Altair

HyperWorks

HM-4615: Model Importing, Airbags, Exporting Displayed, and Contacts using DYNA

In this tutorial, you will learn how to:

- Define *AIRBAG_WANG_NEFSKE for the airbag mesh geometry

- Define an initial velocity of 3 mm/ms in the negative x-direction for the head with *INITIAL_VELOCITY_GENERATION

- Define a contact between the airbag and head with *ICONTACT_AUTOMATIC_SURFACE_TO_SURFACE

- Define *CONTACT_AIRBAG_SINGLE_SURFACE for the airbag

- Define a contact between the plate and the airbag with *CONTACT_NODES_TO_SURFACE

Import a LS-DYNA model

Warning and Error Messages

When importing a LS-DYNA model, HyperMesh warning and error messages will be written to a file named `dynakey.msg` or `dynaseq.msg`, depending on the FE input translator being used. This file is created in the same folder from which HyperMesh was started.

Unsupported Cards

On import, the LS-DYNA cards not supported by HyperMesh are written to the **unsupp_cards** panel. Access this panel by clicking ***Setup*** > ***Create*** > ***Control Cards*** from the menu bar. Unsupported cards will be exported with the remaining model.

Care should be taken if an unsupported card points to an entity in HyperMesh. An example of this is when an unsupported material references a *PART. HyperMesh stores unsupported cards as text and does not consider pointers.

LSTC Dummy Files

You can read LSTC Hybrid III dummy files into HyperMesh by first converting the tree file to FTSS/ARUP tree file format.

Include Files

HyperMesh supports *INCLUDE. When Include files are imported into HyperMesh, the IDs of non-existing entities are maintained and will not be used for new entities.

Use the **Include files** import option to specify whether to merge, preserve, or skip Include files on import. Access this option by clicking ***File*** > ***Import*** > ***Solver Deck*** from the menu bar.

Support of Include files:

- The LSDYNA keywords *INCLUDE, *INCLUDE_STAMPED_PART, *INCLUDE_STAMPED_PART_SET, *INCLUDE_TRANSFORM, and *INCLUDE_COMPENSATION_OPTION are mapped to Include files.

△ Altair

- Switch to different types of Include files using the context menu Include File option in the Model browser, with the exception of *INCLUDE_TRANSFORM, *INCLUDE_STAMPED_PART, and *INCLUDE_STAMPED_PART_SET.

- Manage INCLUDE_TRANSFORM using the Transformation Manager.

- During import, if the same Include file is referenced more than once using the *INCLUDE_TRANSFORM, then it will be imported, but appended with .#, where # = 1…n is shown in the Model browser. These will not be exported unless you clear the **Instance** checkbox.

- During import, *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET are imported as "read only" by default to preserve the associativity.

- During import, an Include file can be read when *INCLUDE comes after *INCLUDE_TRANSFORM.

- When importing Include files, use **Solver options** in the **Import** browser to choose the type of Include file to import. This option enables you to set the Include file type to one of the following: INCLUDE, INCLUDE_STAMPED_PART, INCLUDE_STAMPED_PART_SET, or INCLUDE_COMPENSATION_options. This is the only option which enanables you to attach any file of type INCLUDE_STAMPED_PART or INCLUDE_STAMPED_PART_SET.

Export Displayed

From the **Export - Solver Deck** tab, select the *Export* > **Displayed** option to export only displayed nodes and elements. Only model data associated to the displayed nodes and elements are exported. This model data includes materials and their associated curves, properties, portions of contacts, and output requests.

Create and Review Contacts

The table below describes how all slave and master set types are created and specified in contacts.

* For slave surface only

Slave and master set type	LS-DYNA card	Panel used to create card	Equivalent type in Interfaces panel, add subpanel
EQ. 0: set segment id	*SET_SEGMENT	**set_segment (contactsurfs)** or …	**csurfs**
		Interfaces, add subpanel	**entity**
EQ. 1: shell element	*SET_SHELL_Optio	**Entity Sets** or…	**sets**

set id	n	**Interfaces**, add subpanel	**entity**
EQ. 2: part set id	*SET_PART_LIST	**Entity Sets** or…	**sets**
		Interfaces, add subpanel	**comps**
EQ. 3: part id	*PART	**Collectors**	**comps**
* EQ. 4: node set id	*SET_NODE_Option	**Entity Sets** or…	**sets**
		Interfaces, add subpanel	**entity**
* EQ. 5: include all		**Interfaces**, add subpanel	**all**
* EQ. 6: part set id for exempted parts	*SET_PART_LIST	**Interfaces**, add subpanel and then **card image** sub-panel	**sets**

Add subpanel

While the **Interfaces** panel, **add** subpanel has several master and slave entity types to choose from in order to specify the LS-DYNA master or slave set for a *CONTACT, only the valid master and slave types are selectable for the particular contact you are creating.

When the master or slave type is set to **comps** and only one component is selected, the LS-DYNA type is 3, part ID, and *PART is created. When multiple components are selected, the LS-DYNA type is 2, part set ID, and *SET_PART_LIST is created.

When the master or slave type is set to **sets**, only those sets valid for the particular contact you are creating are selectable. For example, for *CONTACT_NODES_TO_SURFACE, only a list of node sets is available for slave; you will not see a list of other set types, like element or part sets.

Review Contacts

Review contacts by clicking **review** in the **Interfaces** panel, **add** subpanel.

Model Files

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

Exercise: Define Airbag, Velocity, and Contacts for the Airbag Analysis

This exercise will help you learn how to define LS-DYNA airbags, loads, and contacts using HyperMesh.

In this exercise, you will define an airbag, velocity, and contacts for a LS-DYNA analysis of a head impacting an inflating airbag.
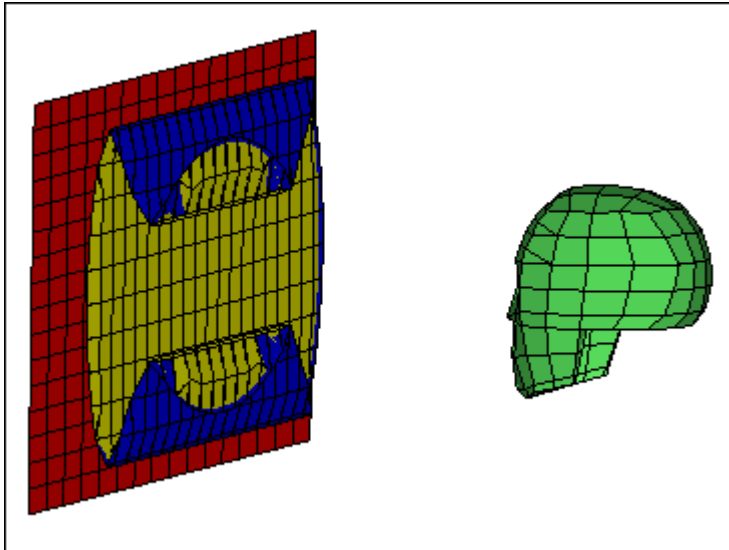
Head and airbag model

Step 1: Load the LS-DYNA user profile

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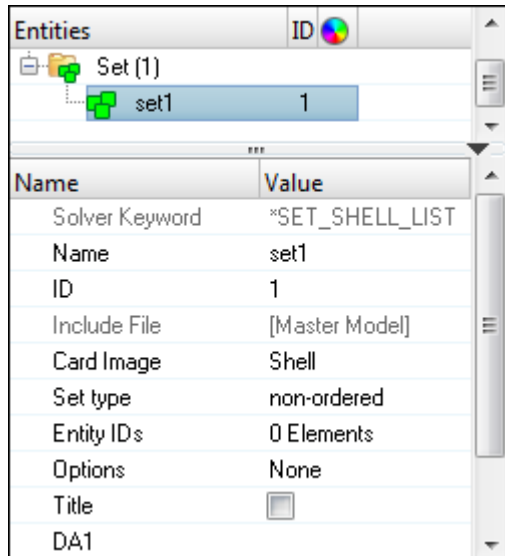2.   In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Import the LS-DYNA model

1.   From the menu bar, click *File* > *Import* > *Solver Deck*. The **Import - Solver Deck** tab opens.
2.   In the **File** field, open the file `airbag_start.key`.
3.   Click *Import*.

Steps 3-5: Define *AIRBAG_WANG_NEFSKE for the airbag mesh geometry

△ Altair

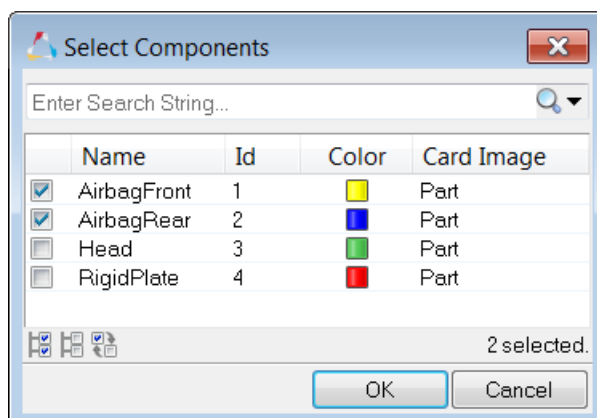### Step 3: Create a set of parts, *SET_PART_LIST, containing the AirbagFront and AirbagRear components

1. In the **Model** browser, right-click and select *Create* > *Set* from the context menu. A new set opens in the **Entity Editor**.

Entities	ID 🌈
⊟ 🗀 Set (1)	
🔲 set1	1

Name	Value
Solver Keyword	*SET_SHELL_LIST
Name	set1
ID	1
Include File	[Master Model]
Card Image	Shell
Set type	non-ordered
Entity IDs	0 Elements
Options	None
Title	☐
DA1	

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

3. Set **Card Image** to *Part*.

4. For **Entity IDs**, click *0 Components* >> *Components*.

Card Image	Part
Set type	non-ordered
Entity IDs	Components ➡ ◄
Options	None
Title	☐
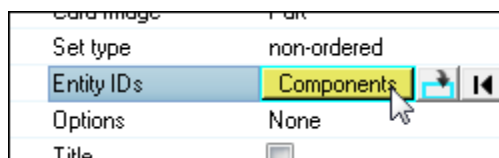
5. In the **Select Components** dialog, select *AirbagFront* and *AirbagRear* and then click *OK*.

	Name	Id	Color	Card Image
☑	AirbagFront	1	🟨	Part
☑	AirbagRear	2	🟦	Part
☐	Head	3	🟩	Part
☐	RigidPlate	4	🟥	Part

Select Components

Enter Search String...

2 selected.
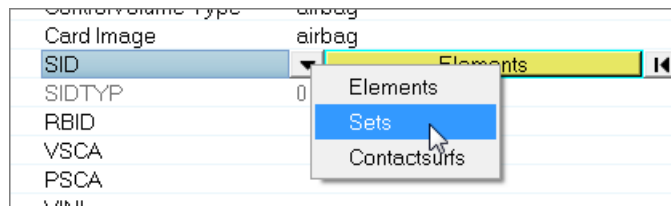
OK   Cancel

△ Altair
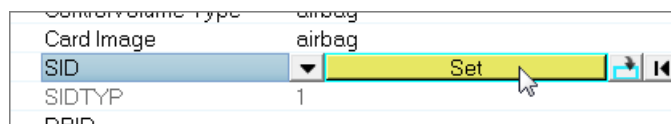
Step 4: Define the airbag (*AIRBAG_WANG_NEFSKE)

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

2. In the **Solver** browser, right-click and select **Create** > ***AIRBAG** > ***AIRBAG_WANG_NEFSKE** from the context menu. A new control volume opens in the **Entity Editor**.

Entities	ID 🔴 🔷
⊟ 📇 *AIRBAG (1)	
⊟ 📑 *AIRBAG_WANG_NEFSKE (1)	
🔧 controlvol1	1
⊟ *CONTROL (5)	

Name	Value
Solver Keyword	*AIRBAG_WANG_NEFSKE_ID
Name	controlvol1
ID	1
Include File	[Master Model]
ControlVolume Type	airbag
Card Image	airbag
SID	0 Elements
SIDTYP	0
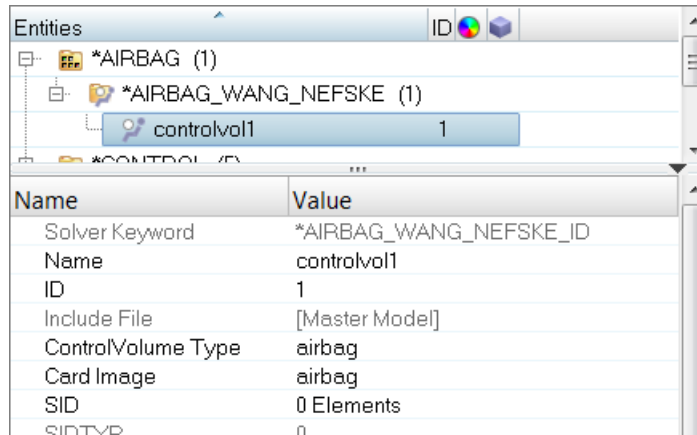
3. For **Name**, enter `airbag`.

4. Click **SID**. The entity selector becomes active.

5. Set the entity selector to **Sets**.

ControlVolume Type	airbag	
Card Image	airbag	
SID	▼	Elements ◀
SIDTYP	0	Elements
RBID		Sets
VSCA		Contactsurfs
PSCA		
VINI		

6. Click **Set**.

ControlVolume Type	airbag	
Card Image	airbag	
SID	▼	Set ➡ ◀
SIDTYP	1	
RBID		

7. In the **Select Set** dialog, select ***airbag_set*** and then click **OK**.

   Note: The parts in this set define the airbag's geometry.

Select Set	✖
Enter Search String...	🔍 ▼

	Name	Id	Card Image
◉	airbag_set	1	Part

OK	Cancel
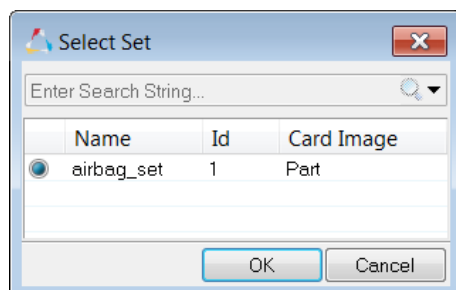
8. For **CV** (Heat capacity at constant volume), enter `1023.0`.

9. For **CP** (Heat capacity at constant pressure), enter `1320.0`.

10. For **T** (Temperature of input gas), enter `780.0`.

11. Click **LCMT** (Load curve specifying input mass flow rate) >> **Curve**.

12. In the **Select Curve** dialog, select *airbag LCMT curve* and then click **OK**.

13. For **C23** (Vent orifice coefficient), enter `1.0`.

14. Click **LCA23** (Load curve defining vent orifice area as a function of pressure) >> **Curve**.

15. In the **Select Curve** dialog, select *airbag LCA23 curve* and then click **OK**.

16. For **CP23** (Orifice coefficient for leakage), enter `1.0`.

17. For **PE** (Ambient pressure), enter `1.0E-4`.

18. For **RO** (Ambient density), enter `1.0E-9`.

19. For **GC** (Gravitational conversion constant), enter `1.0`.

Step 5: Define an initial velocity of 3 mm/ms in the negative x-direction for the head with *INITIAL_VELOCITY_GENERATION

1. In the **Model** browser, right-click and select *Create* > *Load Collector* from the context menu. A new load collector opens in the **Entity Editor**.
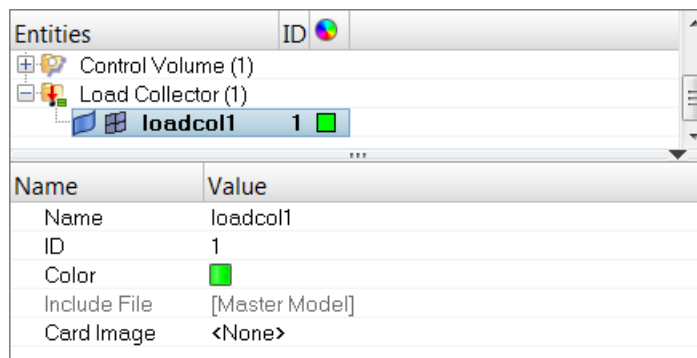
2. For **Name**, enter `velocity`.

3. Set **Card Image** to *InitialVel*.

4. Under **Options**, right-click on **NSID** and select **Create** from the context menu. The **Create Sets** dialog opens.

5. For **Entity IDs**, click **0 Nodes** >> **Nodes**.
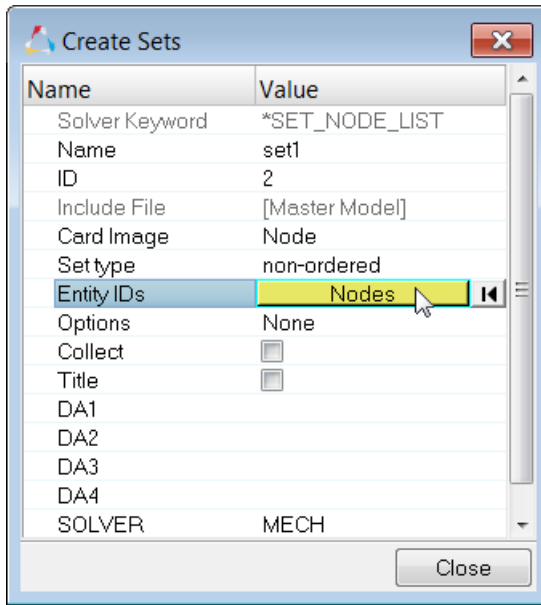
6. In the panel area, click **nodes** >> **by Collector**.

7. Select the component, **Head**.
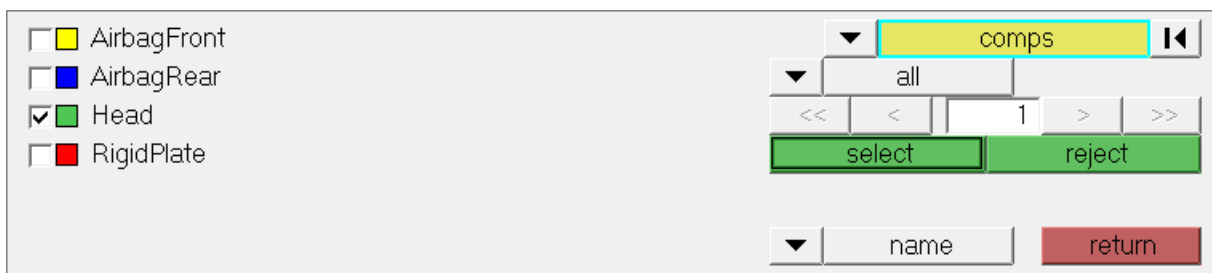
8. Click **select**.

9. Click **proceed**.

10. In the **Create Sets** dialog, click **Close**.

11. In the **Solver** browser, ***INITIAL** > ***INITIAL_VELOCITY** folder, right-click on **velocity** and select **Review** from the context menu. HyperMesh highlights the load collector and greys out all of the other entities.
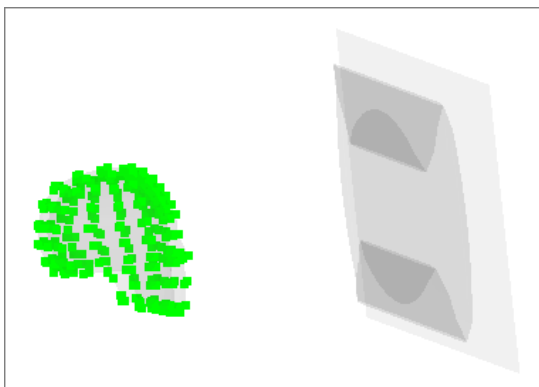
12. Return all of the entities to their original display color by righting-click on **velocity** and selecting **Reset Review** from the context menu.

Steps 6-12: Define a contact between the airbag and head with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

Step 6: Create a HyperMesh group with the card image SurfaceToSurface

1. In the **Solver** browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE** > ***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE** from the context menu. A new group opens in the **Entity Editor**.
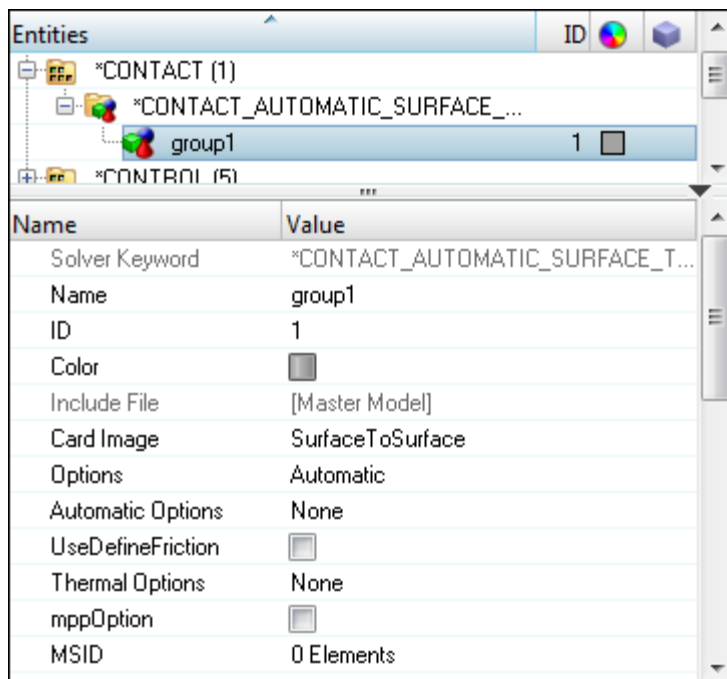
2. For **Name**, enter `Airbag_Head`.

Step 7: Specify the head to be the master surface with surface type 3, part ID

In this step, the **Entity Editor** should still be open for the **Airbag_Head** group.

1. Click **MSID**.
2. Set the entity selector to **Components**.
3. Click **Components**.

4. In the **Select Components** dialog, select **Head** and then click **OK**.

Step 8: Specify all of the airbag to be the slave surface with surface type 2, part set ID

In this step, the **Entity Editor** should still be open for the **Airbag_Head** group.

1. Click **SSID**.
2. Set the entity selector to **Set**.
3. Click **Set**.
4. In the **Select Set** dialog, select **airbag_set** and then click **OK**.

    Note: This set contains the components, **AirbagFront** and **AirbagRear**.

Step 9: View the master and slave entities

In this step, the **Airbag_Head** group should still be selected in the **Solver** browser.

1. In the **Solver** browser, right-click on **Airbag_Head** and select **Review** from the context menu. The master and slave entities temporarily display in blue and red, respectively. All of the other entities temporarily display grey.
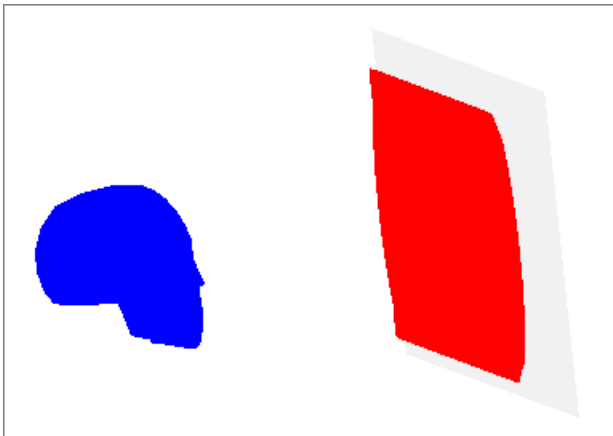
2. Return all of the entities to their original display color by right-clicking on **Airbag_Head** and selecting **Reset Review** from the context menu.

Step 10: Define *CONTACT_AIRBAG_SINGLE_SURFACE for the airbag

1. In the **Solver** browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_AIRBAG_SINGLE_SURFACE** from the context menu. A new group opens in the **Entity Editor**.
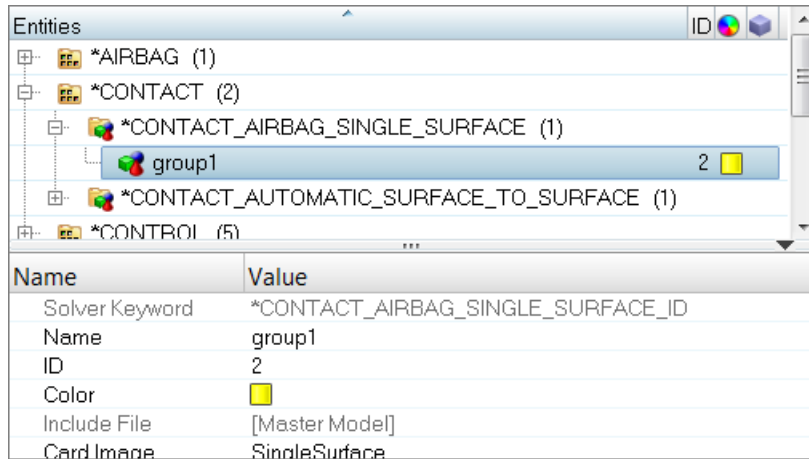
2.   For **Name**, enter `airbag`.

## Step 11: Define all of the airbag to be the slave surface with slave set type 2, part set ID

In this step, the **Entity Editor** should still be open for the **airbag** group.

1.   Click *SSID*.

2.   Set the entity selector to *Set*.

3.   Click *Set*.

4.   In the **Select Set** dialog, select *airbag_set* and then click *OK*.

Step 12: View the slave entities

In this step, the **airbag** group should still be selected in the **Solver** browser.

1.   In the **Solver** browser, right-click on *airbag* and select *Review* from the context menu. The master and slave entities temporarily display blue and red, respectively. All of the other entities temporarily display grey.
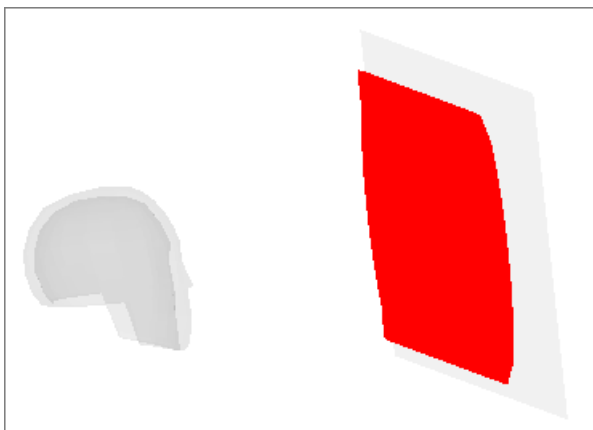
2. Return all of the entities to their original display color by right-click on **airbag** and selecting **Reset Review** from the context menu.

Steps 13- 18: Define a contact between the plate and the airbag with *CONTACT_NODES_TO_SURFACE

Step 13: Due to the dynamics of the contact, define the AirbagRear component to be the master surface with master type 0, set segment ID
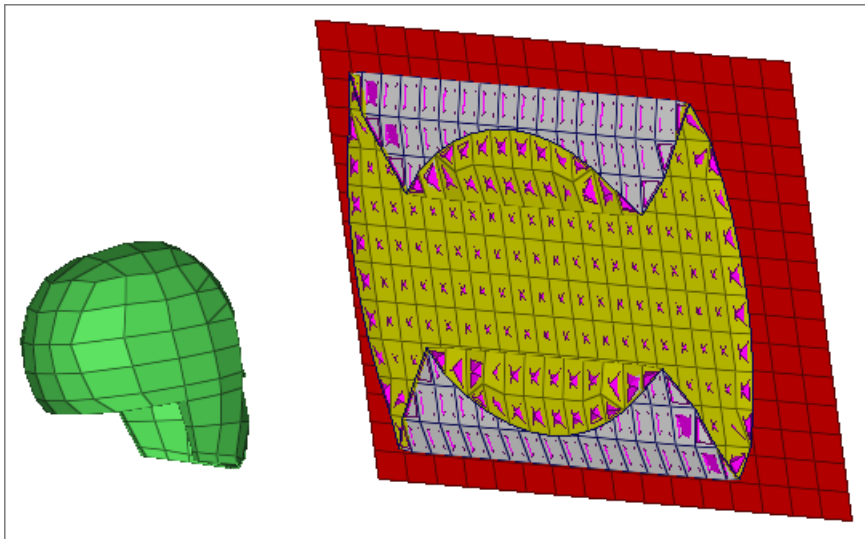
1. In the **Solver** browser, right-click and select **Create** > ***SET** > ***SET_SEGMENT** > ***SET_SEGMENT** from the context menu. A new contactsurf opens in the **Entity Editor**.
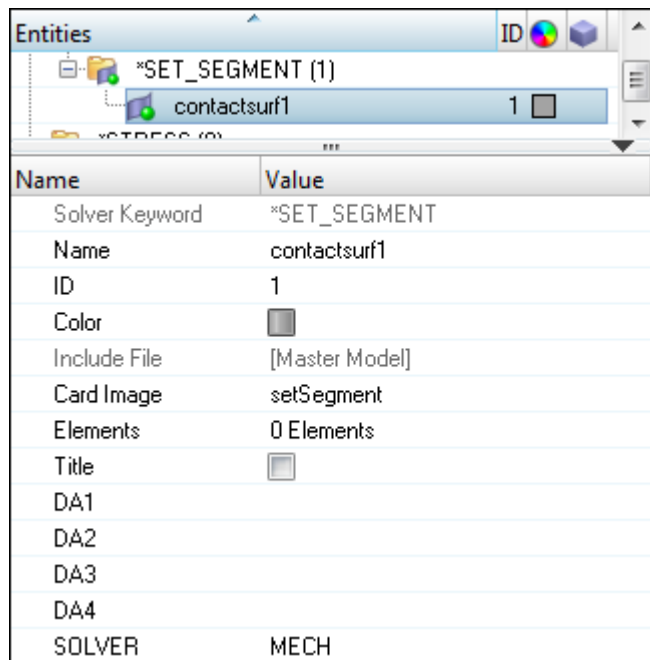
2. For **Name**, enter `AirbagRear_master`.

3. Optional. Click the **Color** icon and select a color for the contactsurf.

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

5. In the panel area, set the second switch to **elems**.
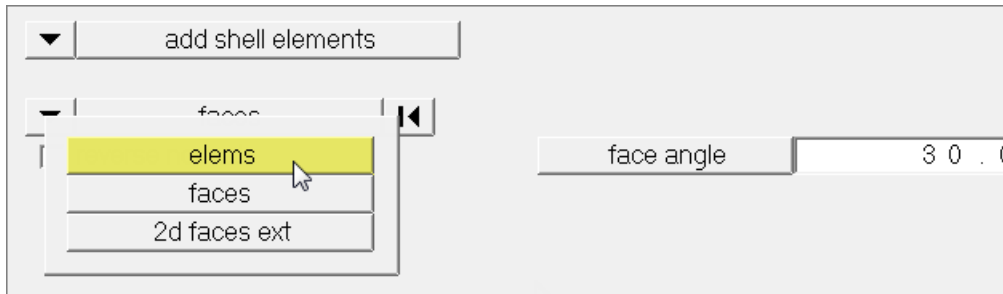
6. Click **elems** >> **by collector**.
7. Select the component, **AirbagRear**.
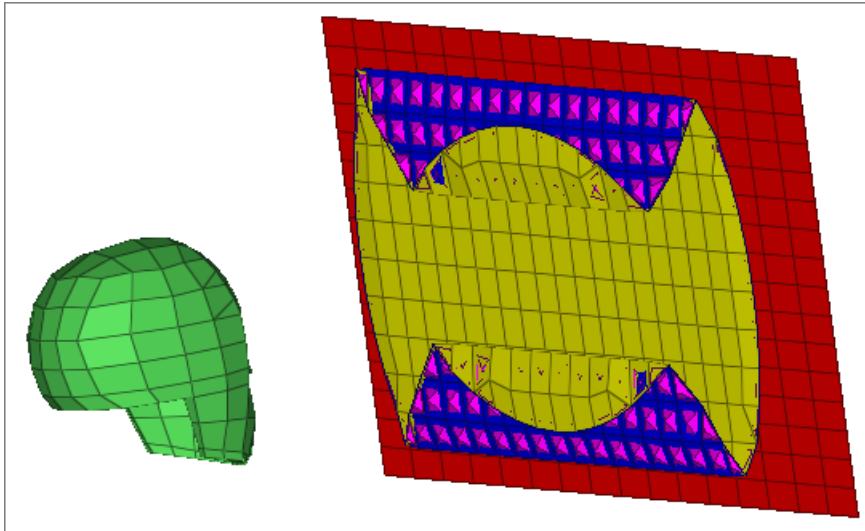8. Click **select**.
9. Click **add**.

Step 14: Reverse the contactsurf's pyramids so they point out of the airbag

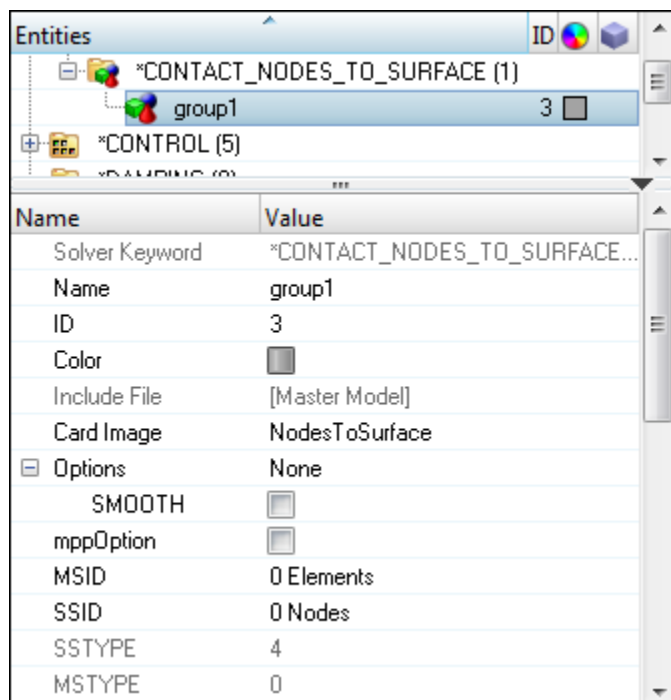1. In the panel area, set the first switch to **adjust normals**.
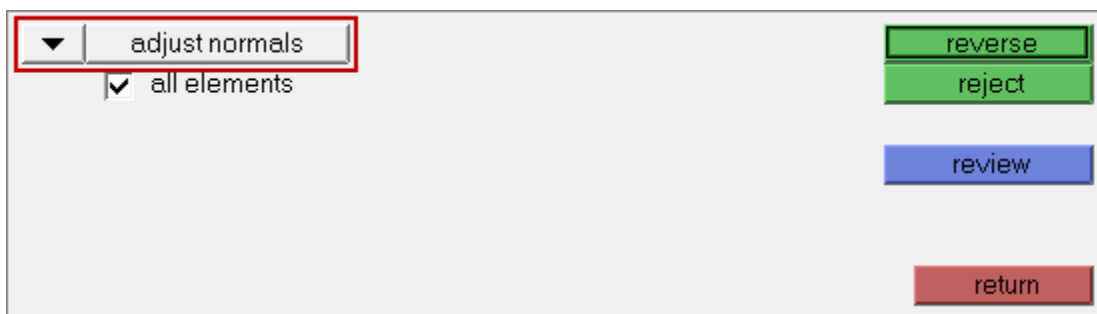
2. Select the **all elements** checkbox.
3. Click **reverse**.

4.   Click *return*.

Step 15: Create *CONTACT_NODES_TO_SURFACE card

1.   In the *Solver* browser, right-click and select *Create* > *CONTACT* > *CONTACT_NODES_TO_SURFACE* > *CONTACT_NODES_TO_SURFACE* from the context menu. A new group opens in the **Entity Editor**.

2.   For **Name**, enter Airbag_Plate.

Step 16: Specify the AirbagRear_master contactsurf for the contact's master surface

In this step, the **Entity Editor** should still be open for the **Airbag_Plate** group.

1. Click *MSID*.

2. Set the entity selector to *Contactsurfs*.

3. Click *Contactsurfs*.

4. In the **Select Contactsurfs** dialog, select *AirbagRear_master* and then click *OK*.

Step 17: Define the plate to be the contact's slave surface with slave type 4, node set ID

1. For **SSID**, click *0 Nodes* >> *Nodes*.

2. In the panel area, set the switch to nodes.

3. Click *nodes* >> *by collector*.

4. Select the component, *RigidPlate*.

5. Click *select*.
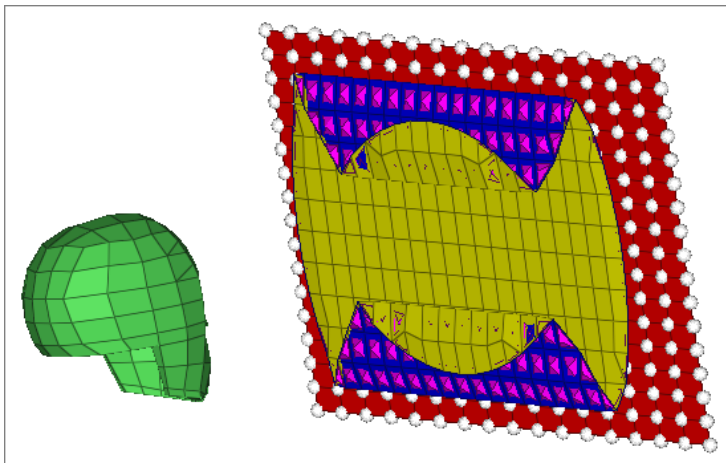
6. Click *add*. HyperMesh adds the slave selection to the group **Airbag_Plate**.
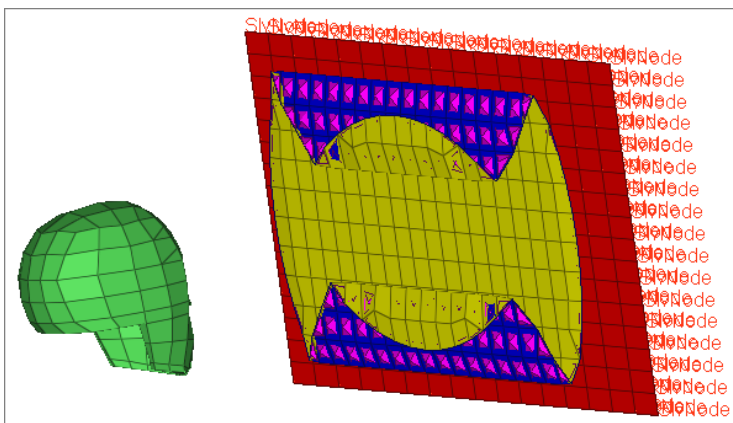
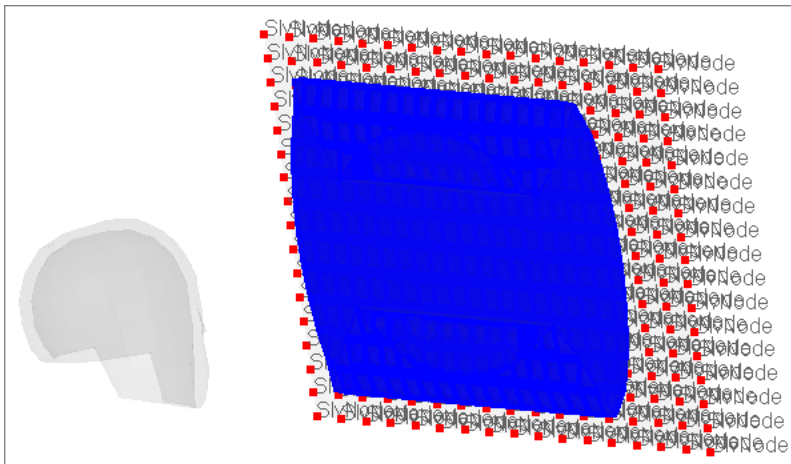7. Click *return*.

Step 18: View the master and slave entities

1. In the **Solver** browser, ***CONTACT** > ***CONTACT NODES TO SURFACE** folder, right-click on *Airbag_Plate* and select *Review* from the context menu. The master and slave entities temporarily display blue and red, respectively. All of the other entities temporarily display grey.
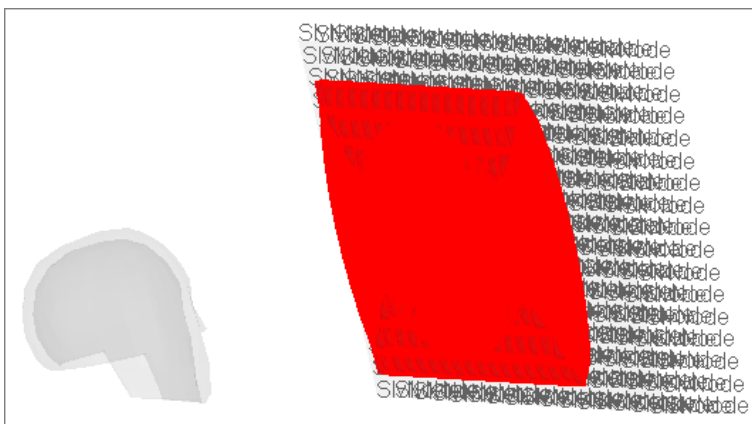

2. Return all of the entities to their regular display color by right-clicking on *Airbag_Plate* and selecting *Reset Review* from the context menu.
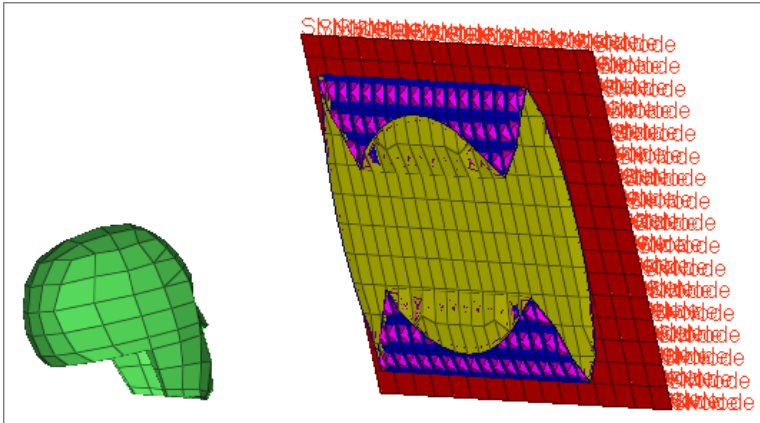
Step 19: Review the created solver entities using the Solver Browser

1. In the **Solver** browser, ***CONTACT** > ***CONTACT_AIRBAG_SINGLE_SURFACE** folder, right-click on *airbag* and select *Review* from the context menu. The master and slave entities temporarily display blue and red, respectively. All of the other entities temporarily display grey.

   Note: Only slave (red) entities are shown because there are no master entities for this type of contact.
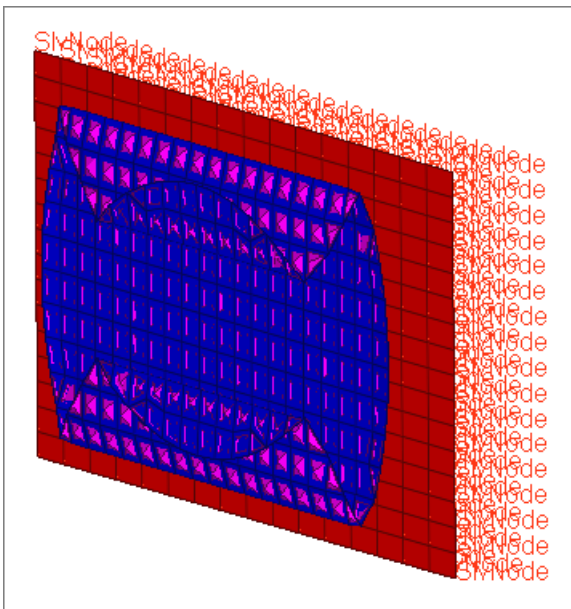
2. Return all of the entities to their regular display color by right-clicking on ***airbag*** and selecting ***Reset Review*** from the context menu.


3. In the **Solver** browser, ***CONTACT** > ***CONTACT NODES TO SURFACE** folder, right-click on ***Airbag_Plate*** and select ***Isolate Only*** from the context menu. Only the elements/components that are implicated in this contact display.

   Tip: If master and slave entities are not visible, make sure the **Show/Isolate/IsolateOnly/Attached** checkbox is selected in the **Options** tab of the **Browser Configuration** dialog. Access the **Browser Configuration** dialog by right-clicking in the **Model** browser and selecting ***Configure Browser*** from the context menu.

4. In the **Solver** browser, ***CONTACT** > ***CONTACT_AIRBAG_SINGLE_SURFACE** folder, right-click on ***airbag*** and select ***Show*** from the context menu. The entire airbag displays, as this entity contains the entire airbag.
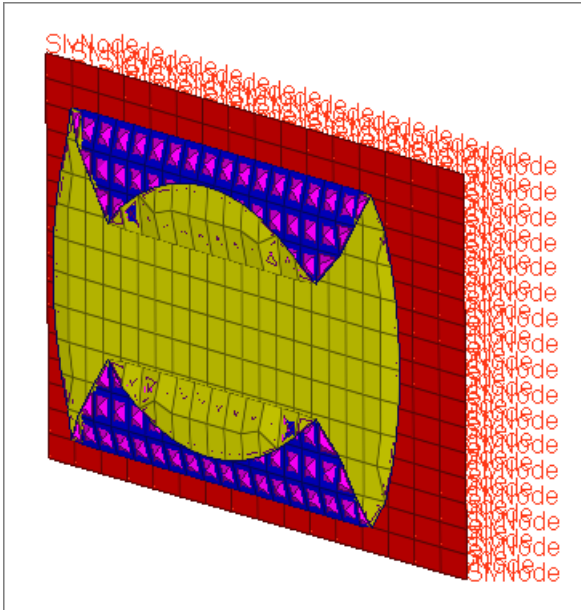
5. In the ***INITIAL** > ***INITIAL_VELOCITY** folder, right-click on *velocity* and select *Review* from the context menu. The nodes on which velocity was applied displays.
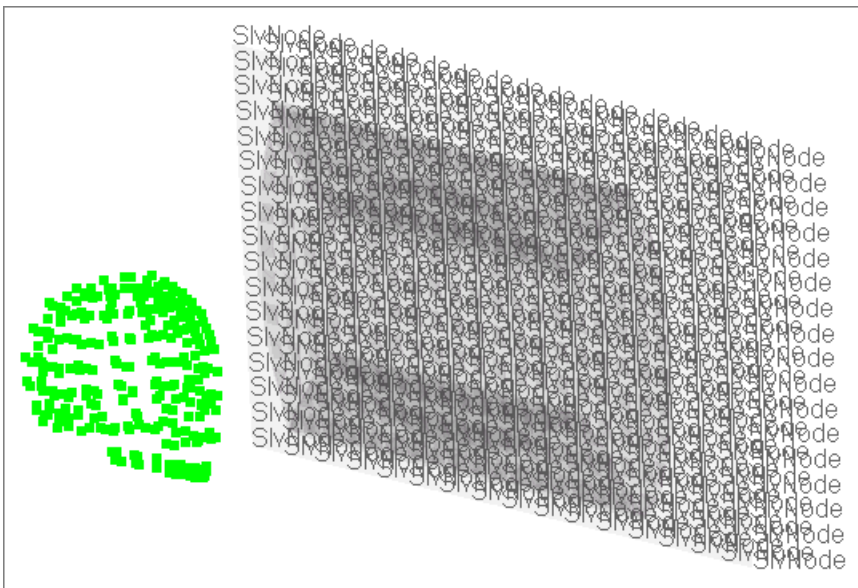
6. Return all of the entities to their regular display color by right-clicking on *velocity* and selecting *Reset Review* from the context menu.

Step 20: Export the model to an LS-DYNA 971 formatted input file

1. From the menu bar, click *File* > *Export* > *Solver Deck*. The **Export - Solver Deck** tab opens.

2. Set **Template** to *Keyword971*.

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

4. Next to **Export options**, click ⊻.

5. Set **Export** to *All*.

6. Click *Export*.

Step 21 (Optional): Submit the LS-DYNA input file to LS-DYNA 970

1. From the **Start** menu, open the **LS-DYNA Manager** program.

2. From the **solvers** menu, select *Start LS-DYNA analysis*.

3. Load the file `airbag_complete.key`.

4. Start the analysis by clicking *OK*.

Step 22 (Optional): View the results in HyperView

The exercise is complete. Save your work to a HyperMesh file.