPHAD	CGOMGA
CNVTOL	NEQIT	/SOLU	BETAD	CMDOMEGA
DELTIM	NLGEOM	SOLU	PSTRES	CMOMEGA
DOF	NSUBST	ANTYPE	EXPASS	DCGOMG

5. Set **type** to **STATIC** and **status** to **NEW**.

ANTYPE	type STATIC	status NEW
User Comments		reject
▼ Hide In Menu/Export		default
		abort
		return

6. Click **return**.
7. Click **SOLVE**.

Note: If you do not see the **SOLVE** control card, click **next**.

SUBOPT	SOLVE	UNSU_PREP_END
EMUNIT	LSSOLVE	
EORIENT	STEF	
UNSU_PREP7	HEMIOPT	
UNSU_END	RADOPT	
/UNITS	SPCTEMP	
NUMOFF	UNSU_PREP_MID	

→

SOLVE
User Comments
▼ Hide In Menu/Export

8. Click **return** to exit the card image.
9. Click **return** to exit the **Control Cards** panel.