Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4700: Using the PAM-CRASH 2G Interface in HyperMesh

HM-4700: Using the PAM-CRASH 2G Interface in HyperMesh

For this tutorial it is recommended that you complete the introductory tutorial, HM-1000: Getting Started with HyperMesh.

This tutorial introduces the HyperMesh interface to PAM-CRASH 2G. The following exercises are included:

- Load a prepared HyperMesh file
- Select the PAM-CRASH 2G user profile
- Create control cards
- Assign element types
- Define materials
- Define HyperMesh groups: sliding interface
- Define a rigid wall
- Creating boundary conditions
- Create time histories
- Creating a function
- Creating a sensor card
- Exporting a PAM-CRASH 2G data deck from HyperMesh

Model Files

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

Exercise

Step 1: Select the PAM-CRASH 2G user profile

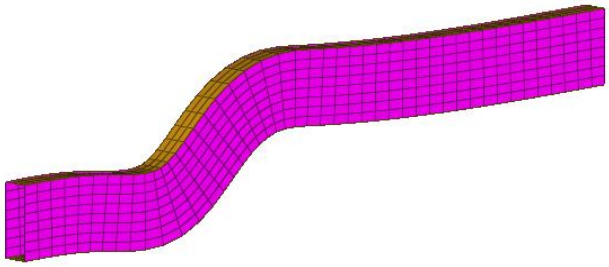
In order to use HyperMesh with a specific solver, the solver user profile must be loaded. Upon opening, you are prompted to select a user profile. In the **User Profiles** dialog, select the ***Pamcrash2G2012*** profile.

Selecting the PAM-CRASH 2G user profile sets the FE input reader to PAM-CRASH 2G and loads the PAM-CRASH 2G 2012 FE output template. It also loads the PAM-CRASH 2G **Utility** menu, which contains numerous tools specific to this interface. The graphical user interface is tailored to PAM-CRASH 2G users.

Step 2: Load a prepared HyperMesh file

A prepared model with elements and nodes is included in the `<install_directory>\tutorials\hm\interfaces\pamcrash` directory. The file name of the example is `rail-dyna.hm`. This is the basic example on which the tutorial is based.

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 `rail-dyna.hm` file.



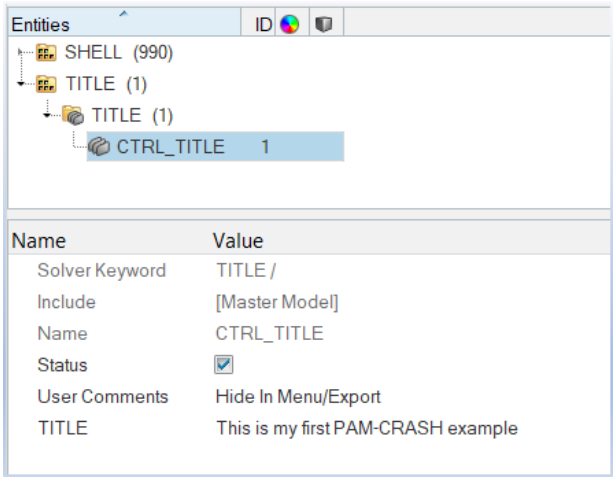
Steps 3-6: Create Control Cards for PAM-CRASH 2G

This section explains how to create control cards for the CONTROL SECTION of the PAM-CRASH 2G deck.

> **Note:** The settings of the control cards influence the default values for defining materials. No PAM-CRASH 2G deck can be executed without error if control cards are undefined.

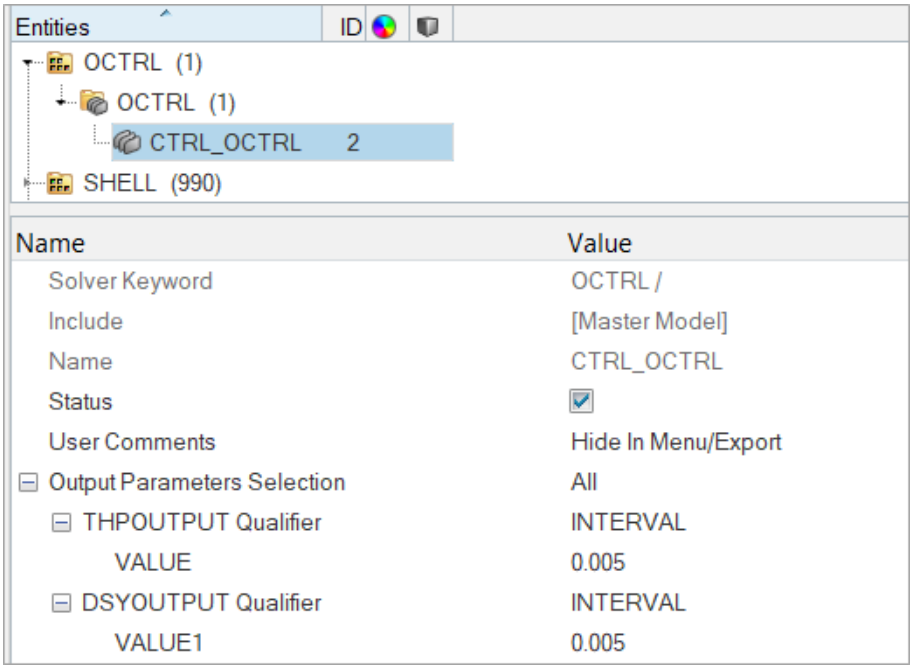
Step 3: Define the title card

1. In the **Solver** browser, right-click and select ***Create*** > ***CONTROL CARDS*** > ***TITLE*** from the context menu. A new control card opens in the **Entity Editor**.
2. In the **TITLE** field, enter `This is my first PAM-CRASH example`.

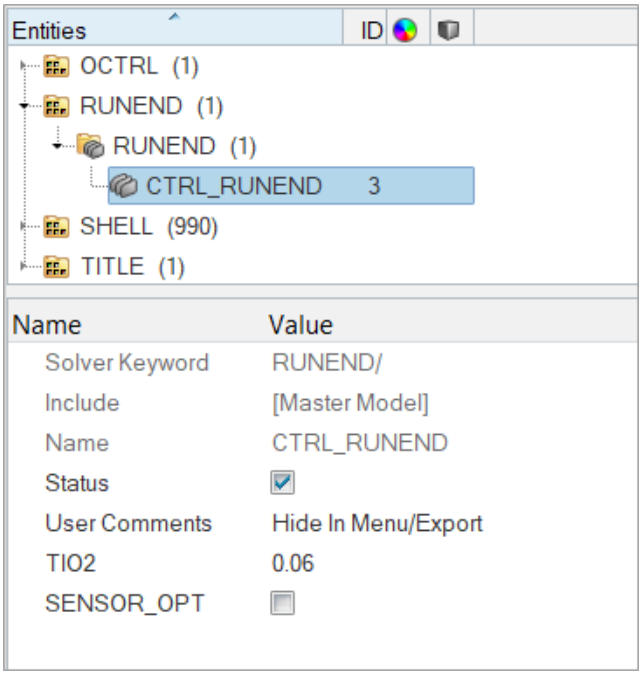


Step 4: Define the output control cards

1. In the **Solver** browser, right-click and select ***Create*** > ***CONTROL CARDS*** > ***OCTRL*** from the context menu. A new control card opens in the **Entity Editor**.
   Note: The OCTRL control card defines output control parameters.
2. For **THPOUTPUT**, enter `0.005`.
3. For **DSYOUTPUT**, enter `0.005`.

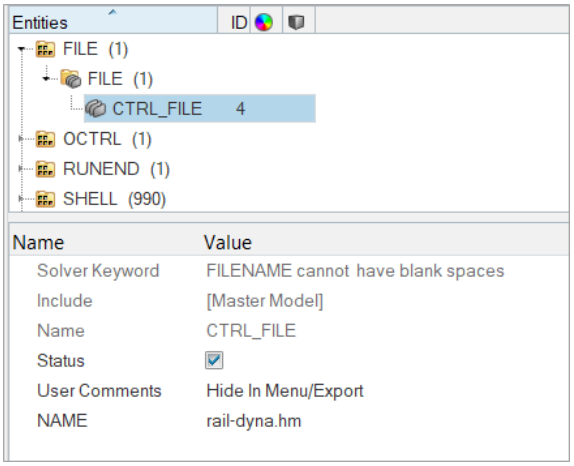


4. In the **Solver** browser, right-click and select ***Create*** > ***CONTROL CARDS*** > ***RUNEND*** from the context menu. A new control card opens in the **Entity Editor**.
   Note: The RUNEND control card defines end of run parameters.
5. For **TIO2**, enter `0.06`.



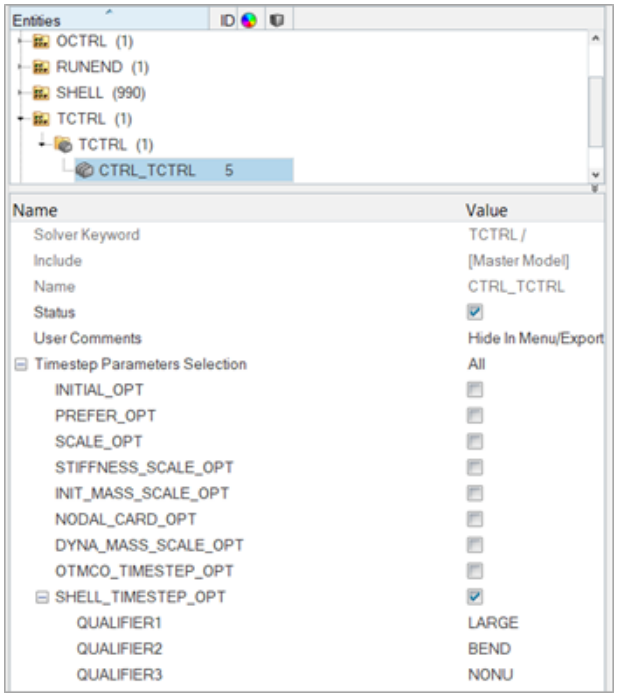
Step 5: Define the file optional keyword

1. In the **Solver** browser, right-click and select ***Create*** > ***CONTROL CARDS*** > ***FILE*** from the context menu. A new control card opens in the **Entity Editor**.
2. For **NAME**, enter `rail-dyna.hm`.



Step 6: Define the time step optional keyword

1. In the **Solver** browser, right-click and select ***Create*** > ***CONTROL CARDS*** > ***TCTRL*** from the context menu. A new control card opens in the **Entity Editor**.
2. Select the ***SHELL_TIMESTEP_OPT*** checkbox.
3. Set **QUALIFIER 1** to ***LARGE*** (default).
4. Set **QUALIFIER 2** to ***BEND*** (default).



Step 7: Assign Element Types for PAM-CRASH 2G

Depending on the analysis requirement, the HyperMesh basic element type can be changed.

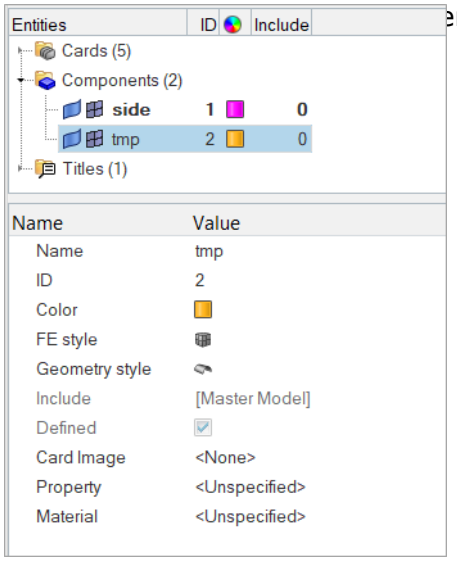
For example, a quad4 can be a SHELL or a MEMBR element. The tria3 element can be a TRIA_C, SHELL, or MEMBR element. The tetra4, the penta6, and the hexa8 elements define the SOLID elements of PAM-CRASH. Properties can be added for the selected element type using control cards.

1. Open the **Element Type** panel by clicking ***Mesh*** > ***Assign*** > ***Element Type*** from the menu bar.
2. Go to the **2D & 3D** subpanel.
3. Click ***quad4 =*** and select **SHELL**.
4. Click ***elems*** >> ***all***.
5. Click ***update***.
6. Click ***return*** to exit the panel.

Steps 8-10: Define material and /PART cards for PAM-CRASH 2G

Before proceeding with the tutorial you will rename the component tmp.

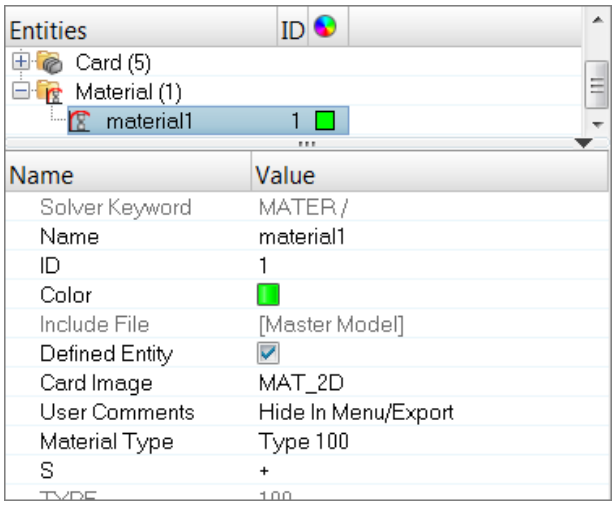
1. In the **Model** browser, **Component** folder, click ***tmp*** component. The **Entity Editor** ent's card data.



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

Step 9: Define a Material Type 102

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

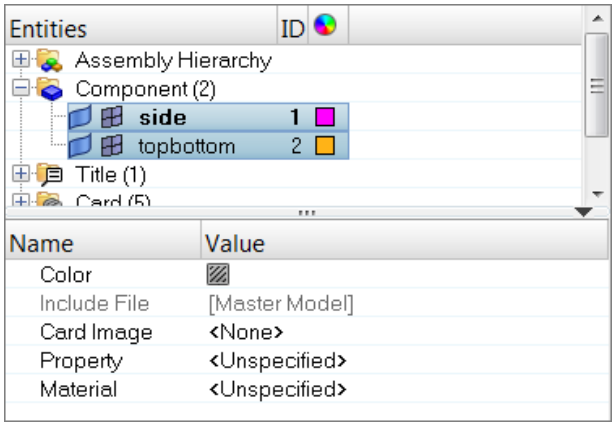


2. For **Name**, enter `new mat`.
3. Set ***Card Image*** to ***MAT_2D***.

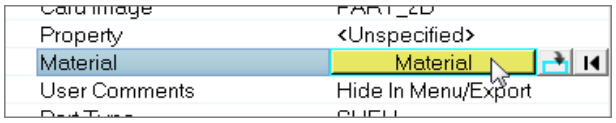
   Note: The template provides MAT_1D, MAT_2D, and MAT_3D dictionaries. Material types from 200 to 230 are defined with MAT_1D. Materials types from 100 to 151 are defined with MAT_2D. Material types from 1 to 41 are defined with MAT_3D. To switch the material type, use the card previewer.
4. Set **Material Type** to ***Type 102***.
5. For **RHO**, enter `7.85e-9`.
6. For **E**, enter `20000`.
7. For **SIGMay**, enter `250`.
8. For **NU**, enter `0.3`.

Step 10: Assign material and thickness to side and topbottom collectors

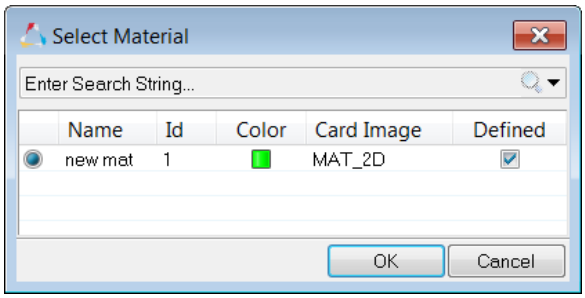
1. In the **Model** browser, **Component** folder, select ***side*** and ***topbottom***. The **Entity Editor** opens and displays the selected component's common card data.



2. Set **Card Image** to ***Part_2D***.
3. For **Material**, click ***Unspecified*** >>***Material***.



4. In the **Select Material** dialog, select ***new mat*** and then click ***OK***. HyperMesh assigns the **new mat** material to the selected components.



5. For ***h*** (thickness), enter `2.5`.

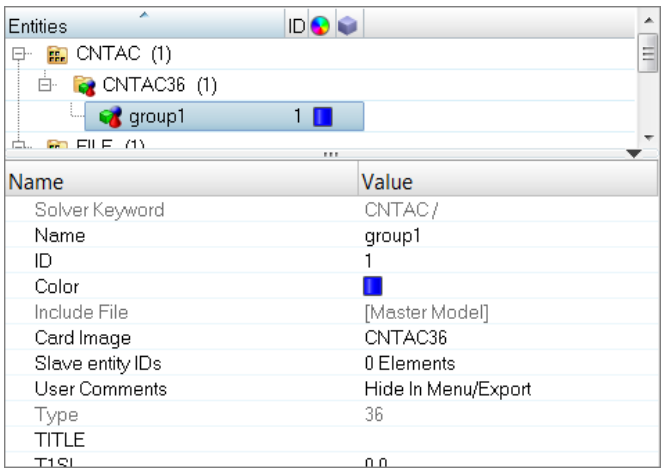
Steps 11-14: Define HyperMesh Groups: Sliding Interface for PAM-CRASH 2G

This section describes how to define a self contacting sliding interface. A second interface is defined only for tutorial purposes.

The procedure below explains how to define a type 36 self contacting sliding interface.

Step 11: Define the group

1. In the **Solver** browser, right-click and select ***Create*** > ***CNTAC*** > ***CNTAC36*** from the context menu. A new group opens in the **Entity Editor**.

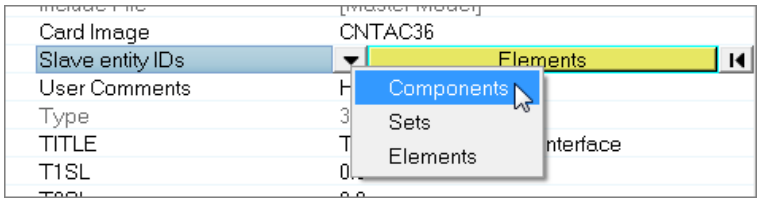


2. For **Name**, enter `self_impact`.
3. Select a new color for the group.
4. For **SLFACM**, enter `1.0`.

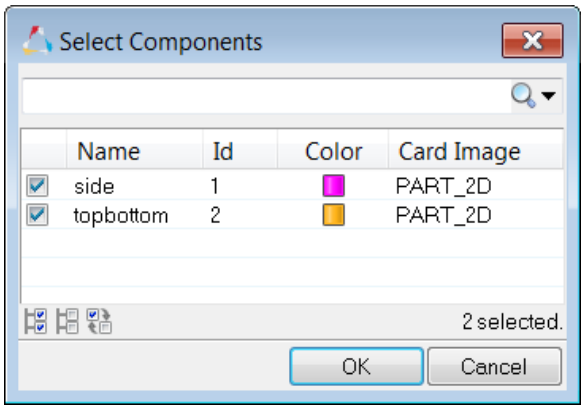
Step 12: Add the slave components

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

1. Click ***Slave entity IDs***.
2. Set the entity selector to ***Components***.



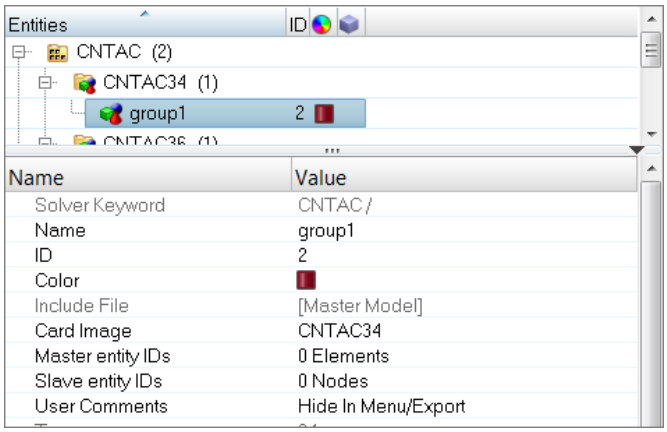
3. Click ***Components***.
4. In the **Select Components** dialog, select ***side*** and ***topbottom*** and then click ***OK***.



Step 13: Define an additional contact

This procedure explains how to define a type 34 master slave (element - node) contact.

1. In the **Solver** browser, right-click and select ***Create*** > ***CNTAC*** > ***CNTAC34*** from the context menu. A new group opens in the **Entity Editor**.

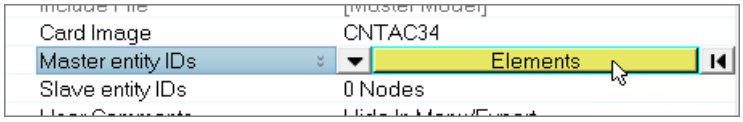


2. For **Name**, enter `masterslave`.
3. Select a new color for the group.
4. For **SLFACM**, enter `1.0`.

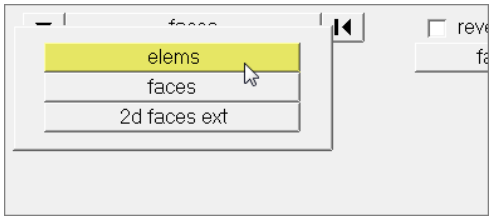
Step 14: Add the master elements and slave nodes

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

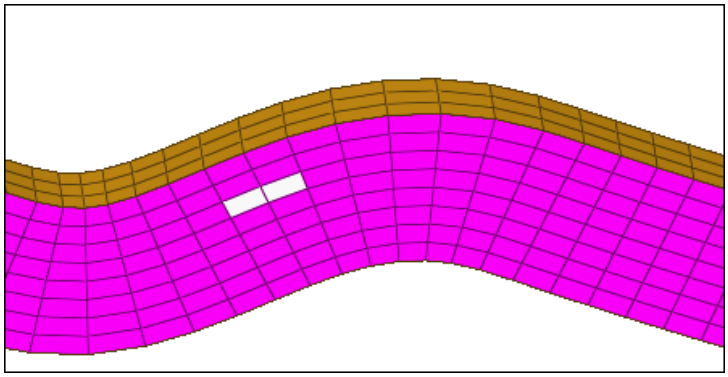
1. For **Master entity IDs**, click ***0 Elements*** >> ***Elements***.



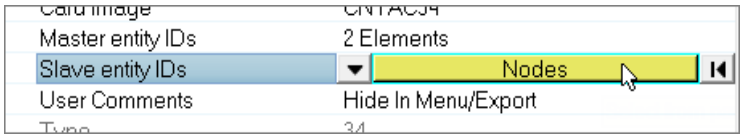
2. In the panel area, set the switch to ***elems***.



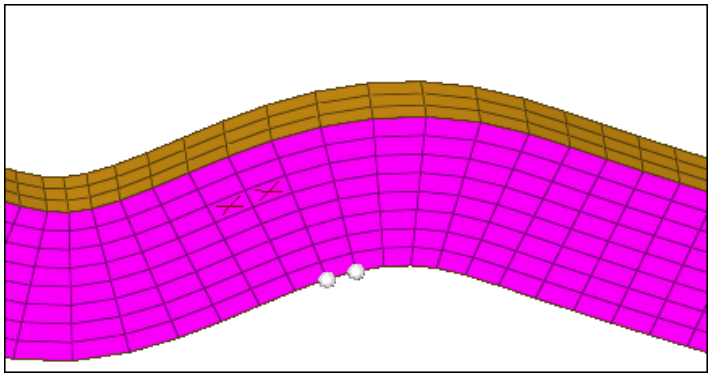
3. Using the **elems** selector, select two elements.



4. Click ***add***.
5. Click ***return***.
6. For **Slave entity IDs**, click ***0 Nodes*** > ***Nodes***.



7. In the panel area, set the switch to ***nodes***.
8. Using the **nodes** selector, select two nodes.



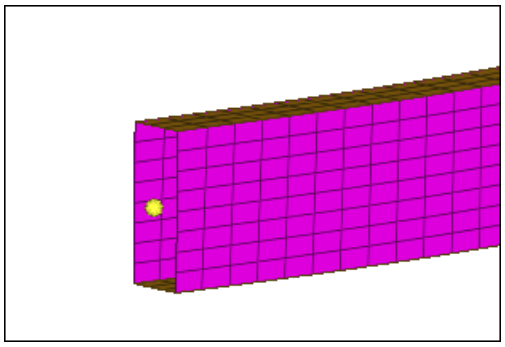
9. Click ***add***.
10. Click ***return***. The master elements are marked with and X, and the slave nodes displays.

Steps 15-19: Define a Rigid Wall for PAM-CRASH 2G

This section explains how to define a type 4 infinite rigid wall with a base node at -1.00, 0.0, 0.0.

Step 15: Create a base node for the rigid wall

1. From the menu bar, click ***Geometry*** > ***Create*** > ***Nodes*** > ***XYZ***.
2. For **x**, enter `-1.0`.
3. For **y**, enter `0.0`.
4. For **z**, enter `0.0`.
5. Click ***create***.

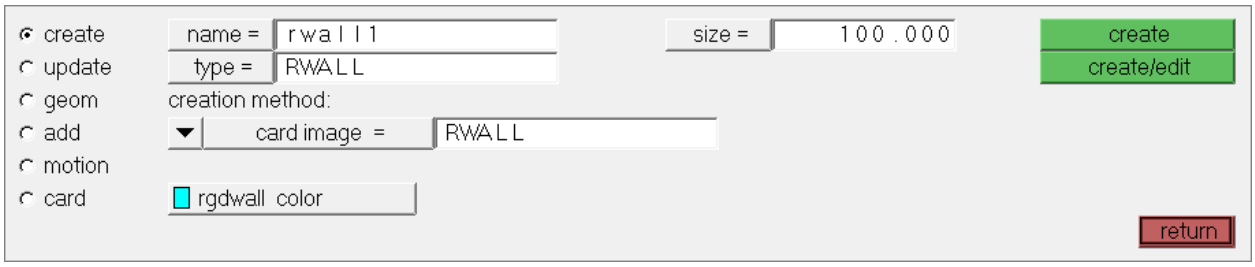


6. Click ***return*** to exit the panel.

Step 16: Create and define the rigid wall card

1. From the menu bar, click ***BCs*** > ***Create*** > ***Rigid Walls***.
2. In the **Rigid Walls** panel, **Name** field, enter rwall1.
3. Click ***type =*** and select ***RWALL***.
4. Click ***rgdwall color*** and select a color.
5. In the **size =**, enter `100`.

   Note: This specifies the display size of the rigid wall.

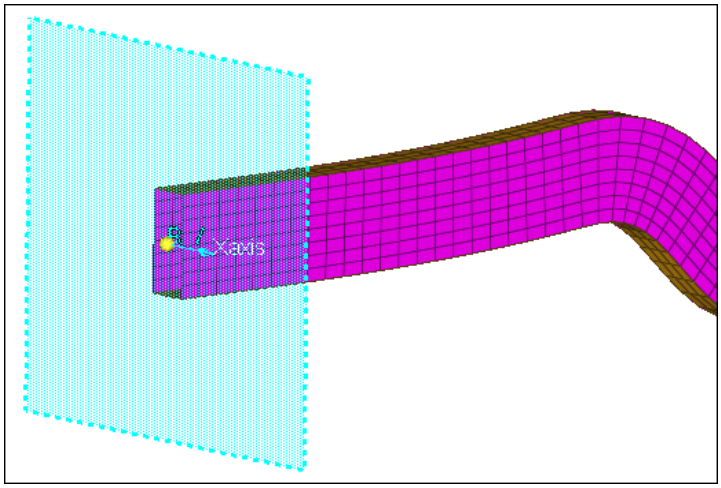


6. Click ***create***.

Step 17: Define rigid wall geometry

1. Go to the **geom** subpanel.
2. Double-click **name=**, and select ***rwall1***.
3. Set the switch after **shape** to ***plane***.
4. Set the toggle after **shape** to ***infinite***.
5. Under **normal vector**, set the switch to ***vectors***.
6. Click the second switch and select ***x-axis***.

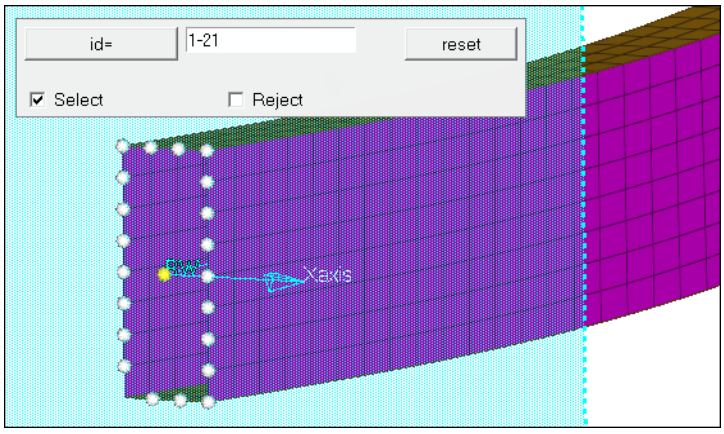
7. Using the **base node** selector, select the node you created in step 15.
8. Click ***update***. The rigid wall displays.



Step 18: Add slave nodes for the rigid wall

1. Go to the **add** subpanel.
2. Set **slaves** to ***nodes***.
3. Click ***nodes*** >> ***by id***.
4. In the **id** field, enter `1-21` and then press ***ENTER***.

   Note: 21 nodes at the interface of the rail and the rigid wall highlight. One of the nodes was not selected.

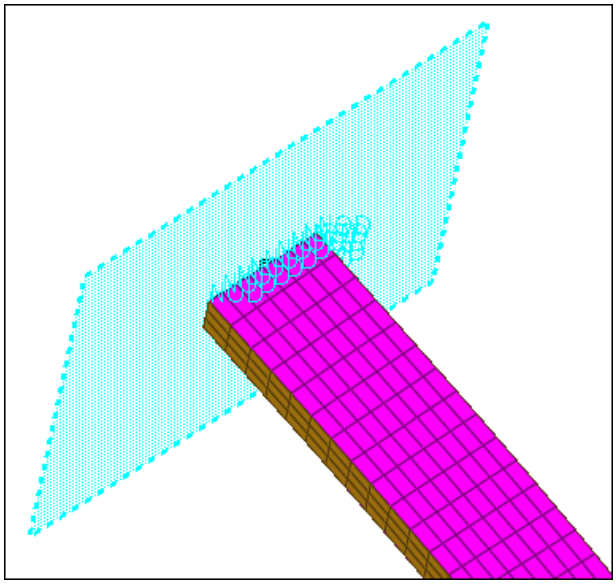


5. Click the node that was not highlighted.

   Or

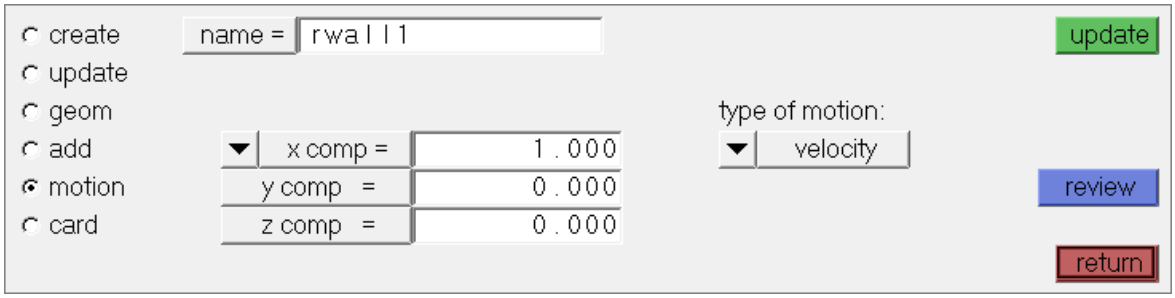
   Click ***nodes*** >> ***id***, and enter `1012` in the **id** field.

6. Click ***add***. The selected nodes are now set as slaves.



Step 19: Add motion to the rigid wall

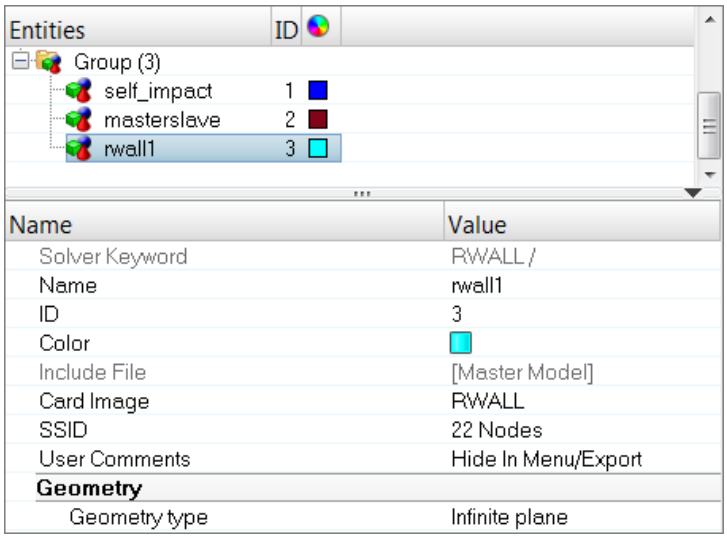
1. Go to the **motion** subpanel.
2. Set the switch below **name** to **components**.
3. In the ***x comp*** field, enter `1.0`.
4. Set **type of motion** to **velocity**.



5. Click ***update***.
6. Click ***return*** to exit the panel.

Step 20: Define attributes in the card previewer

1. In the **Model** browser, **Group** folder, click ***rwall1***. The **Entity Editor** opens and displays the group's card data.



2. Set **Friction type flag** to ***No Sliding***.
3. Set **Rigid Wall Descriptor – Plane Type** to ***Type 4***.

   Note: The parameters in the **Entity Editor** according to the definitions made. It is now possible to define the mass and the initial velocity for moving rigid wall with finite mass.
4. For **mRW**, enter `1`.
5. For **VINIT**, enter `2000`.

Steps 21-22: Create boundary conditions for PAM-CRASH 2G

This section explains how to create model boundary conditions.

Step 21: Create a load collector

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**.

   Note: The new load collector becomes the current collector. Any new loads created will be placed in this collector.
2. For **Name**, enter `boundary conditions`.
3. Select a **Card image**.
4. Click the ***Color*** box, and select a color.

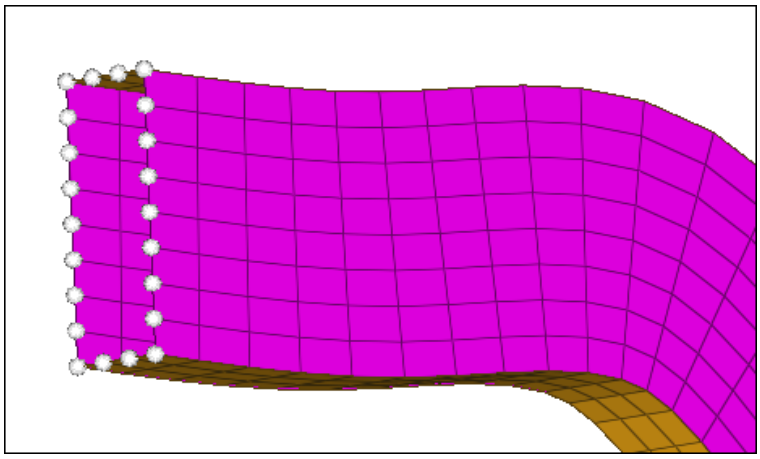
Step 22: Specify the constraints

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

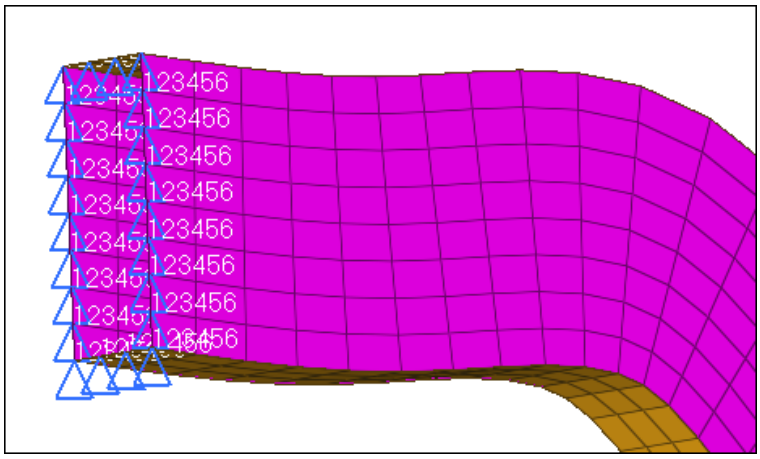
2. Click ***load types***, and select ***BOUNC***.

   Note: All constraints that are now created will be displacement boundary conditions.

3. Click ***nodes*** >> ***by id***.
4. In the **id** field, enter `990-1011`.
5. Press ***ENTER***. 22 nodes highlight.



6. In the ***size*** field, enter `10`.
7. Click ***create***. Constraints are now added to the selected nodes.



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

Steps 23-26: Create Time Histories for PAM-CRASH 2G

For PAM-CRASH 2G, time histories may be defined for nodes, elements, and local coordinate systems. For this exercise, you will only create time histories for some nodes and elements. The operation is the same for any type of time history that is created.

Step 23: Create a node time history card

1. In the **Model** browser, **Load Collector** folder, right-click on ***Boundary Conditions*** and select ***Hide*** from the context menu. The display of loads turns off.
2. From the menu bar, click ***Setup*** > ***Create*** > ***Output Blocks***.
3. In the **name** field, enter `node_thp`.
4. Using the **nodes** select, select a few nodes.
5. Click ***create***. The time history for nodes is created.

Step 24: Create an element time history card

1. In the **name** field, enter `elem_thp`.
2. Set the entity selector to ***elems***.
3. Select a few elements.
4. Click ***create***.

Step 25: Review time histories entities

1. Click ***review***.
2. Select ***elem_thp***. The entities associated with this time history highlight.
3. Click ***return*** to exit the panel.

Step 26: View the time history card image

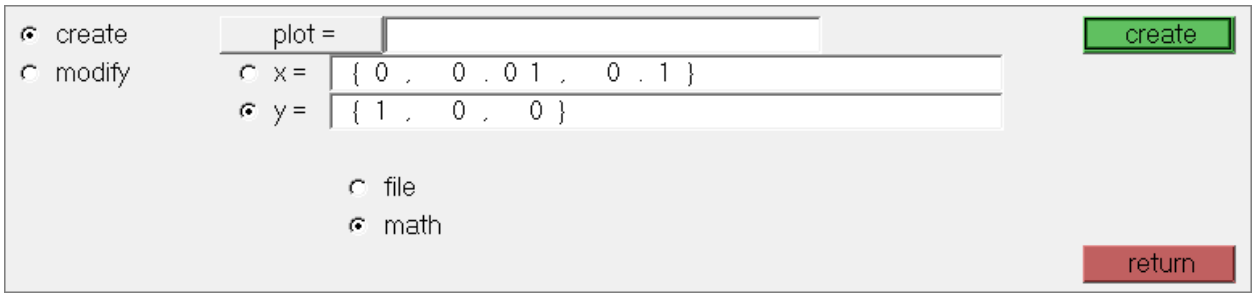
1. In the Model browser, Right click on the ***elem_thp output block*** in the **Model Browser** and click ***Card Edit***.

   The time history card is displayed as it will look in the output.
2. Click ***return*** twice to exit the panel.

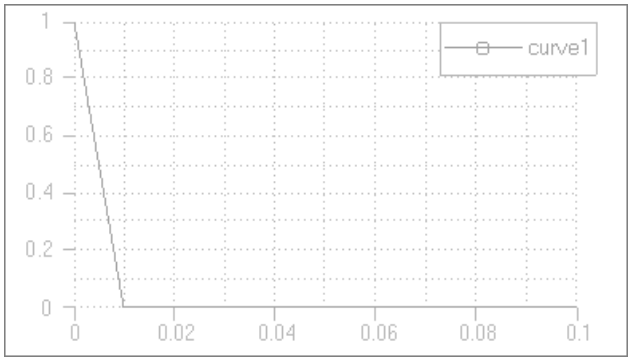
Step 27: Create a function

This section describes how to generate curves, which corresponds to the function cards /FUNCT in PAM-CRASH2G. This curve should serve as a function for a logical sensor switching on and off. At time=0, the sensor is on, at time=0.01 the sensor is switched off.

1. From the menu bar, click ***XY Plots*** > ***Create*** > ***Curves*** > ***Single Curve***.
2. Select ***math***.
3. In the **x =** field, enter `{0, 0.01, 0.1}` (including the brackets).
4. In the **y =** field, enter `{1, 0, 0}` (including the brackets).



5. Click ***create***. HyperMesh generates a curve.

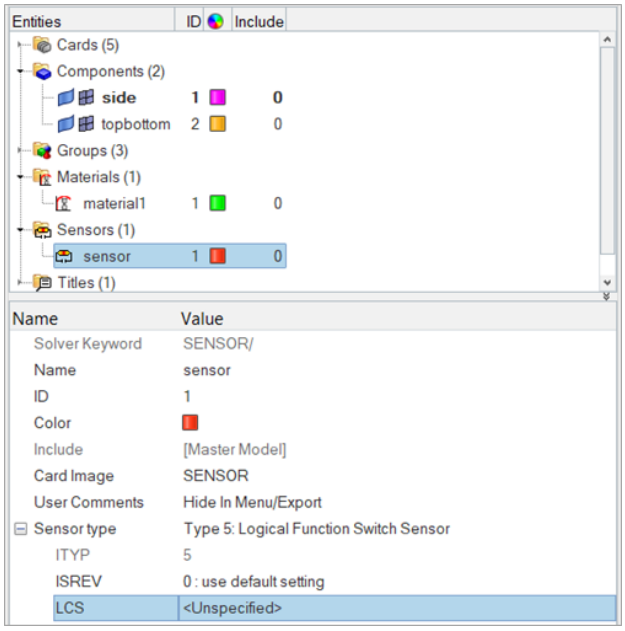


6. Click ***return***

Step 28: Create a sensor card

Sensors are implemented as properties in HyperMesh. In this example we refer to the curve defined in the preceding Help topic.

1. From the **Analysis** page, click ***safety*** >> ***sensors***.
2. In the ***name =*** field, enter `sensor`.
3. Click ***card image =***, and select **SENSOR**.
4. Click ***create/edit***.The **Card Image** dialog opens.
5. Set **Sensor type** to **Type 5** (logical function switch Sensor).
6. Click ***LCS*** >> **curve1**.



8. Click ***return*** twice to exit the panels.

Step 29: Exporting a PAM-CRASH 2G data deck from HyperMesh

This section explains how to generate a PAM-CRASH 2G input deck from HyperMesh.

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***. The **Export - Solver Deck** tab opens.
2. In the **File** field, enter `rail.pc`.

   Note: `rail.pc` is the PAM-CRASH2G file you will create.
3. Set **Template** to `Pamcrash2G2012`.
4. Click ***Export***. HyperMesh writes the deck, and a message in the Status bar indicates the process is completed.