Altair HyperMesh 2019 Tutorials

HM-4610: Using Curves, Beams, Rigid Bodies Joints, and Loads in DYNA

altairhyperworks.com

HM-4610: Using Curves, Beams, Rigid Bodies Joints, and Loads in DYNA

In this tutorial, you will learn how to:

- Create XY curves to define non-linear materials
- Define beam elements with HyperBeam
- Create constrained nodal rigid bodies
- Create joints
- Define *DEFORMABLE_TO_RIGID
- Define *LOAD_BODY
- Define *BOUNDARY_PRESCRIBED_MOTION_NODE
- Use the HyperMesh Component Table tool to review the model's data

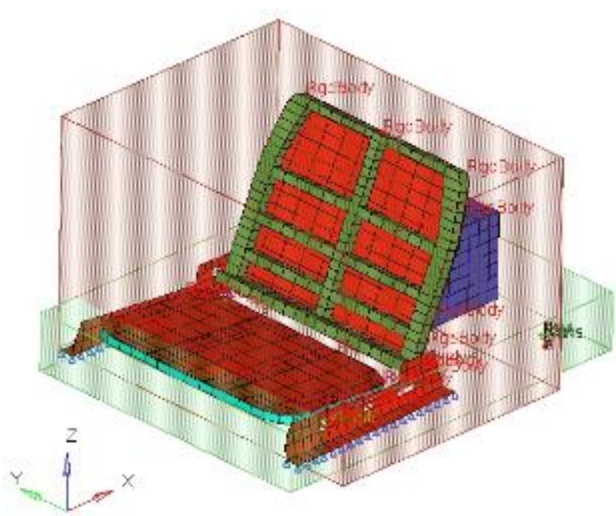
Model Files

This tutorial uses the `seat_start.hm` and `seat_2.hm` files, which can be found in `<hm.zip>/interfaces/lsdyna/`. Copy the file(s) from this directory to your working directory.

Exercise 1: Define Model Data for the Seat Impact Analysis

This exercise will help you become familiar with defining LS-DYNA model data in HyperMesh.

In this exercise you will define and review model data for a LS-DYNA analysis of a vehicle seat impacting a rigid block. The seat and block model is shown in the image below.



Seat and block model

Step 1: Load the LS-DYNA user profile

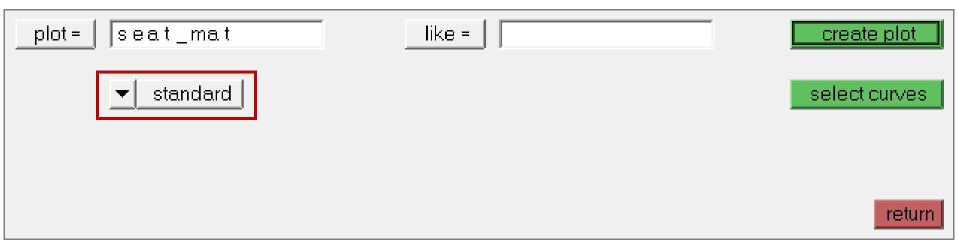
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Retrieve the HyperMesh file

1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `seat_start.hm` file. The model appears in the graphics area.
3. Observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).

Step 3: Create an xy plot

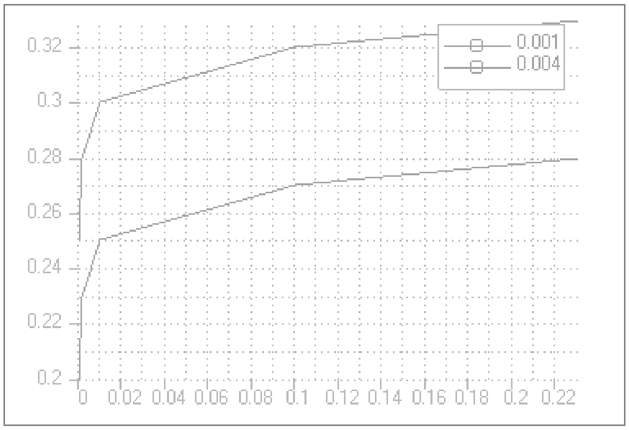
1. Open the **Plots** panel by clicking ***XYPlots*** > ***Create*** > ***Plots*** from the menu bar.
2. In the **plot=** field, enter `seat_mat`.
3. Set plot type to **standard**.



4. Leave the **like =** field empty. When an existing plot is selected, the new plot adopts its attributes.
5. Click ***create plot***.
6. Click ***return.***

Step 4: Input data from a file to create two stress-strain curves

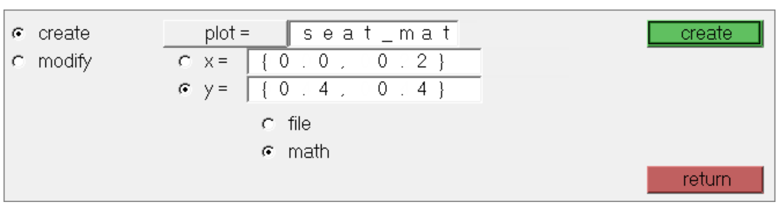
1. Open the **Read Curves** panel by clicking ***XYPlots*** > ***Create*** > ***Curves*** > ***Read Curves*** from the menu bar.
2. Leave the **plot =** field set to **seat_mat**.
3. Click ***browse***.
4. In the **Open** dialog, open the file `seat_mat_data.txt`.
5. Click ***input***. HyperMesh creates two curves, and names them **0.001** and **0.004**.



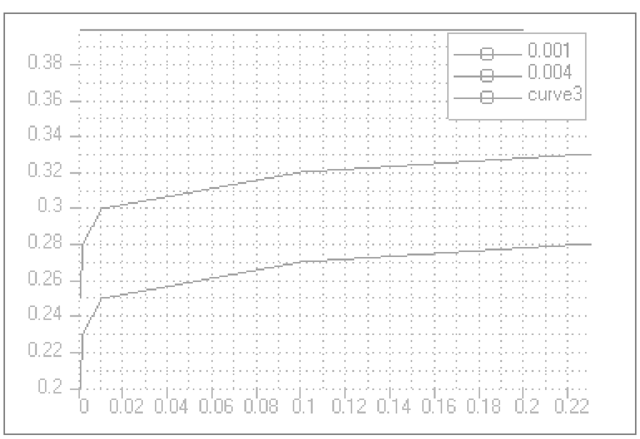
6. Click ***return***.

Step 5: Create a dummy xy curve to be used to create *DEFINE_TABLE

1. Open the **Edit Curves** panel by clicking ***XYPlots*** > ***Edit*** > ***Curves*** from the menu bar.
2. Go to the **create** subpanel.
3. Click **plot =**, and select ***seat_mat***.
4. Select **math**.
5. In the **x =** field, enter `{0.0, 0.2}`.
6. In the **y =** field, enter `{0.4, 0.4}`.



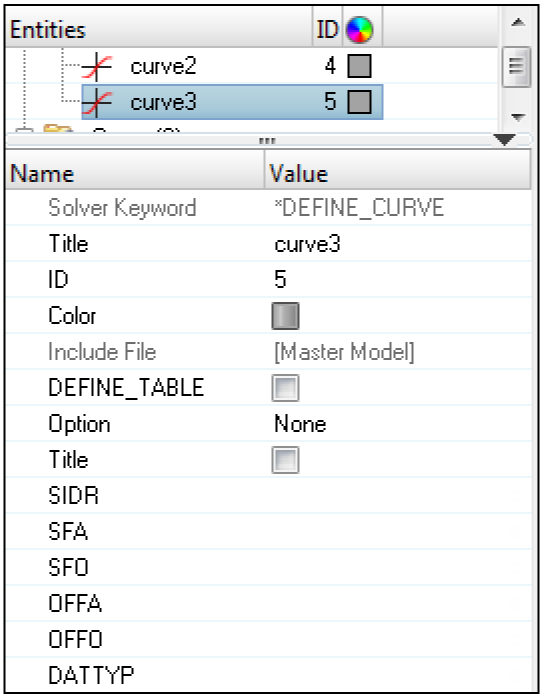
7. Click ***create***. HyperMesh creates a curve in the **seat_mat** plot, and names it **curve3**.



8. Click ***return***.

Step 6: Create *DEFINE_TABLE from the dummy curve

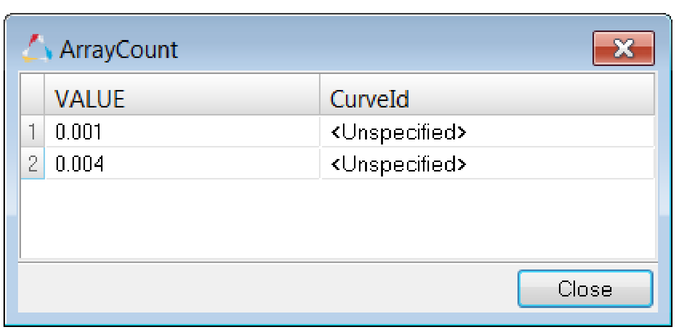
1. In the **Model** browser, **Curve** folder, click ***curve3***. The **Entity Editor** opens, and displays the curve's corresponding data.



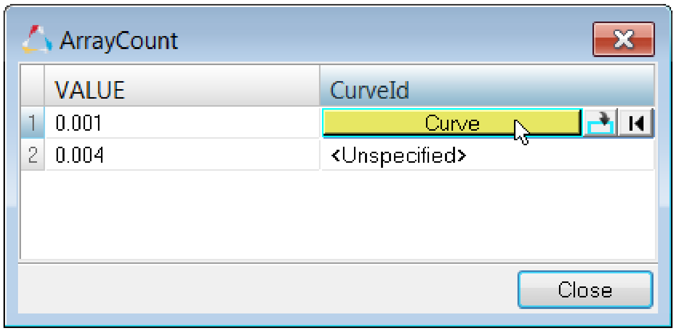
2. Select the ***DEFINE_TABLE*** checkbox.
3. For ***ArrayCount***, enter `2`.

   Note: This is the number of strain rate values to be specified.

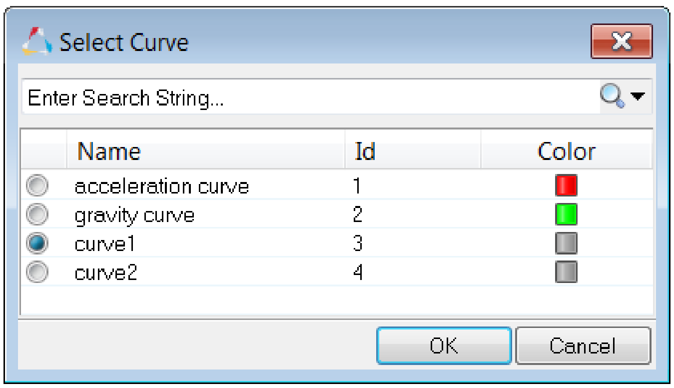
4. In the **Data: VALUE** row, click .
5. In the **ArrayCount** dialog, enter `0.001` in the strain rate **VALUE(1)** field and `0.004` in the strain rate **VALUE(2)** field.



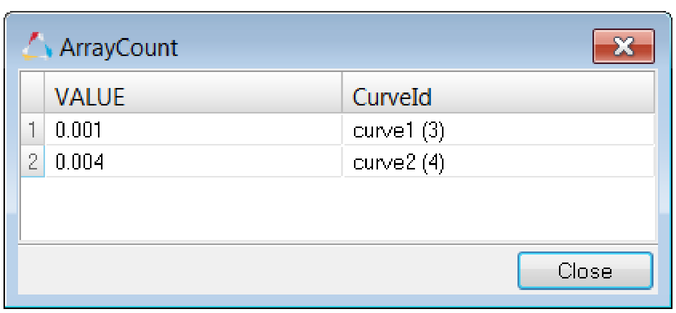
6. In the ***CurveId(1)*** field, click ***Unspecified*** >> ***Curve***.



7. In the **Select Curve** dialog, select ***curve1*** and then click ***OK***.

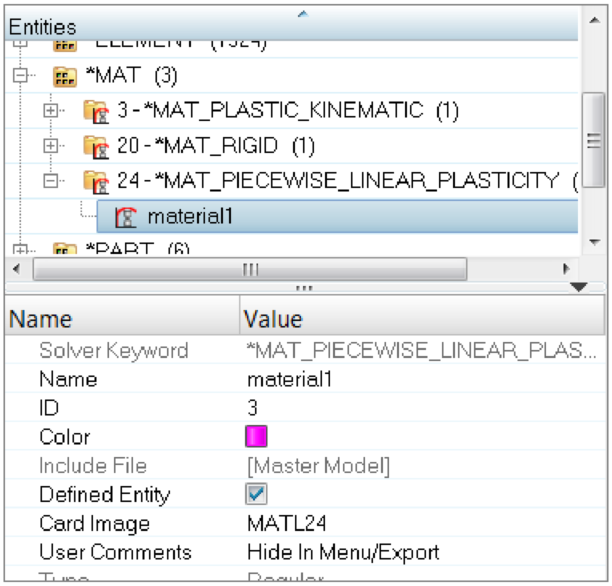


8. In the ***CurveId(2)*** field, click ***Unspecified*** >> ***Curve***.
9. In the **Select Curve** dialog, select ***curve2*** and then click ***OK***.
10. Click ***Close***.

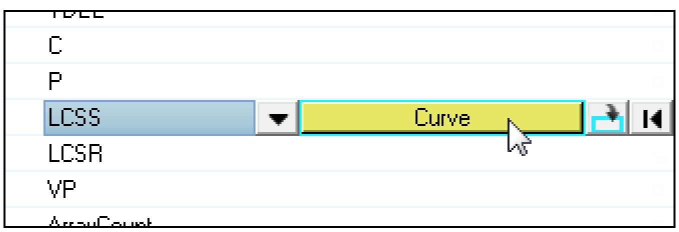


Step 7: Create the non-linear material (*MAT_PIECEWISE_LINEAR_PLASTICITY)

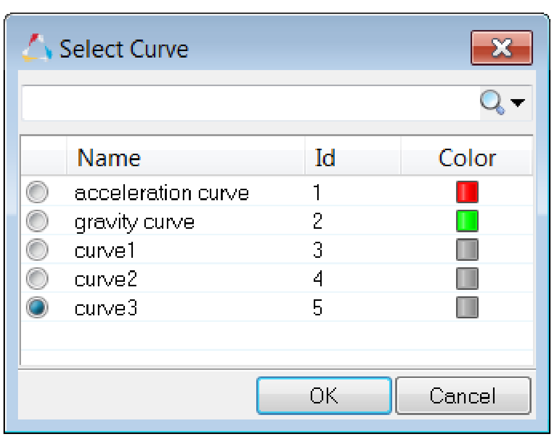
1. Open the **Solver** browser by clicking ***View*** > ***Browsers*** > ***HyperMesh*** > ***Solver*** from the menu bar.
2. In the **Solver** browser, right-click and select ***Create*** > ****MAT*** > ***MAT (1-50)*** > ***24-*MAT_PIECEWISE_LINEAR_PLASTICITY*** from the context menu. HyperMesh creates and opens a new material in the **Entity Editor**.



3. For **Name**, enter `steel`.
4. For **Rho** (Mass density), enter `7.8 E-6`.
5. For **E** (Young modulus), enter `200`.
6. For **NU** (Poisson ratio), enter `0.3`.
7. For **SIGY** (Yield stress), enter `0.25`.
8. Click ***LCSS***, and then click ***curve***.

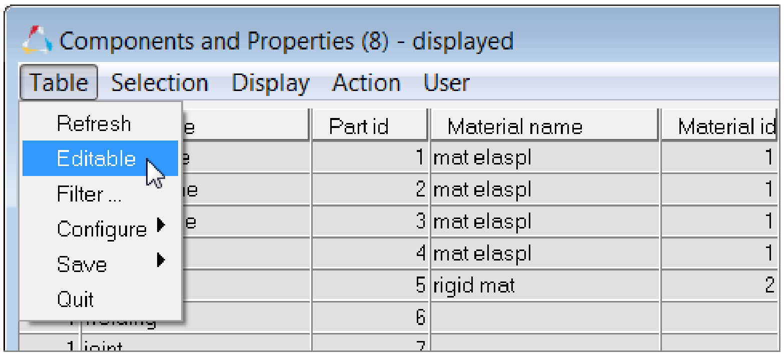


9. In the **Select Curve** dialog, select ***curve3*** and then click ***OK***.

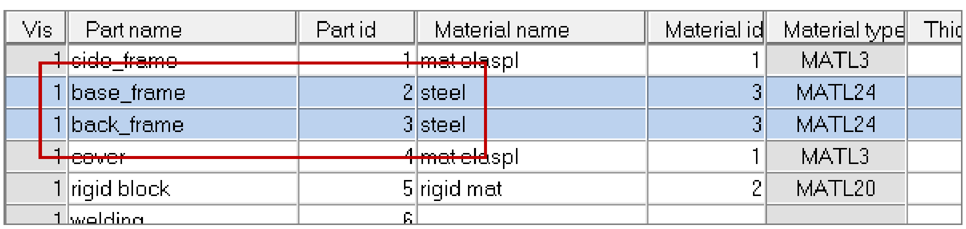


Step 8: Update the base_frame and back_frame components with the new non-linear material

1. From the menu bar, click ***Tools*** > ***Component Table***.
2. In the **Components and Properties** dialog, click ***Table*** > ***Editable*** from the menu bar.



3. Select the component, **base_frame**.
4. Set **Assign Values** to ***Material name***.
5. Set **HM-Mats** to ***steel***.
6. Click ***Set***. HyperMesh assigns the material **steel** to the component **base_frame**.
7. In the **Confirm** dialog, click ***Yes***.
8. Assign the material **steel** to the component, **back_frame**.



Vis	Part name	Part id	Material name	Material id	Material type	Thic
1	side_frame	1	mat elaspl	1	MATL3	
1	base_frame	2	steel	3	MATL24	
1	back_frame	3	steel	3	MATL24	
1	cover	4	mat elaspl	1	MATL3	
1	rigid block	5	rigid mat	2	MATL20	
1	welding	6				

9. From the menu bar, click ***Table*** > ***Quit***.

Steps 9-11: Create a beam element, *ELEMENT_BEAM, to complete the seat's back_frame connection to the side_frame on the left side

Step 9: Restore a pre-defined view

1. In the **Model** browser, **View** folder, right-click on ***Beam_view*** and select ***Show*** from the context menu.

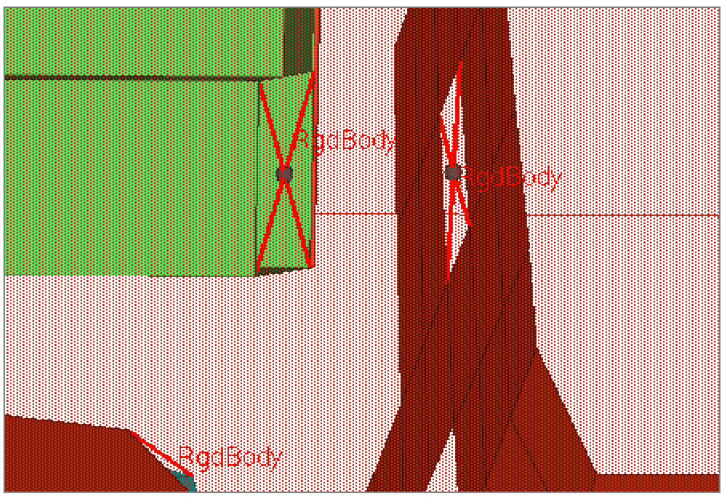
Step 10: Set the current component to beams

1. In the **Model** browser, **Component** folder, right-click on **beams** and select ***Make Current*** from the context menu. HyperMesh sets the **beam** component as the current collector.

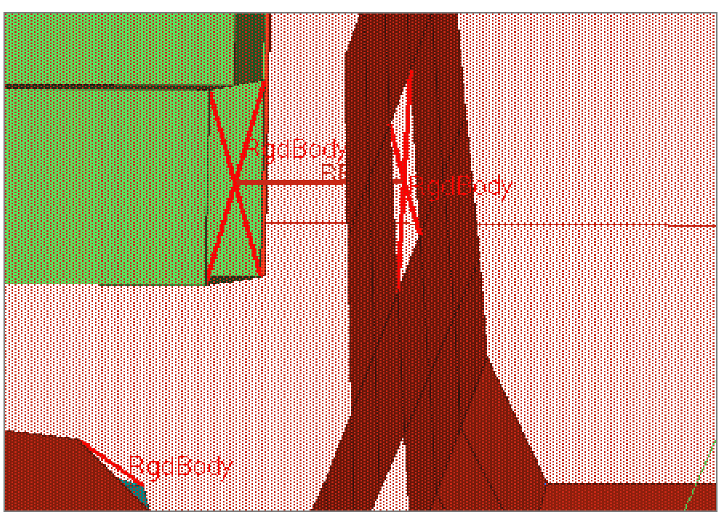
Step 11: Create the beam

1. Opens the **Bars** panel by clicking ***Mesh*** > ***Create*** > ***1D Elements*** > ***Bars*** from the menu bar.
2. Under **orientation**, click the switch and select ***node***.

   Note: You will select a **direction node** later to define the beam's section orientation.
3. Using the **node A** selector, select the center node of the left nodal rigid body.
4. Using the **node B** selector, select the center node of the right nodal rigid body.



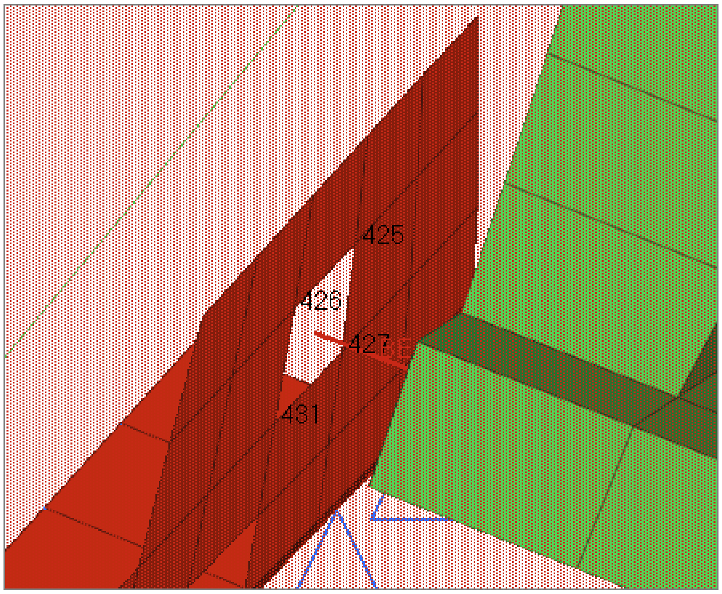
5. Using the **direction node** selector, select any non-center node on one of the nodal rigid bodies. HyperMesh creates the beam.



6. Click ***return***.

Step 12: Display node IDs for ease of following the next steps

1. Open the ***Numbers*** panel by clicking on the **Display** toolbar.
2. Set the entity selector to ***nodes***.
3. Click ***nodes*** >> ***by id***.
4. In the **id=** field, enter `425-427, 431`.
5. Press ***ENTER***.
6. Select the ***display*** checkbox.
7. Click ***on***. HyperMesh displays the IDs.



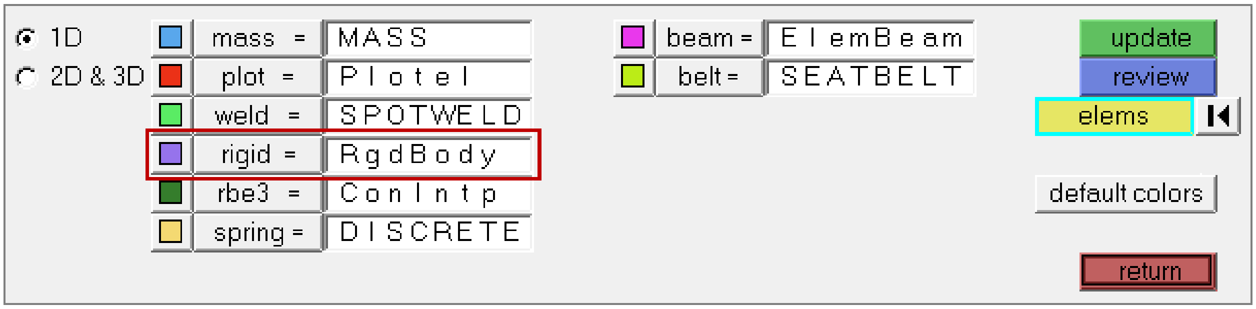
8. Click ***return.***

Step 13: Set the current component to welding

1. In the **Model** browser, **Component** folder, right-click on ***welding*** and select ***Make Current*** from the context menu. HyperMesh sets the **welding** component as the current collector.

Step 14: Select the RgdBody type for the HyperMesh rigid configuration

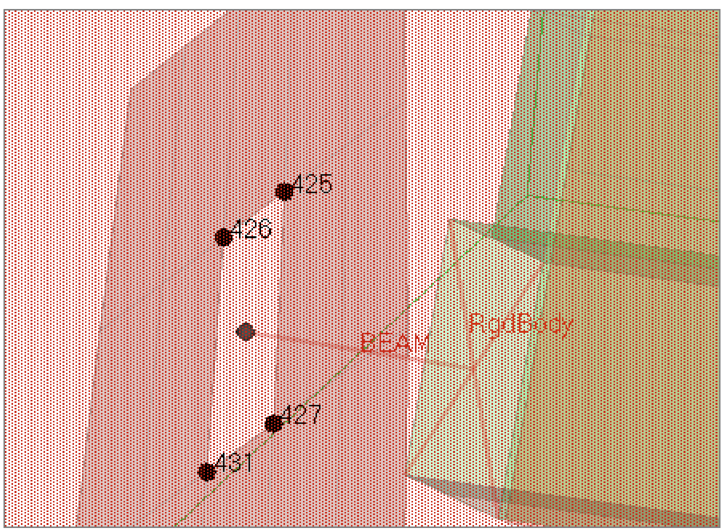
1. Open the **Element Type** panel by clicking ***Mesh*** > ***Assign*** > ***Element Type*** from the menu bar.
2. Select the elements to update.
3. Click ***rigid =***, and then select ***RgdBody***.



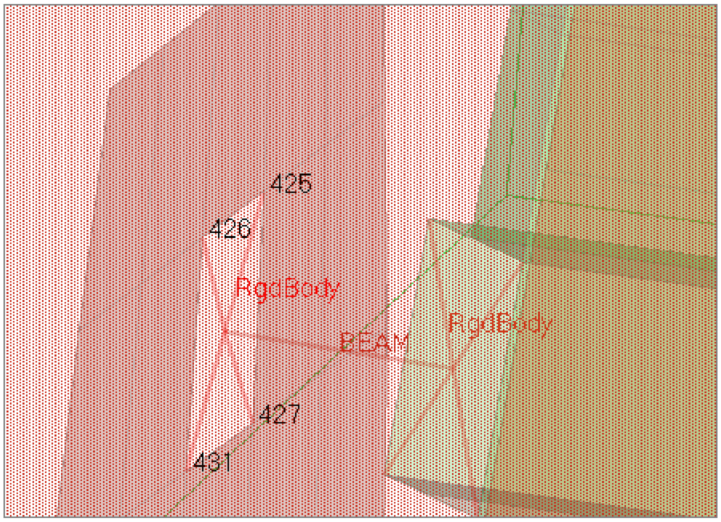
4. Click ***update***.
5. Click ***return.***

Step 15: Create the nodal rigid body (*CONSTRAINED_NODAL_RIGID_BODY)

1. In the **Solver** browser, right-click and select ***Create*** > ****CONSTRAINED*** > ****CONSTRAINED_NODAL_RIGID_BODY*** > ****CONSTRAINED_NODAL_RIGID_BODY*** from the context menu.
2. In the **Rigids** panel, set the **nodes 2-n** selector to ***multiple nodes***.
3. Using the **node1** selector, select the beam's free end.
4. Click ***nodes 2-n: nodes*** >> ***by id***.
5. In the **id=** field, enter `425`, `426`, `427`, `431`.
6. Press ***ENTER***.



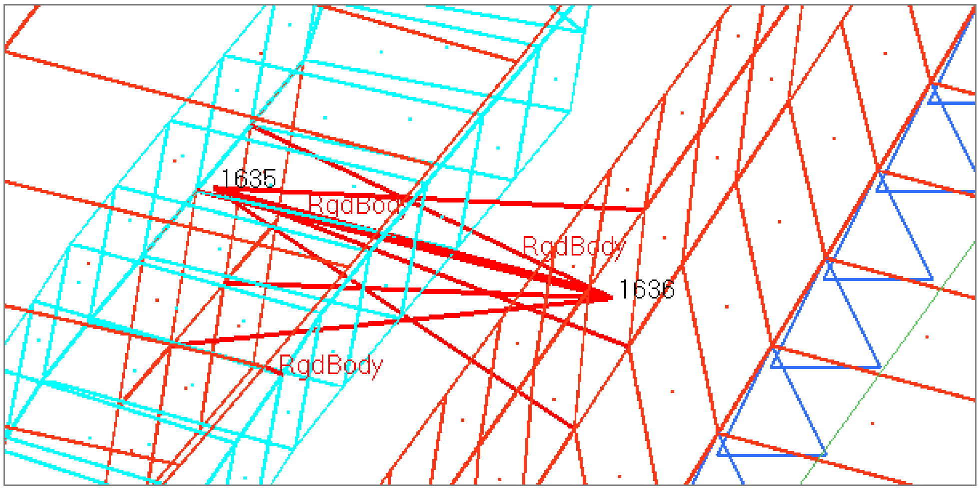
7. Clear the **attach nodes as set** checkbox selected.
8. Click ***create***. HyperMesh creates the nodal rigid body.



9. Click ***return.*** HyperMesh does not create *CONSTRAINED_JOINT_STIFFNESS; it is not needed for this joint to work.

Step 16: Display node IDs for ease of following the next steps

1. On the **Visualization** toolbar, click  to display the model's elements as wireframe elements skin only.
2. Opens the ***Numbers*** panel.
3. Set the entity selector ***nodes***.
4. Click ***nodes*** >> ***by id***.
5. In the **id=** field, enter `1635, 1636`.
6. Press ***ENTER***.
7. Select the ***display*** checkbox.
8. Click ***on***. HyperMesh displays the IDs.



9. Click ***return.***

Step 17: Activate coincident picking

1. Open the **Graphics** panel by clicking ***Preferences*** > ***Graphics*** from the menu bar.
2. Select the ***coincident picking*** checkbox.
3. Click ***return***.

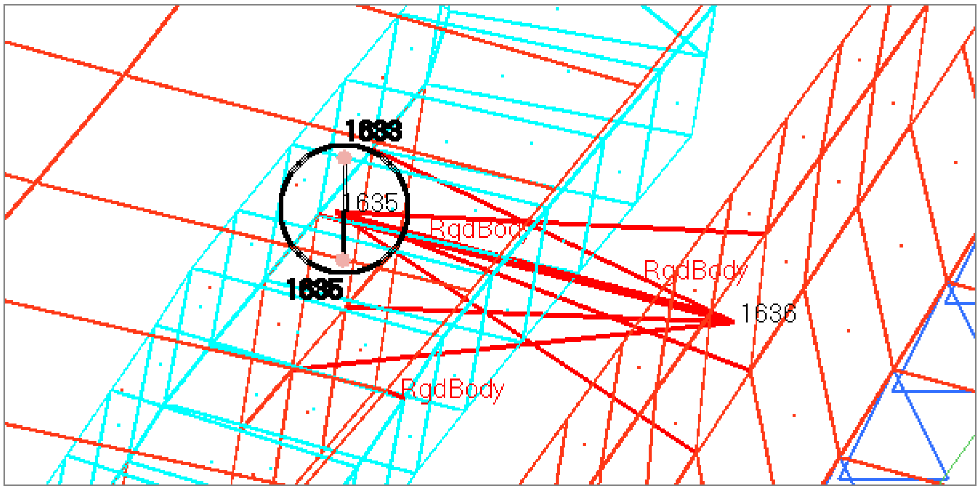
Step 18: Set the current component to joint

1. In the **Model** browser, **Component** folder, right-click on ***joint*** and select ***Make Current*** from the context menu. HyperMesh sets the **joint** component as the current collector.

Step 19: Create a revolute joint between two nodal rigid bodies (*CONSTRAINED_JOINT_REVOLUTE)

Rigid bodies must share a common edge along which to define a joint. This edge, however, must not have nodes merged together. Two rigid bodies will rotate relative to each other along the axis defined by the common edge.

1. In the **Solver** browser, right-click and select ***Create*** > ****CONSTRAINED*** > ****CONSTRAINED_JOINT_REVOLUTE*** > ****CONSTRAINED_JOINT_REVOLUTE*** from the context menu.
2. In the **Joints** panel, set **joint type** to ***revolute***.
3. Using the **node 1** selector, click node ***1635***. The coincident picking mechanism displays two nodes: **1635** and **1633**.



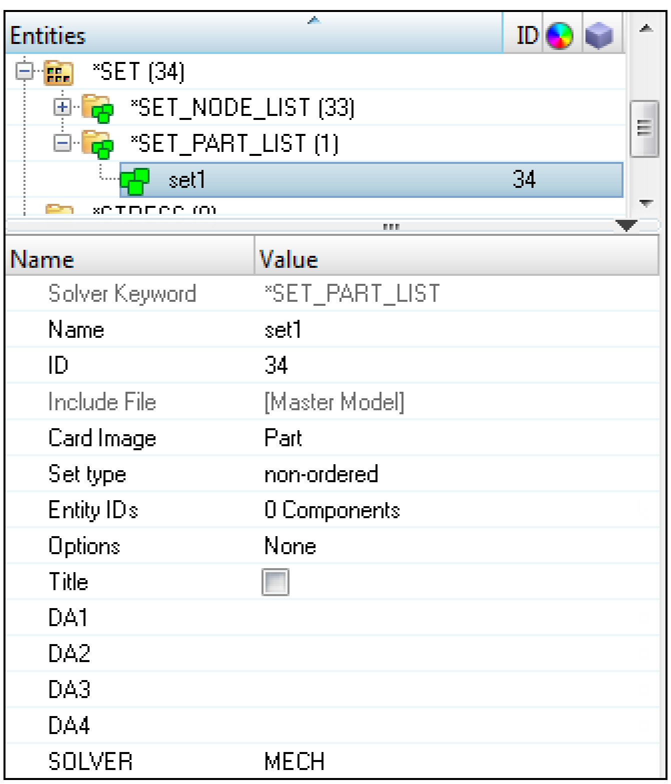
4. From the coincident picking mechanism, click node ***1635***. Hypermesh selects node 1635 for node 1 in rigid body A.
5. Using the **node 2** selector, click node ***1635***. The coincident picking mechanism displays two nodes: **1635** and **1633**.

6. From the coincident picking mechanism, click node ***1633***. HyperMesh selects node 1633 for node 2 in rigid body B.
7. Using the **node 3** selector, click node ***1636***. The coincident picking mechanism displays two nodes: **1636** and **1634**.
8. From the coincident picking mechanism, click node ***1636***. HyperMesh selects node 1636 for node 3 in rigid body A.
9. Using the **node 4** selector, select node ***1634*** for node 4 in rigid body B.
10. Click ***create***. HyperMesh creates the joint.
11. Click ***return.***

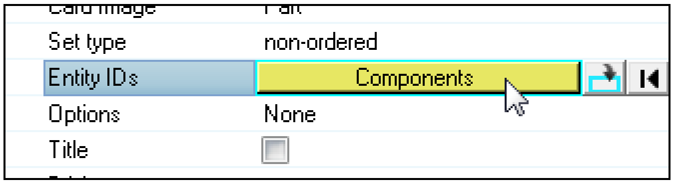
Steps 20-22: Define *DEFORMABLE_TO_RIGID to set up the moving seat as rigid until the time of impact with the block, to reduce computation time

Step 20: Create an entity set that contains the components base_frame, back_frame, and cover

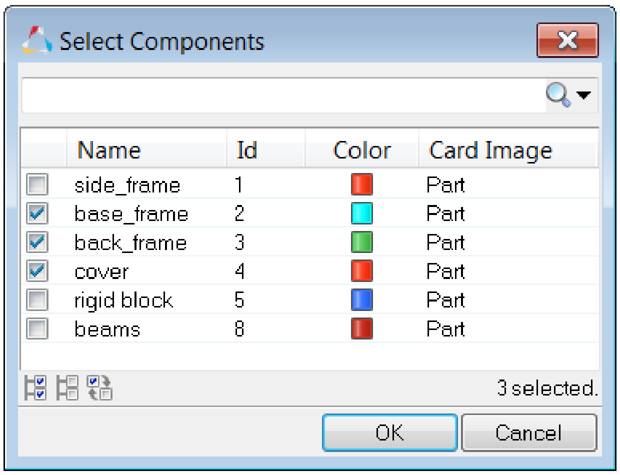
1. In the **Solver** browser, right-click and select ***Create*** > ****SET*** > ***SET_PART** > ****SET_PART_LIST*** from the context menu. HyperMesh creates and opens a new set in the **Entity Editor**.



2. For **Name**, enter `set_part_seat`.
3. For **Entity IDs**, click ***0 Components*** >> ***Components***.

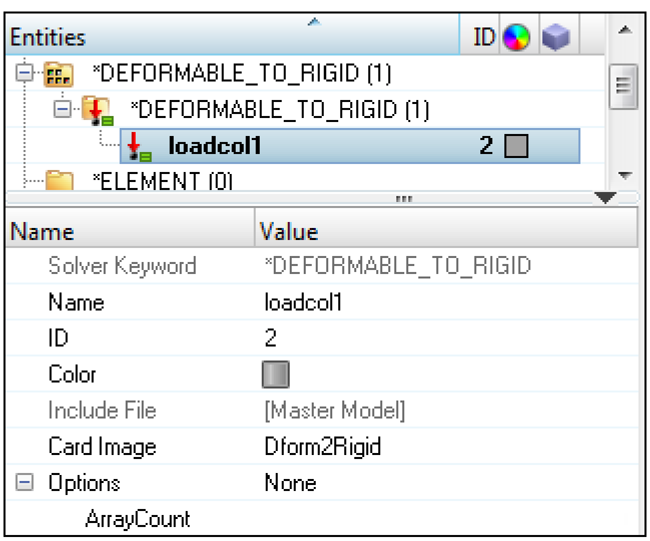


4. In the **Select Components** dialog, select ***base_frame***, ***back_frame***, and ***cover*** and then click ***OK***.

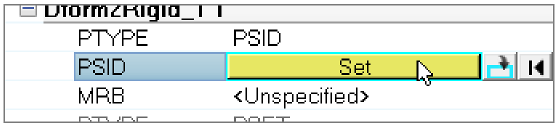


Step 21: Define *DEFORMABLE_TO_RIGID to switch the deformable seat to rigid at the beginning of the analysis

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



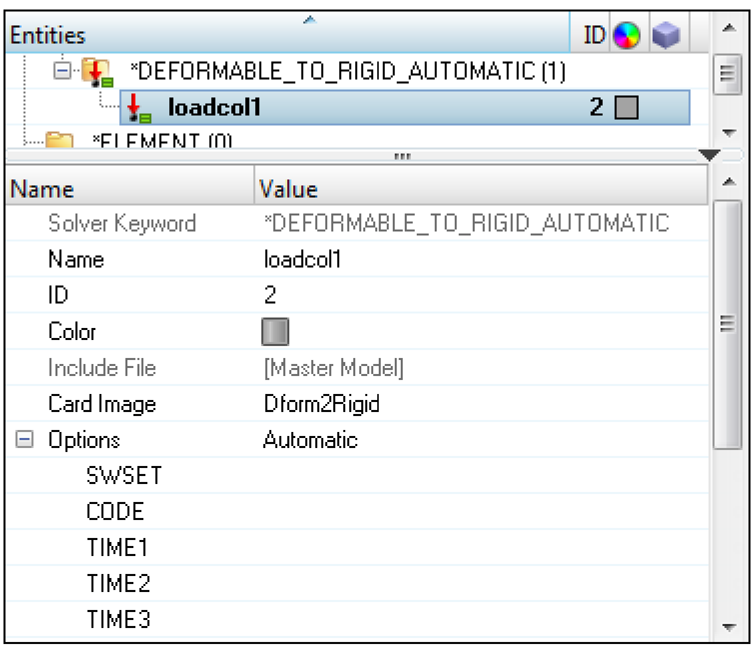
2. For **Name**, enter `dtor`.
3. For **ArrayCount**, select ***1***.
4. For **PSID**, click ***Unspecified*** >> ***Set***.



5. In the **Select Set** dialog, select ***set_part_seat*** and then click ***OK***.
6. For **MRB**, click ***Unspecified*** >> ***Component***.
7. In the **Select Component** dialog, select ***rigid block*** and then click ***OK***.
8. Click ***Close***.

Step 22: Create *DEFORMABLE_TO_RIGID_AUTOMATIC to switch the rigid seat to deformable when contact between the seat and block is detected

1. In the **Solver** browser, right-click and select ***Create*** > ****DEFORMABLE_TO_RIGID*** > ****DEFORMABLE_TO_RIGID_AUTOMATIC*** from the context menu. HyperMesh creates and opens a new load collector in the **Entity Editor**.



2. For **Name**, enter `dtor_automatic`.
3. For ***SWSET*** (set number of this automatic switch set), enter `1`.
4. Set ***CODE*** (activation switch code) to ***0***.

   Note: The switch will take place at **[TIME1]**.
5. For **TIME1**, enter `175`.

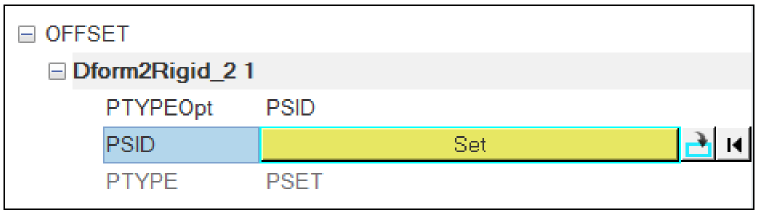
Note: The switch will not take place before this time.

6. For **R2D**, select ***1***.

   Note: On export, the number of rigid parts to be switched to deformable is written to the **R2D** field (card 2, field 6). This number is based on the number of parts in the entity set you select next.

7. Click ***PSID*** >> ***Set***.

   Note: PSIDR2D is the part ID of the part which is switched to a rigid material.



8. In the **Select Set** dialog, select ***set_part_seat*** and then click ***OK***.

Steps 23-27: Review the model’s component data using the Model Browser, Solver Browser or Component Table tool

Method 1: Using the Model browser

Step 23: Display only parts with a particular material (Ex: steel)

1. In the **Model** browser, click .

2. In the **ELASTIC-PLASTIC** folder, **MATL24** folder, right-click on ***steel*** and select ***Isolate*** from the context menu. HyperMesh only displays the components that have the selected material assigned.

3. Review several materials, click , select a material, and scroll through the material using the arrow keys in the **Model** browser. The corresponding parts are automatically isolated in the view.

4. Follow the above steps to select properties using the **Property View** option.

Step 24: Display all components

1. In the **Model** browser, click .

Step 25: Rename a part

1. Right-click on the part you would like to rename, and then select ***rename*** from the context menu.
2. In the editable field, enter a new name for the entity. The part's new name changes in the **Solver** and **Model** browsers.

Step 26: Renumber a part ID

1. In the **Model** browser, click on a part's ***ID*** field. The ID field becomes editable.
2. Enter a number that does not conflict with the existing part IDs, and then press ***Enter***.

Method 2: Using the Solver browser

Step 23: Display only parts with a particular material (Ex: steel)

1. In the **Model** browser, **Materials** folder, right-click on ***Steel*** and select ***Isolate*** from the context menu.
2. In the **Solver** browser, ***SECTION** folder, select components based on properties.

Step 24: Display all components

1. In the **Solver** browser, click the ***MAT** folder.

Step 25: Rename a part

1. In the **Solver** browser, select the part you would like to rename. The **Entity Editor** opens, and displays the part's corresponding data.
2. For **Name**, and enter a new name for the part. The part's new name changes in the **Solver** and **Model** browsers.

Step 26: Renumber a part ID

1. In the **Solver** browser, select the part you would like to change the ID of. The **Entity Editor** opens, and displays the part's corresponding data.
2. For ***ID***, enter a new ID for the part. The part's new ID changes in the **Solver** and **Model** browser.

Method 3: Using the Component Table

Step 23: Display only parts with a particular material (Ex: steel)

1. From the menu bar, click ***Tools*** > ***Component Table***.
2. In the **Components and Properties** dialog, click ***Display*** > ***By Material*** from the menu bar.
3. In the panel area, click ***mats***.
4. Select the material, ***steel***.
5. Click ***Select***.
6. Click ***proceed***. The **Component Table** only displays the components with the material steel assigned. All other components are turned off.
7. To select components using the **By Properties** and **By thickness** options, repeat the above steps.

Step 24: Display all components

1. From the menu bar, click ***Display*** > ***All***. The table displays all of the components in the model.

Step 25: Rename a part

1. From the menu bar, click **Table** > ***Editable***. The table becomes editable. You can edit any of the columns that have a white background. For example, **Part name**, **Part id**, **Thickness**, and so on.
2. Click any **Part name** field. The field becomes editable.
3. Enter a new name for the part.
4. In the **Confirm** dialog, click ***Yes***. The part's new name changes in the **Solver** and **Model** browsers.

Step 26: Renumber a part ID

1. From the menu bar, click **Table** > ***Editable***. The table becomes editable.
2. Click any ***Part Id*** field. The field becomes editable.
3. Enter a new ID that does not conflict with any existing **part IDs**.
4. In the **Confirm** dialog, click ***Yes***. The part's new ID changes in the **Solver** and **Model** browsers.

Step 27: Review the model’s data using the Solver Browser

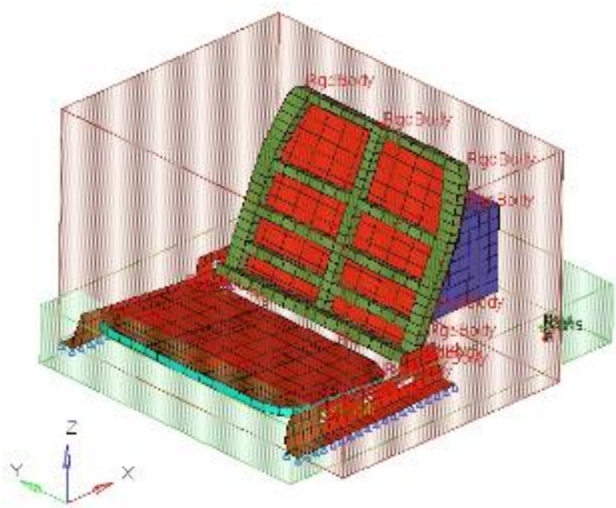
The created solver entities are listed in the **Solver** browser, within their corresponding folders. Use the following options on each entity to help navigate through the model: ***Show***, ***Hide***, ***Isolate,*** and ***Review***.

1. In the **Solver** browser, ***DEFORMABLE_TO_RIGID** folder, right-click on ***dtor*** and select ***Isolate Only*** from the context menu. HyperMesh only displays the entities that are referred in this keyword.
2. Highlight the entities that are referenced by this keyword by right-clicking on ***dtor*** and selecting ***Review*** from the context menu.
3. Right-click on the folder ****BOUNDARY*** and then select ***Show*** from the context menu. HyperMesh displays the entities on which the loads in the folder are defined, as well as the load handles.

Exercise 2: Define Boundary Conditions and Loads for the Seat Impact Analysis

This exercise will help you become familiar with defining LS-DYNA boundary conditions and loads using HyperMesh.

In this exercise, you will define boundary conditions and load data for an LS-DYNA analysis of a vehicle seat impacting a rigid block. The seat and block model is shown in the image below.



Seat and block model

This exercise contains the following three tasks.

- Define gravity acting in the negative z-direction with *LOAD_BODY_Z
- Define the seat's acceleration with *BOUNDARY_PRESCRIBED_MOTION_NODE
- Export the model to an LS-DYNA 970 formatted input file and submit it to LS-DYNA

Step 1: Make sure the LS-DYNA user profile is still loaded

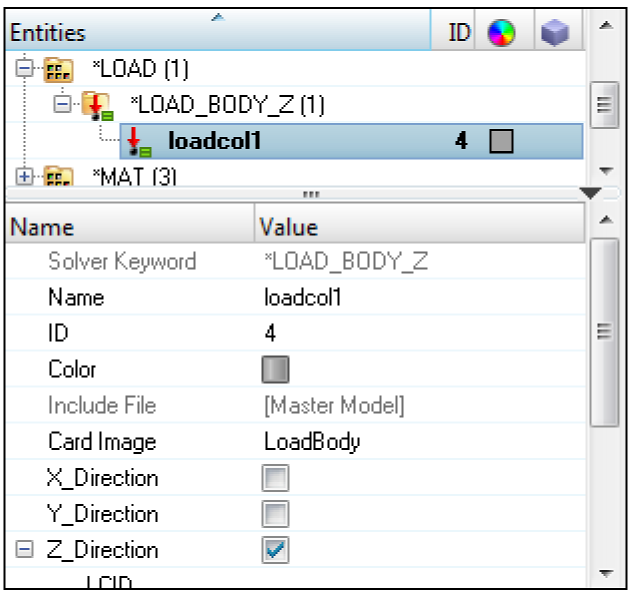
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Retrieve the HyperMesh file

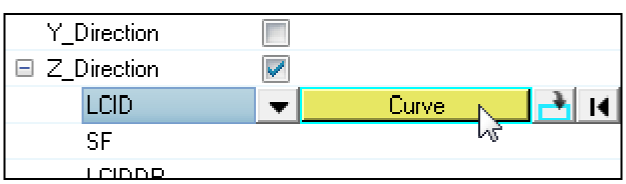
1. To open a model file, click ***File*** > ***Open*** > ***Model*** from the menu bar, or click  on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `seat_2.hm` file. The model appears in the graphics area.
3. Observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).

Step 3: Define gravity acting in the negative z-direction with *LOAD_BODY_Z

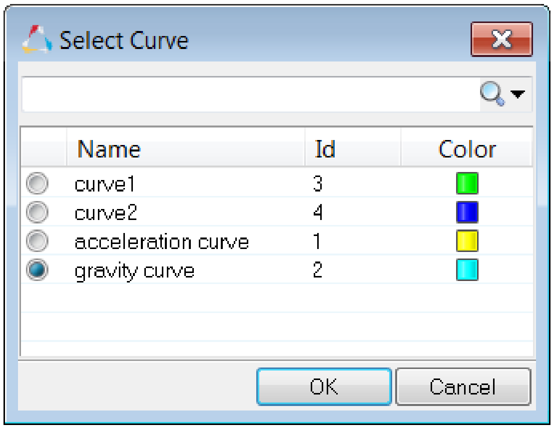
1. In the **Solver** browser, right-click and select ***Create*** > **LOAD** > ***LOAD_BODY_Z*** from the context menu. HyperMesh creates and opens a new load collector in the **Entity Editor**.



2. For **Name**, enter `gravity`.
3. Click ***LCID***, and then click ***curve***.



4. In the **Select Curve** dialog, select ***gravity curve*** and then click ***OK***.

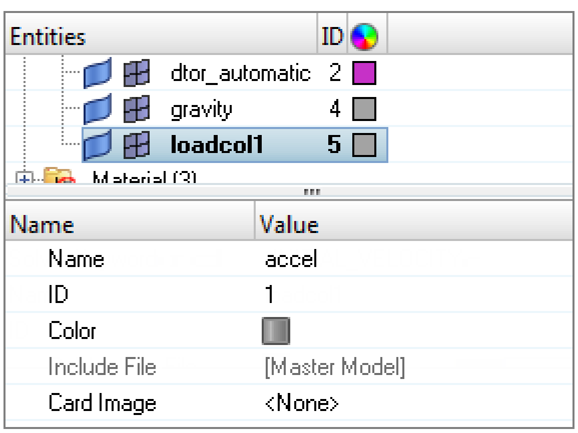


5. For ***SF*** (scale factor for acceleration in z-direction), enter `0.001`.

Steps 4-6: Define the seat's acceleration with *BOUNDARY_PRESCRIBED_MOTION_NODE

Step 4: Create a load collector for the acceleration loads to be created

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



2. For **Name**, enter `accel`.
3. Set **Card Image** to ***<None>***.
4. Optional. Click the ***Color*** icon, and select a color for the load collector.

Step 5: Create acceleration loads on nodes

1. Open the **Accelerations** panel by clicking ***BCs*** > ***Create*** > ***Accelerations*** from the menu bar.
2. Click **load types =**, and select ***PrcrbAcc_S***.

3. Click ***sets***.
4. Select the set, ***accel_nodes***.
5. Click ***select***.
6. Click the ***magnitude=*** switch, and select ***curve, vector***.
7. In the **magnitude=** field, enter `0.001`.

   Note: This is the scale factor for the pre-defined curve to be specified in the next step for the acceleration loads. It will define the seat's acceleration as a function of time.
8. Set the orientation selector to ***x-axis***.

   Note: This is the x-translational degree of freedom.
9. Double-click ***curve***.
10. Select the curve, ***acceleration curve***.
11. In the **magnitude% =** field, enter `1.0E+7`.

    Note: This is the scale factor for the graphical representation of the acceleration loads. It does not affect the actual acceleration value.
12. Click ***create.*** HyperMesh creates the acceleration loads.
13. Click ***return***.

Step 6: Export the model to an LS-DYNA 971 formatted input file

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.
2. In the **Export - Solver Deck** tab, set **File type** to ***Ls-Dyna***.
3. In the **File** field, navigate to your working directory and save the file as `seat_complete.key`.
4. Click ***Export***.

Step 7 (Optional): Submit the LS-DYNA input file to LS-DYNA 971

1. From the **Start** menu on your desktop, open the **LS-DYNA Manager** program.
2. From the **solvers** menu, select ***Start LS-DYNA analysis***.
3. Load the file `seat_complete.key`.
4. Click ***OK*** to start the analysis.

Step 8 (Optional): View the results in HyperView

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