Altair

HyperWorks

HM-4440: Introduction to the Contact Manager

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. For this tutorial, you will learn how to set up contacts manually. You will create edge to edge contacts between two flat plates. One of the plates will act as a sliding plate (Contact edge), and the other will act as a stationary (Target edge).

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

In this tutorial, you will:

- Load the ANSYS user profile.
- Retrieve the HyperMesh model for this tutorial.
- Create a constraints load collector.
- Create edge to edge contacts.
- Apply the constraints to the model.
- Apply displacement to the two end nodes of the component "contact".
- Activate control cards for Nonlinear Analysis Solution Control.
- Export the deck to ANSYS *.cdb format.

Model Files

This exercise uses the chapter2_3.hm file, which can be found in <hm.zip>/interfaces/ansys/. Copy the file(s) from this directory to your working directory.
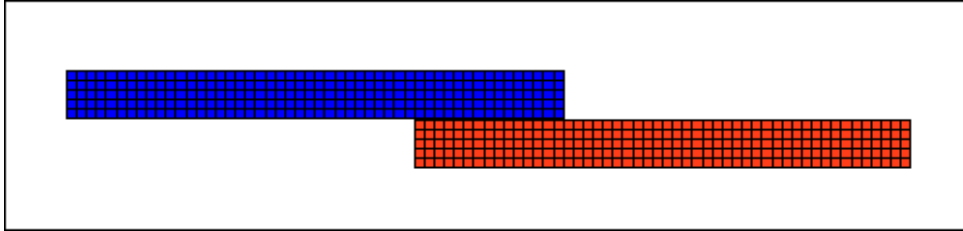
Exercise

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 HyperMesh Model for this Tutorial

1. From the menu bar, click *File* > *Open* > *Model*.
2. In the **Open Model** dialog, open the chapter2_3.hm file. A model loads in the graphics area.
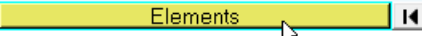
Step 3: Open the Contact Browser

1. Open the **Contact** browser from the menu bar by clicking **View** > **Browsers** > **HyperMesh** > **Contact**.

2. In the first pane of the browser, mask the **target** component to enable better selection of contact surfaces on the **contact** component.
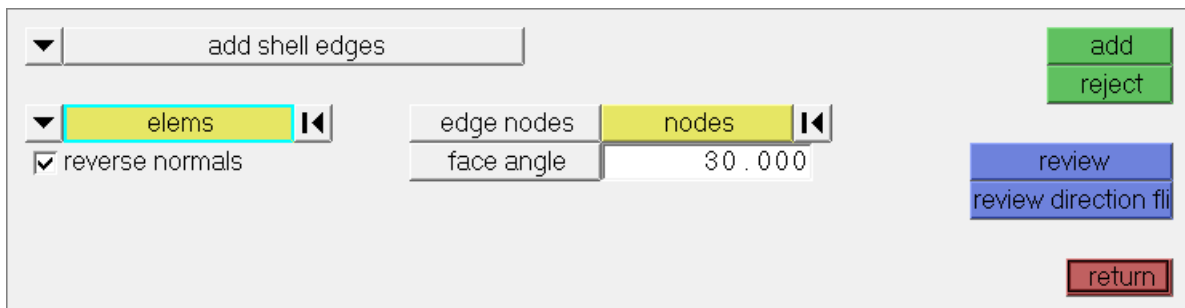
Step 4: Create Contact (Master) Surface

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**.
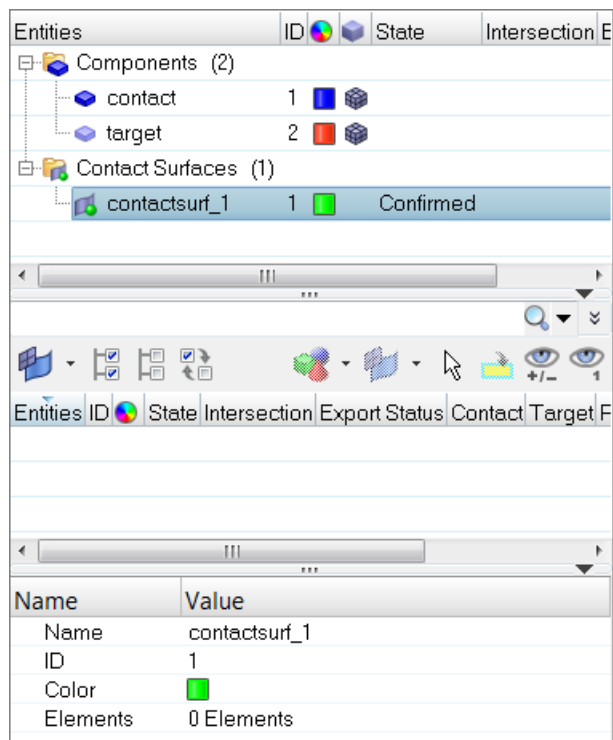
2. In the **Entity Editor**, for **Name**, enter `slidingsurface`.

3. For **Elements**, click **0 Elements** >> **Elements**.

Name	Value
Name	slidingsurface
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`.

▼	add shell edges			add
				reject

| ▼	elems	◄	edge nodes	nodes	◄		review
☑ reverse normals			face angle	30.000			review direction fli

return

7. Set advanced selection 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.
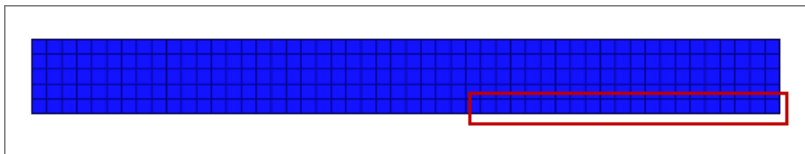
8. Using the **nodes** selector, select a couple nodes along the free edge you have selected to identify the edge on which contact surface are to be created.
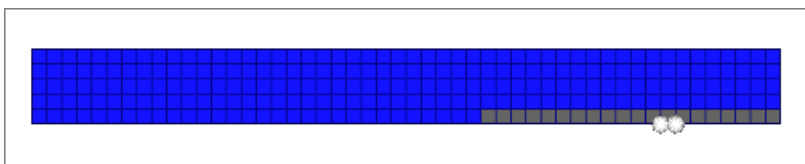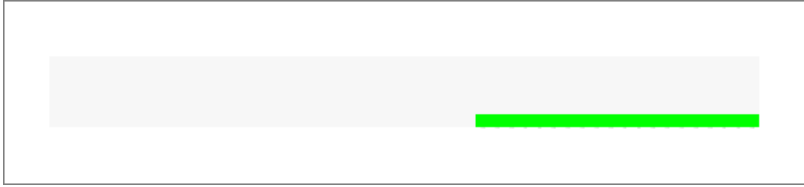
9. Click ***add***. All of the edges of the selected elements are added to the surface.

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

Altair
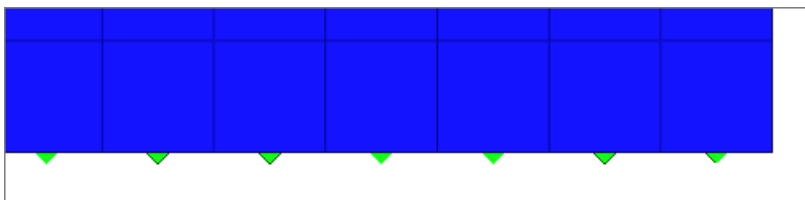
11. In the **Contact** browser, right-click on *slidingsurface* and select *Review* from the context menu. Review the contact surface you just created.
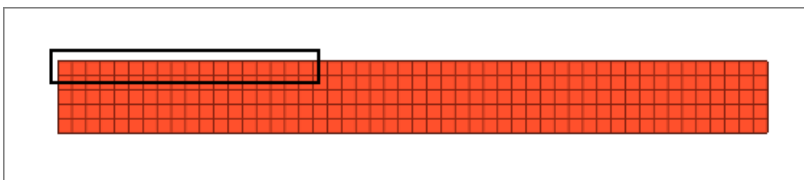

12. Isolate the contact surface for a better visual display. The surface created here will be considered master (contact) surfaces during analysis.

Step 5: Create Target (Slave) Surface

1. Unmask the **target** component, and mask the **contact** component and **slidingsurface** contact surface.

2. 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**.

3. In the **Entity Editor**, for **Name**, enter `targetsurface`.

4. For **Elements**, click *0 Elements* >> *Elements*.

5. In the panel area, set the first selector to *add shell edges*.

6. Select *reverse normals*.

7. For **face angle**, enter `30.0`.

8. Set advanced selection to *elems*, then select the free edges indicated in the image below.
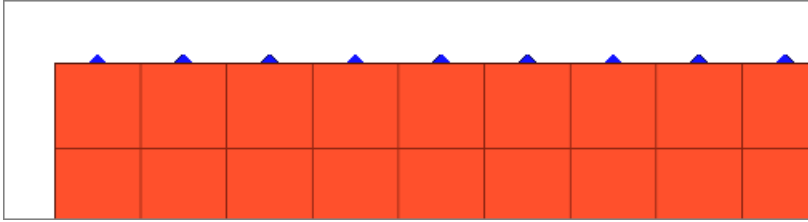
    Tip: Quickly select elements with window selection by pressing *SHIFT* while clicking and dragging your mouse.

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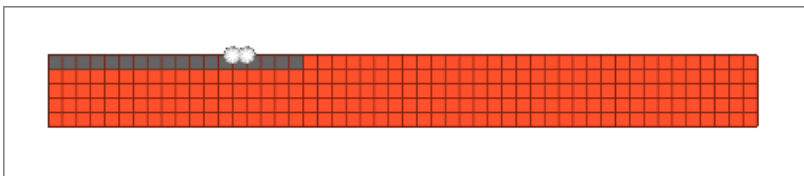
10. Click *add*. All of the edges of the selected elements are added to the surface.

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

12. In the **Contact** browser, right-click on **targetsurface** and select **Review** from the context menu. Review the contact surface you just created.

13. Isolate the contact surface for a better visual display. The surface created here will be considered slave (target) surfaces during analysis.

Step 6: Create a Contact Pair

1. 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**.
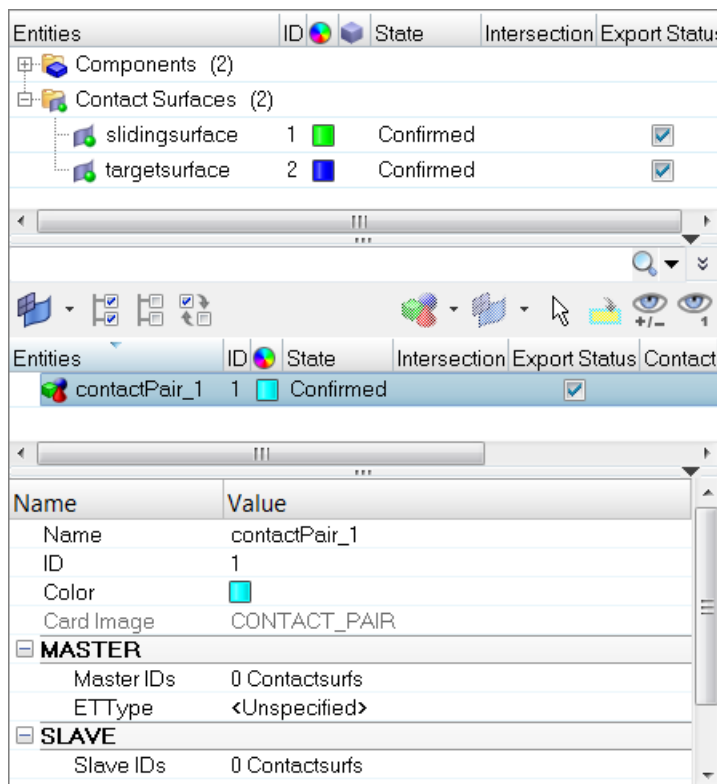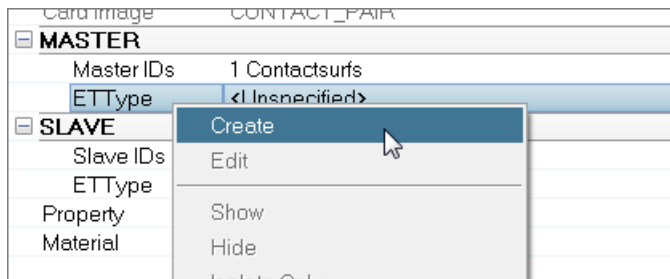
2. In the **Entity Editor**, enter a name and ID, and select a color for the contact pair.

3. Attach the contact (master) surface.

   a. For **Master IDs**, click *0 Contactsurfs* >> *Contactsurfs*.

   b. In the **Select Contactsurfs** dialog, select *slidingsurface* and click **OK**.
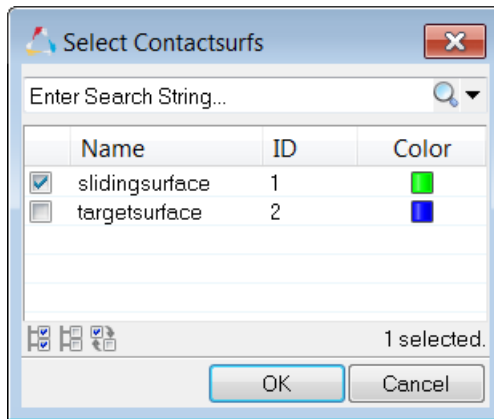
   c. Under **MASTER**, right-click on *ETType*, and select *Create* from the context menu.

   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.

△ Altair

4. Attach the target (slave) surface.

   a. For **Slave IDs**, click *0 Contactsurfs* >> *Contactsurfs*.
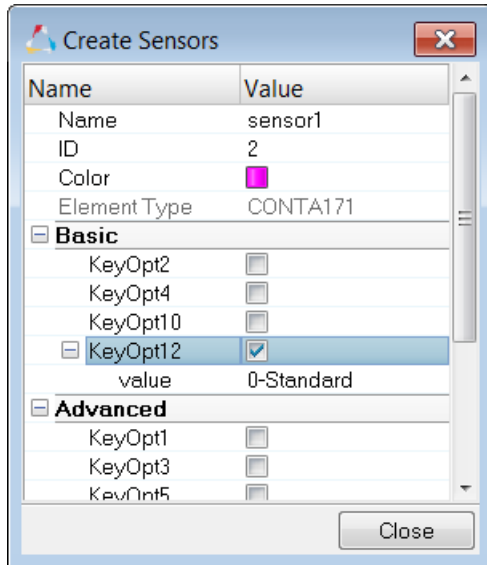
   b. In the **Select Contactsurfs** dialog, select *targetsurface* and click *OK*.

   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 considered as part of the contact pair and it will not recognized.

   a. 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. 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 *Review* from the context menu. The contact area displays.
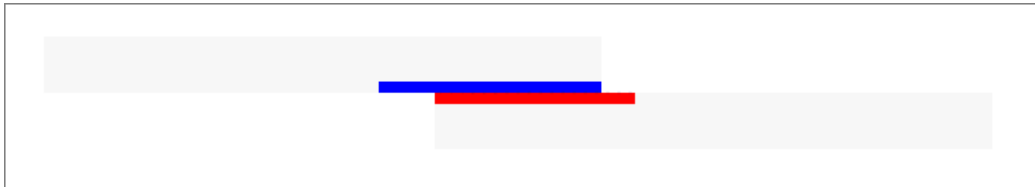
3. Right-click on the contact pair again and select *Reset Review* from the context menu.

## Step 8: Create a Constraints Load Collector

1. In the **Model** browser, right-click and select *Create* > *Load Collector* from the context menu. HyperMesh creates and opens a load collector in the **Entity Editor**.

2. For **Name**, enter `Constraints`.

3. Click the *Color* icon, and select a color for the load collector.

4. You do not need to assign a card image for this load collector.
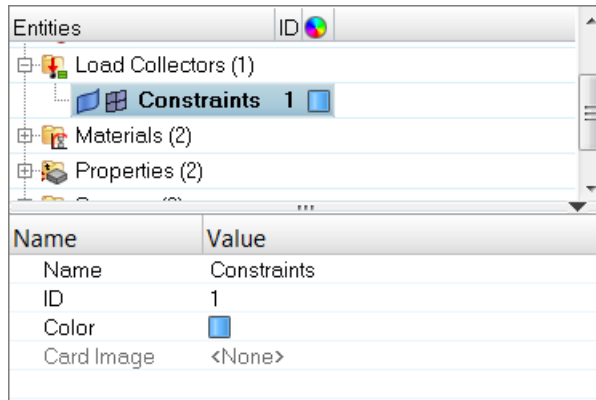
Step 9: Apply Constraints to the Model

1. Open the **Constraints** panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.

2. Select all of the **dof** (degree of freedom) checkboxes.
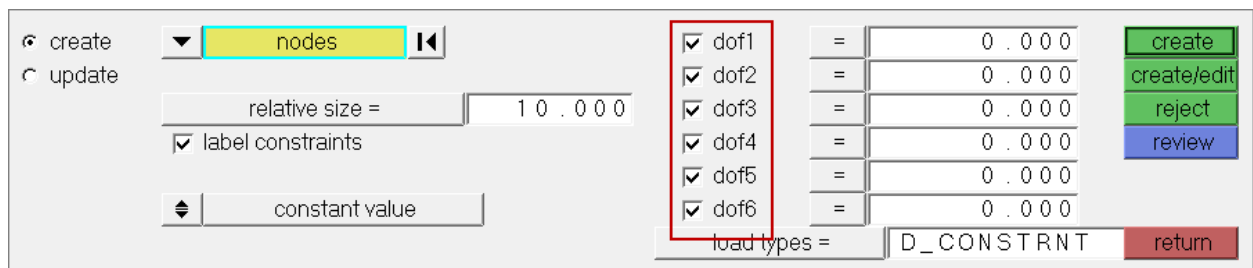
3. Set the entity selector to **nodes**.

4. Select the nodes on the left end of the contact component and on the right end of the target component as indicated in the following image.
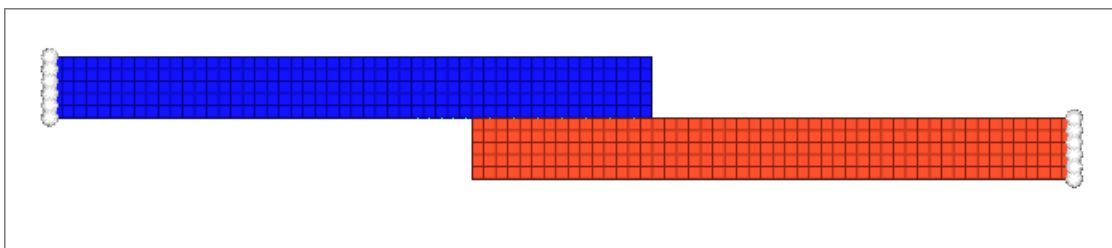
5. In the **relative size=** field, enter 2.

6. Click **create**.

Step 10: Apply a Displacement Constraint to the Two Mid-Nodes of the Component Contact

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

1. Select the two mid nodes at the end of the contact component as indicated in the following image.

2. Clear all of the **dof** checkboxes except for **dof2**.

3. In the **dof2=** field, enter −5.0 for the displacement value of the selected mid nodes at the component contact in the global −Y direction.

4. Click **create**.
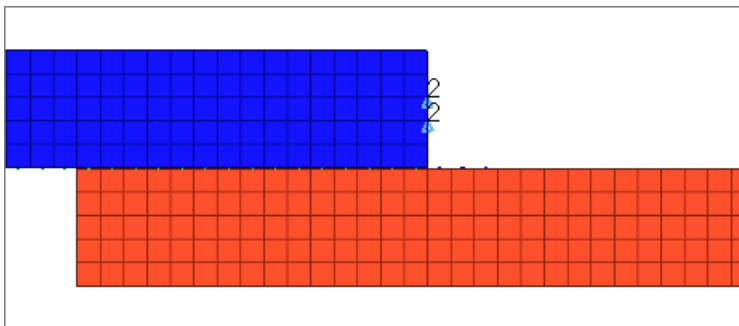
5. Click **return** to exit the **Constraints** panel.
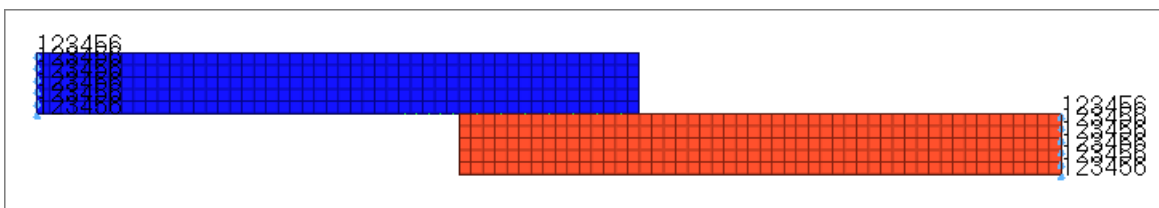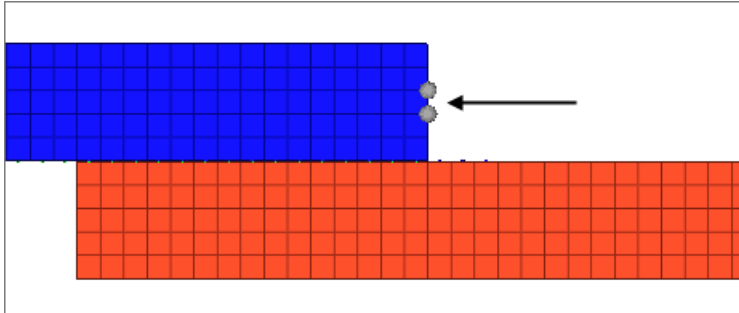
Step 11: Activate Control Cards for Nonlinear Analysis Solution Control

1. Open the **Control Cards** panel by clicking **Setup** > **Create** > **Control Cards** from the menu bar.

2. Specify the analysis type and restart the status by clicking **ANTYPE**.

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

	type	status
ANTYPE	STATIC	NEW

User Comments

▼   Hide In Menu/Export

reject
default

abort
return

4. Click *return* to go back to the control cards page.

5. Activate a line search to be used with Newton-Raphson by clicking *LNSRCH*.

6. Under **Key**, click *OFF* and then select *ON* from the list.

	Key
LNSRCH	ON

User Comments

▼   Hide In Menu/Export

reject
default

abort
return

7. Click *return*.

8. To include large-deflection effects in a static or full transient analysis, click *NLGEOM*.

9. Under **Key**, click *OFF* and then select *ON* from the list.

	Key
NLGEOM	ON

User Comments

▼   Hide In Menu/Export

reject
default

abort
return

10. Click *return*.

11. Specify the number of sub steps to be taken for this load step by clicking *NSUBST*.

12. Click *[NSBSTP]* and enter `100` in the editable field.

　　Note: NSBSTP is the number of sub steps to be used for this load step.

13. Click *[NSBMX]* and enter `1000` in the editable field.

> **Note**: NSBMX is the maximum number of sub steps to be taken (minimum time step size) if automatic time stepping is used.

14. Click **[NSBMN]** and enter `20` in the editable field.

> **Note**: NSBMN is the minimum number of sub steps to be taken (maximum time step size) if automatic time stepping is used.

15. Verify that **Carry** is set to *OFF*.

	[NSBSTP]	[NSBMX]	[NSBMN]	Carry
NSUBST	100	1000	20	OFF

User Comments
▼ | Hide In Menu/Export

reject
default

abort
return

16. Click **return**.

17. Exit the **PREP7 preprocessor** and enter the **SOLU preprocessor** by clicking **/SOLU**.

```
! Exit PREP7 processor

FINISH

! Enter SOLU processor

/SOLU
```

18. Click **return**.

19. Solve the model by clicking **SOLVE**.

SOLVE

User Comments
▼ | Hide In Menu/Export

reject
default

abort
return

20. Click **return**.

> **Note**: When the color of the control card button is green, the card exists in the database and will be written when the **Export Data** panel is used with the current template.

△ Altair

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

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

21. Click *return*.

22. Save your model by clicking *File* > *Save* > *Model* from the menu bar.

Step 12: Export the Model

HyperMesh can export ANSYS model data in script format. The following steps explain the process of exporting an ANSYS model from HyperMesh.

1. Open the **Export** tab by clicking *File* > *Export* > *Solver Deck* from the menu bar.

2. Set **File type** to *Ansys*.

   Note: If you are in the **ANSYS** user profile, HyperMesh will automatically set the **File type** to **Ansys**.

3. In the **File** field, navigate to your working directory and save the file as `exercise2b.cdb`.

4. Click *Export*.

△ Altair