Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4620: Rigid Wall, Model Data, Constraints, Cross Section, and Output using DYNA

altairhyperworks.com

HM-4620: Rigid Wall, Model Data, Constraints, Cross Section, and Output using DYNA

In this tutorial, you will learn how to:

- Create *PART_INERTIA for the vehicle mass component to partially take into account the inertia properties and mass of the missing parts.
- Create velocity on all nodes except barrier nodes with *DEFINE_BOX and *INITIAL_VELOCITY.
- Make the closest row of nodes of the crash boxes a part of the vehicle mass rigid body with *CONSTRAINED_EXTRA_NODES.
- Create a contact between the crash boxes, the bumper, and the barrier with *CONTACT_AUTOMATIC_GENERAL.
- Specify the output of resultant forces for a plane on the left interior and exterior crash boxes with *DATABASE_CROSS_SECTION_PLANE.
- Create a stationary rigid wall to constrain further movement of the barrier after impact with *RIGIDWALL_PLANAR_FINITE.
- Specify some nodes to be output to the ASCII NODOUT file with *DATABASE_HISTORY_NODE.

*PART_INERTIA

The INERTIA option enables inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies only.

When importing a LS-DYNA model into HyperMesh, the *PART_INERTIA IRCS parameter value is changed from 0 to 1. The inertia components are changed from global to local axis. This allows inertia components to be automatically updated when *PART_INERTIA elements are translated or rotated. When selecting *PART_INERTIA elements to translate or rotate, select elements by comp. This selection method ensures the inertia properties are automatically updated.

*CONSTRAINED_EXTRA_NODES

This card defines extra nodes to be part of a rigid body. In HyperMesh, it is created from the Solver browser or Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

*DATABASE_CROSS_SECTION_(Option)

*DATABASE_CROSS_SECTION_(Option) defines a cross section for resultant forces written to the ASCII SECFORC file. The options are PLANE and SET.

For the PLANE option, a cutting plane must be defined. For best results, the plane should cleanly pass through the middle of the elements, distributing them equally on either side.

The SET option requires the equivalent of the automatically generated input via the cutting plane to be identified manually and defined in sets. All nodes in the cross-section and their related elements contributing to the cross-sectional force resultants should be defined in sets.

*DATABASE_CROSS_SECTION_SET and *DATABASE_CROSS_SECTION_PLANE are created from the Solver browser or Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

*RIGIDWALL

A *RIGIDWALL provides a method for treating contact between a rigid surface and nodal points of a deformable body.

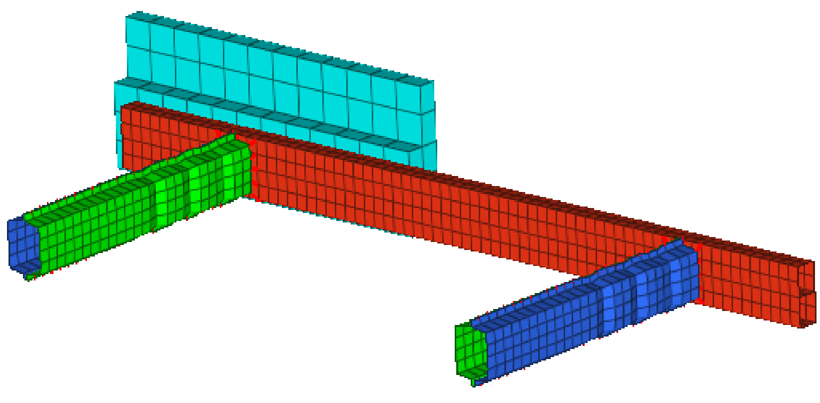
In HyperMesh, *RIGIDWALL keyword cards are created from the Solver browser or Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

Model Files

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

Exercise: Set Up the Bumper Model for Impact Analysis

In this exercise, you will define model data, loads, constraints, a cross section, a rigid wall, and output for an LS-DYNA analysis of a bumper in a 40% frontal offset crash. The bumper model is shown in the image below.



Step 1: Load the LS-DYNA user profile

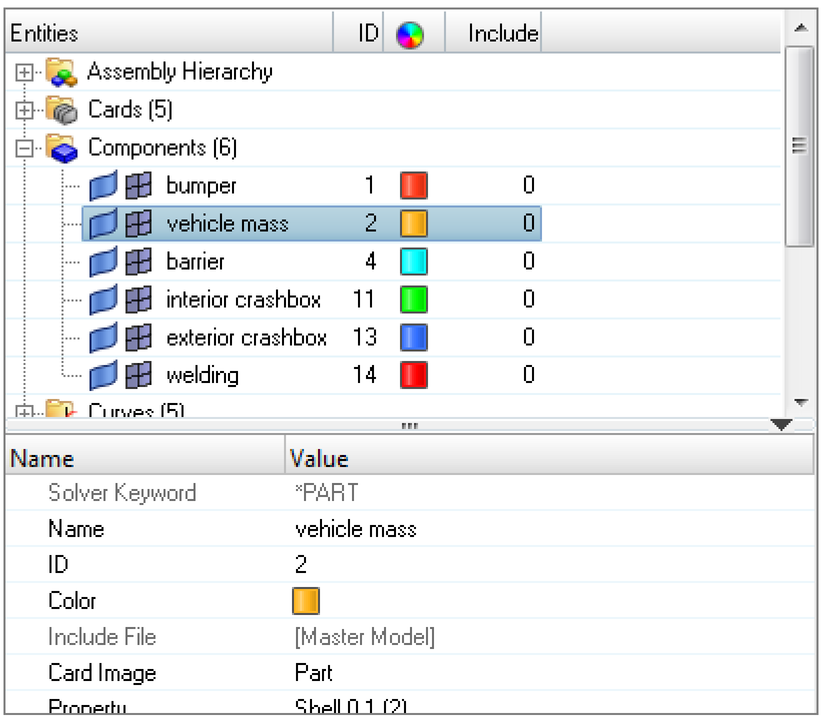
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Import the LS-DYNA model `bumper_start.key`

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

Step 3: Define *PART_INERTIA for the vehicle mass component to partially take into account the inertia properties and mass of the missing parts

1. In the **Model** browser, **Component** folder, click **vehicle mass**. The **Entity Editor** opens, and displays the component's card data.



2. In the **Entity Editor**, edit the component's card data.
   a. Set **Options** to ***Inertia***.
   b. For **XC** (X coordinate of center of mass), enter `700`.
   c. For **YC** (Y coordinate of center of mass), enter `0.0`.
   d. For **ZC** (Y coordinate of center of mass), enter `170`.

e. For **TM** (translational mass), enter `800`.

f. For **IXX** (XX component of target inertia), enter `1.5E+07`.

g. For **IXY** (XY component of target inertia), enter `-5.0E+03`.

h. For **IXZ** (XZ component of target inertia), enter `-8.0E+06`.

i. For **IYY** (YY component of target inertia), enter `5.0E+07`.

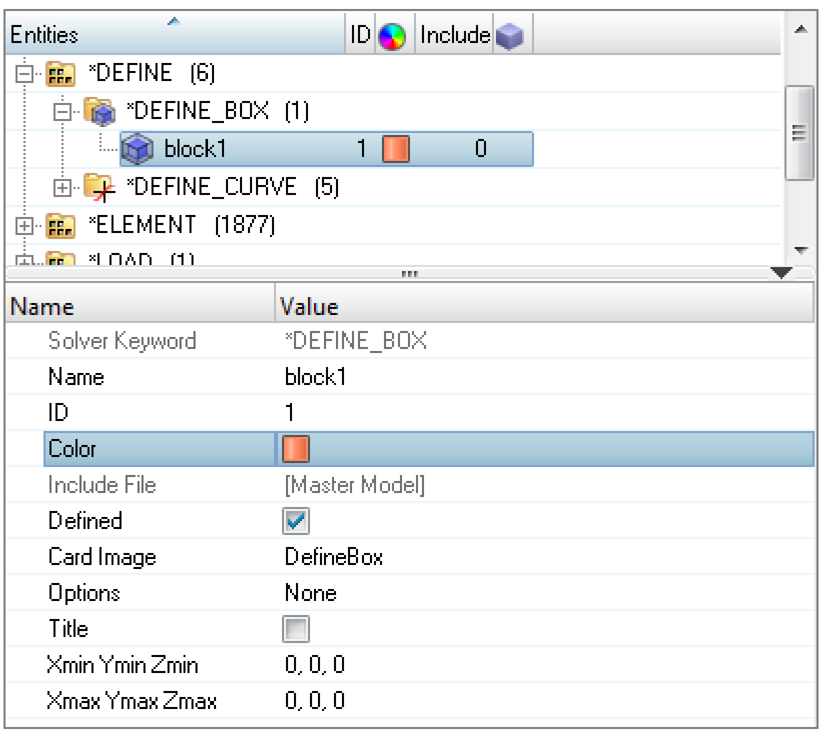
j. For **IYZ** (YZ component of target inertia), enter `-900`.

k. For **IZZ** (ZZ component of target inertia), enter `6.0E+07`.

l. For **VTX** (Initial translational velocity of rigid body in x direction), enter `-10`.

Step 4: Create a *DEFINE_BOX that contains all nodes except barrier nodes

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*** > ****DEFINE*** > ****DEFINE_BOX*** from the context menu. A new block opens in the **Entity Editor**.

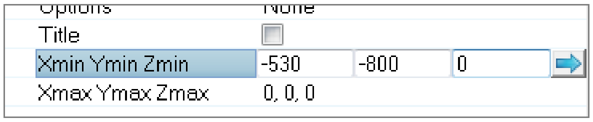


3. In the **Entity Editor**, define the block.

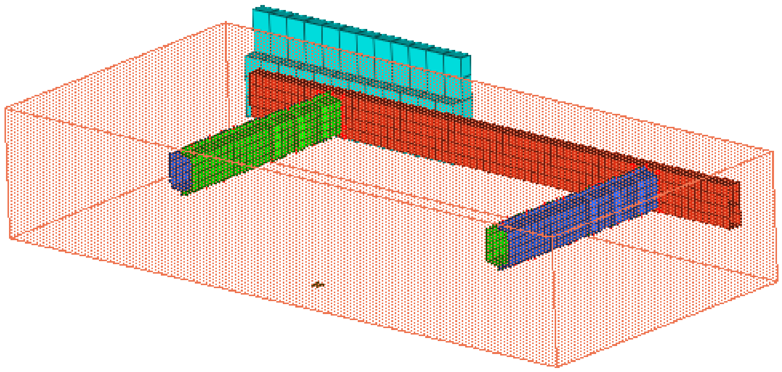
a. For **Name**, enter `box velocity`.

b. Optional. Click the ***Color*** icon, and select a color for the block.

c. For **Xmin Ymin Zmin**, enter `-530`, `-800`, `0`.



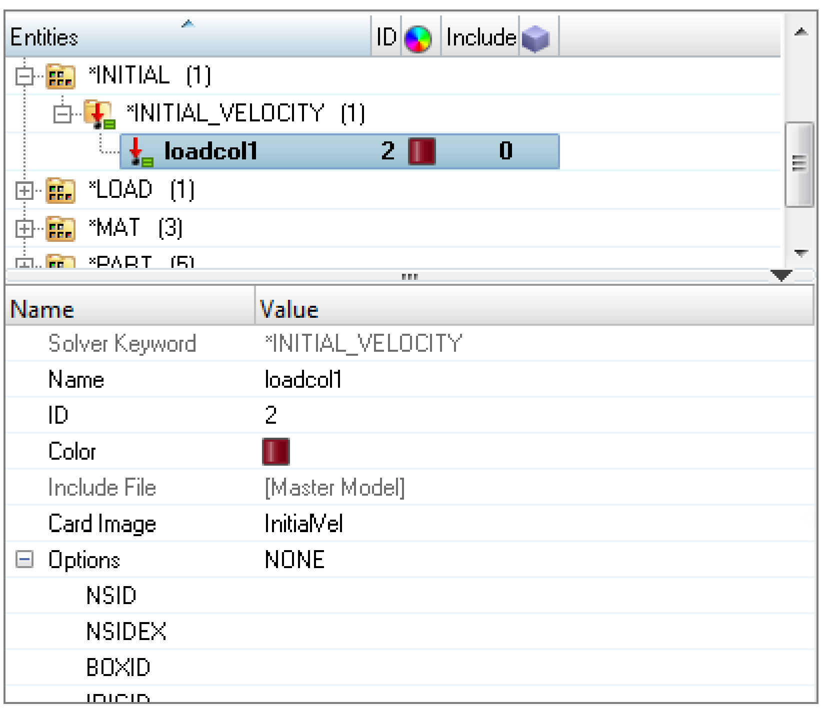
d. For **Xmax Ymax Zmax**, enter `200`, `800`, `300`.



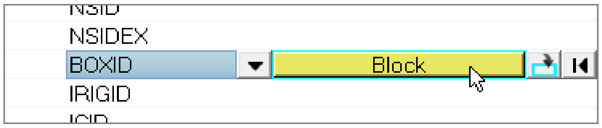
Step 5: Create initial velocity on all nodes except barrier nodes

A velocity boundary condition can also be created on a set of nodes from the Solver browser or Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

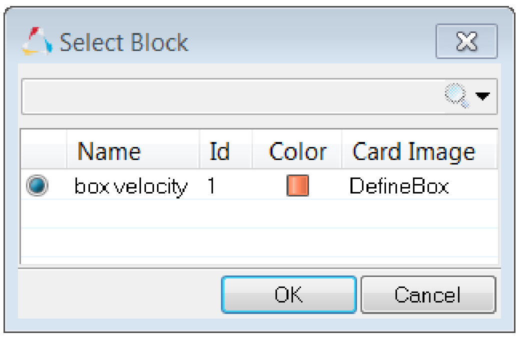
1. In the **Solver** browser, right-click and select ***Create*** > ****INITIAL*** > ****INITIAL_VELOCITY*** from the context menu. A new load collector opens in the **Entity Editor**.



2. In the **Entity Editor**, define the load collector.
   a. For **Name**, enter `velocity`.
   b. For **VX** (Initial velocity in the global X direction), enter `-10`.
   c. Click ***BOXID***, and then click ***Block***.



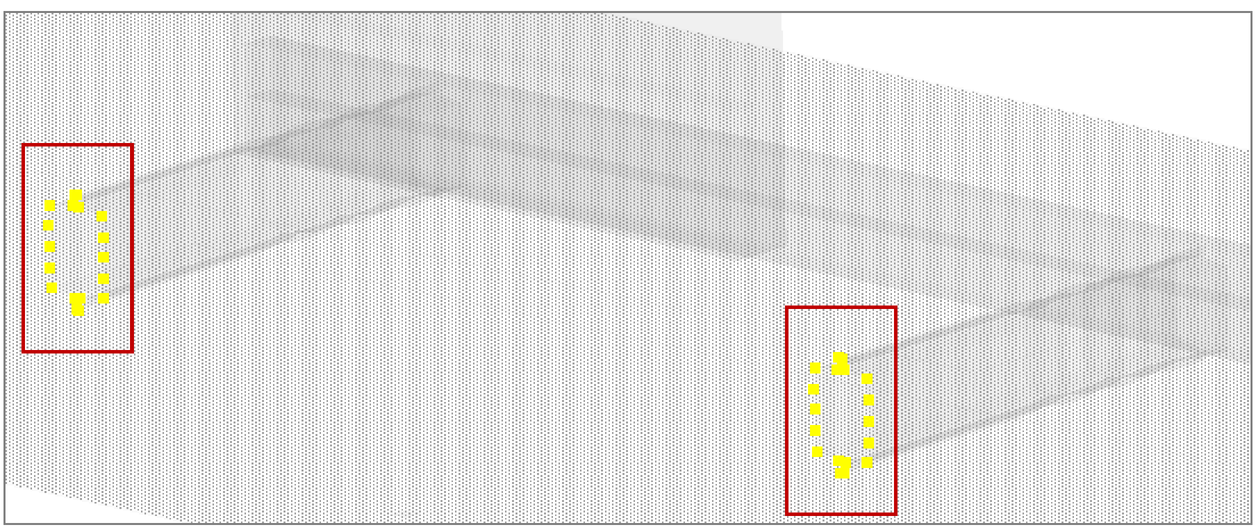
   d. In the **Select Block** dialog, select ***box velocity*** and then click ***OK***.



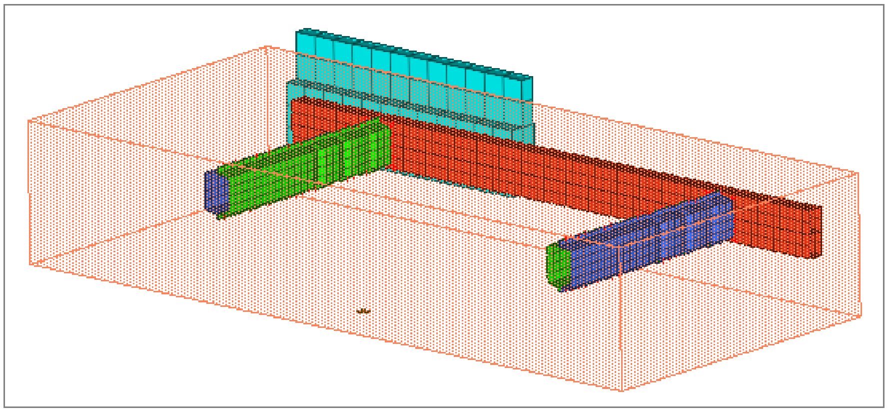
Step 6: View the closest nodes which are in the pre-defined node entity set (*SET_NODES_LIST) named Constrain Vehicle

Method 1

1. In the **Solver** browser or **Model** browser, right-click on ***Constrain Vehicle*** and select ***Review*** (press ***Q***) from the context menu. The set's nodes highlight.



2. Return all of the entities to their original display color by right-clicking on ***Constrain Vehicle*** and selecting ***Review*** (press ***Q***) from the context menu.

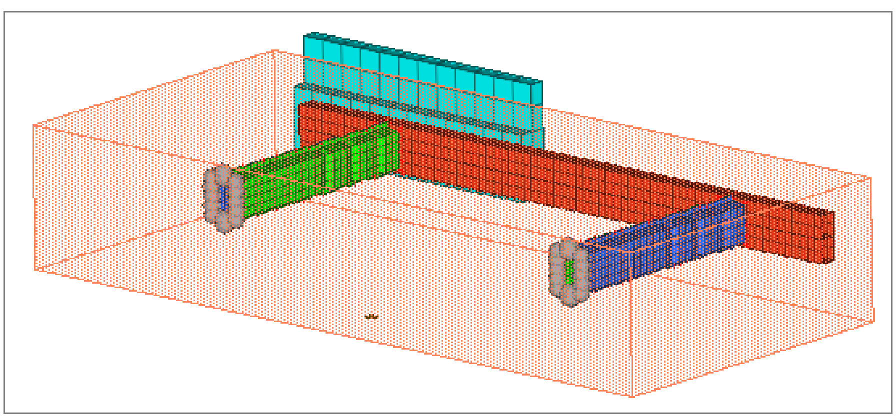


Method 2

1. From the menu bar, click ***Tools*** > ***Edit*** > ***Sets***.
2. In the **Entity Sets** panel, click ***review***.
3. Set the **display RLs**/**hide RLs** toggle to ***hide RLs***.

   Note: This option filters all nodal rigid body sets from the list.

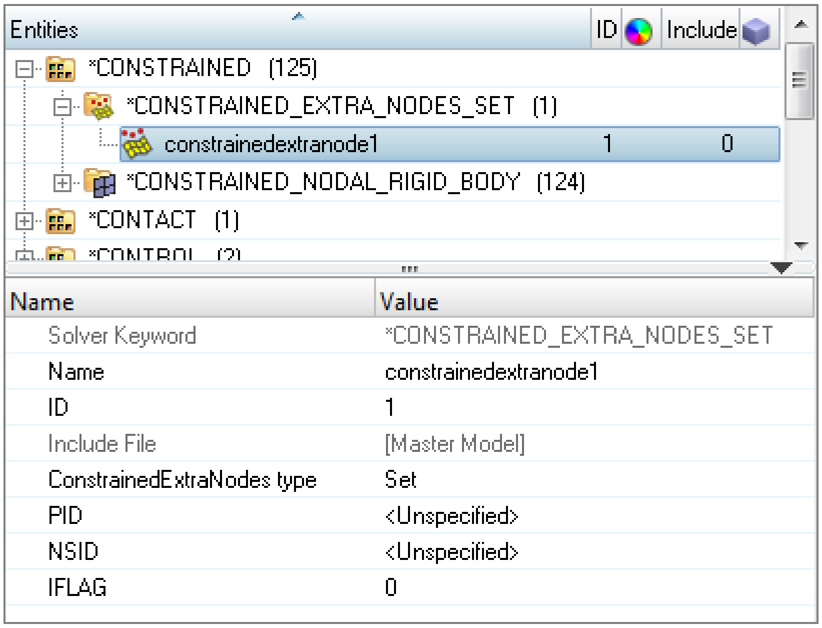
4. Select the set, **Constrain Vehicle**. The set's nodes highlight.



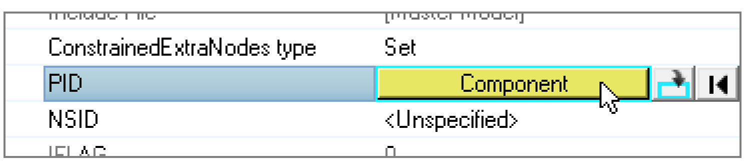
5. Close the panel by clicking ***return***.

Step 7: Create *CONSTRAINED_EXTRA_NODES_SET

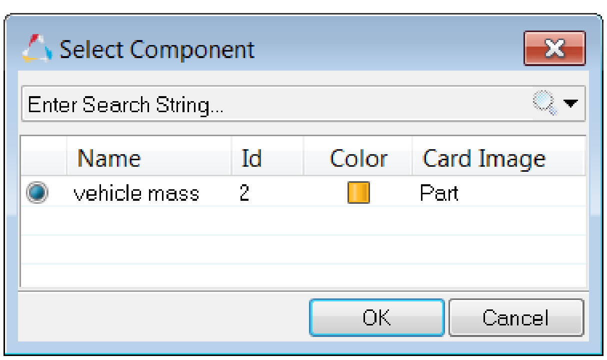
1. In the **Solver** browser, right-click and select ***Create*** > ****CONSTRAINED*** > ****CONSTRAINED_EXTRA_NODES_SET*** from the context menu. A new constrained extra node opens in the **Entity Editor**.



2. In the **Entity Editor**, define the constrained extra node.
   a. For **Name**, enter `ExtraNodes`.
   b. For **PID**, click ***Unspecified*** >> ***Component***.



   c. In the **Select Component** dialog, select ***vehicle mass*** and then click ***OK***.



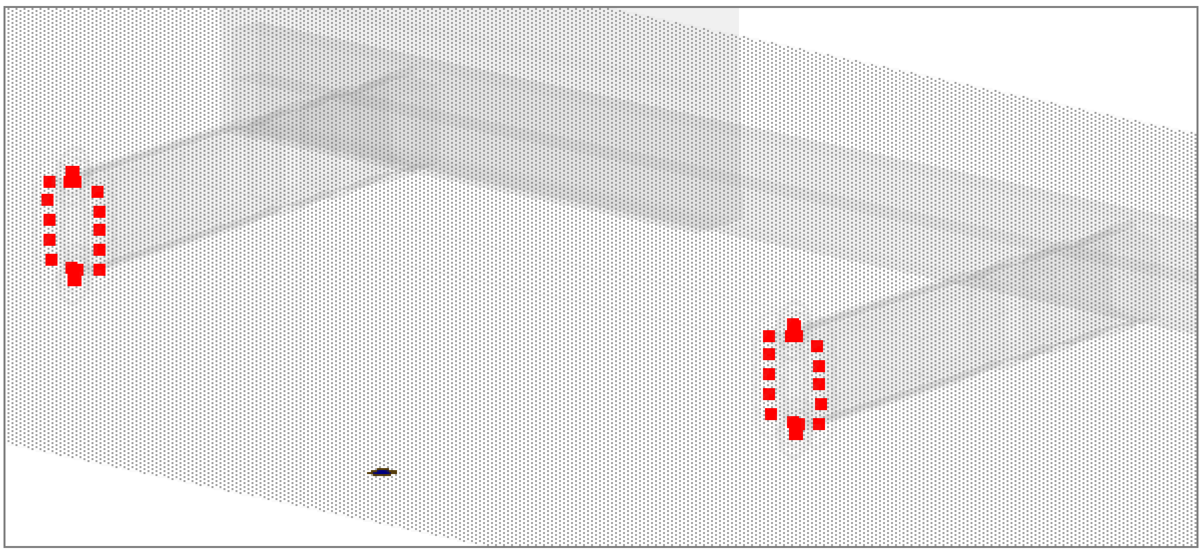
Step 8: Define the nodes in the Constrain Vehicle set to be a part of the vehicle mass rigid body

In this step, the **Entity Editor** should still be open for the ExtraNodes constrained extra node.

1. For **NSID**, click ***Unspecified*** >> ***Set***.
2. In the **Select Set** dialog, select ***Constrain Vehicle*** and then click ***OK***.

Step 9: View the extra nodes that are a part of the vehicle mass rigid body

1. In the **Solver** browser or **Model** browser, right-click on ***ExtraNodes*** and select ***Review*** (press ***Q***) from the context menu. The extra nodes temporarily display red, and PID (vehicle mass) displays blue. All of the other entities temporarily display grey.



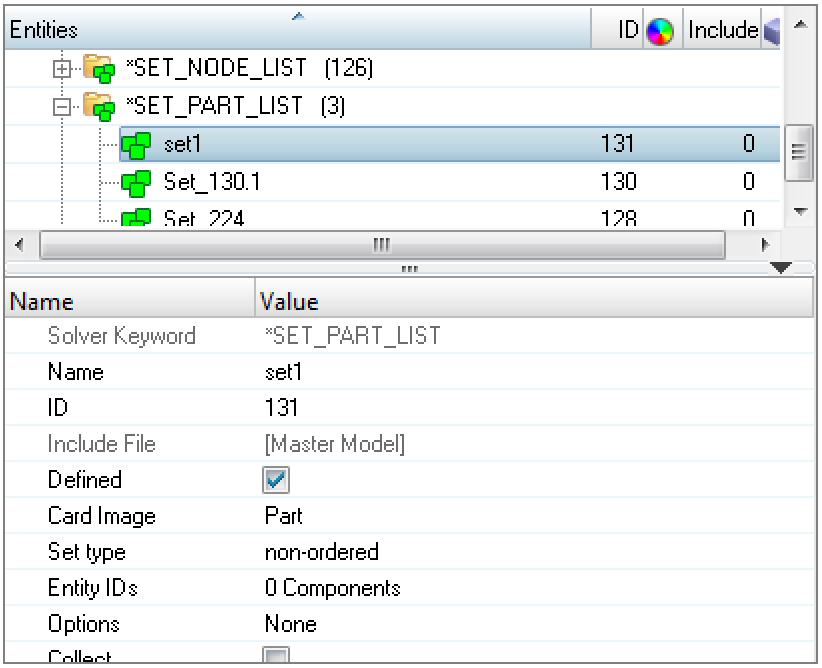
2. Return all of the entities to their original display color by right-clicking on ***ExtraNodes*** and selecting ***Review*** (press ***Q***) from the context menu.

Step 10: Create an entity set, *SET_PART_LIST, for the vehicle mass component

All other components not in this set will be included in the contact.

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

   Tip: You can also create a *SET_PART_LIST from the Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.



2. In the **Entity Editor**, define the set.

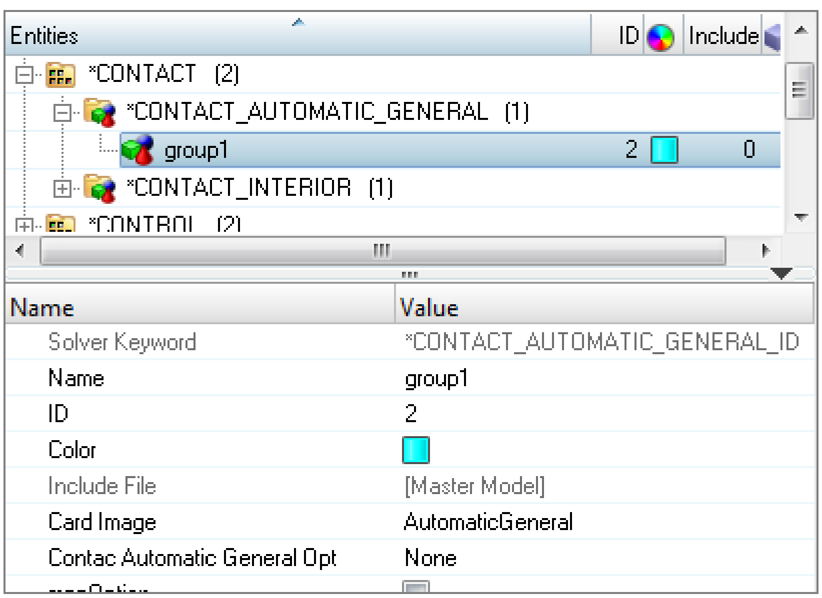
   a. For **name**, enter `Exempt Parts`.

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

   c. In the **Select Components** dialog, select ***vehicle mass*** and then click ***OK***.

Step 11: Create *CONTACT_AUTOMATIC_GENERAL contact

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

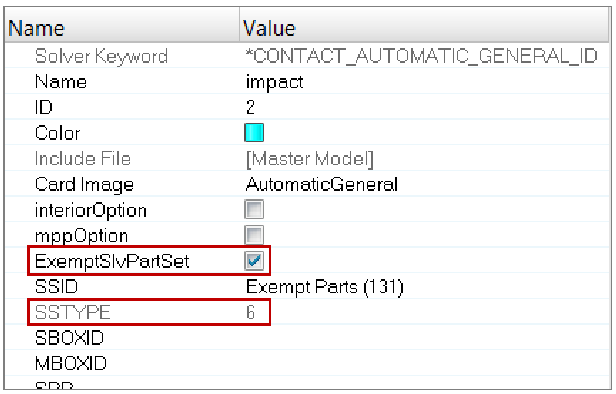


2. For **name**, enter `impact`.

Step 12: Define the slave surface with slave set type 6, part set ID for exempted parts

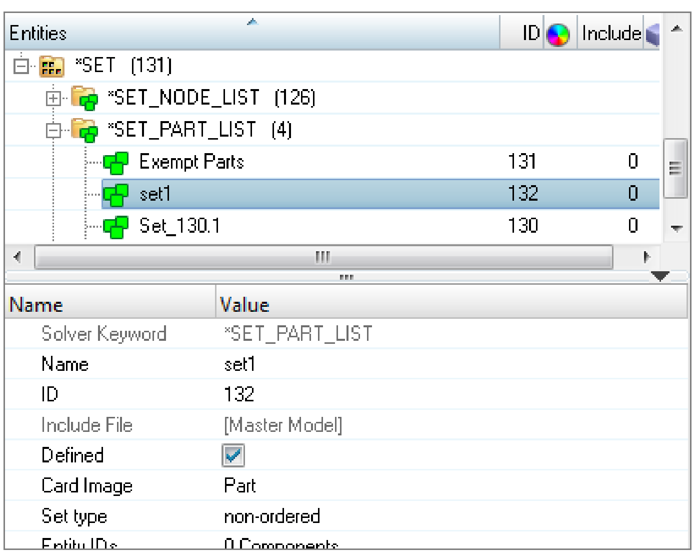
In this step the **Entity Editor** should still be open for the **impact** group.

1. Click ***SSID***.
2. Set the entity selector to ***Set***.
3. Click ***Set***.
4. In the **Select Set** dialog, select ***Exempt Parts*** and then click ***OK***.
5. Select the ***ExemptSlvPartSet*** checkbox. The **SSTYPE** (slave surface type) value changes from 2 (part set ID) to 6 (part set ID for exempted parts).



Step 13: Create an entity set, *SET_PART_LIST, to specify the elements that will contribute to the cross-sectional force results

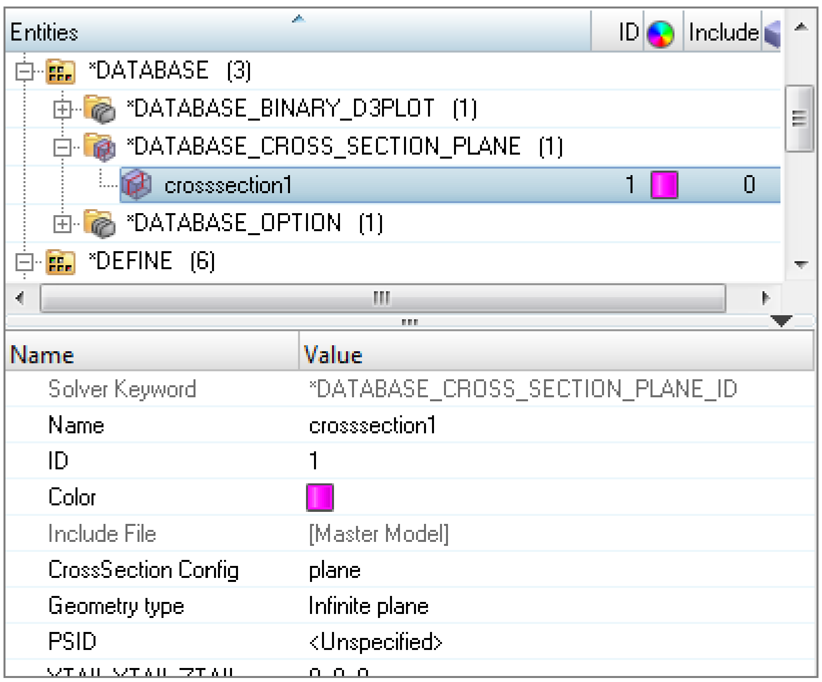
1. In the **Solver** browser, right-click and select ***Create*** > ****SET*** > ****SET_PART*** > ****SET_PART_LIST*** from the context menu. A new set opens in the **Entity Editor**.



2. In the **Entity Editor**, define the set.
    a. For **Name**, enter `CrossSectionPlane-Parts`.
    b. For **Entity IDs**, click ***0 Components*** >> ***Components***.
    c. In the **Select Components** dialog, select **interior crashbox** and **exterior crashbox**.
    d. Click ***OK***.

Step 14: Define a section by creating *DATABASE_CROSS_SECTION_PLANE

1. In the **Solver** browser, right-click and select ***Create*** > ****DATABASE*** > ****DATABASE_CROSS_SECTION_PLANE*** from the context menu. A new cross section opens in the **Entity Editor**.



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

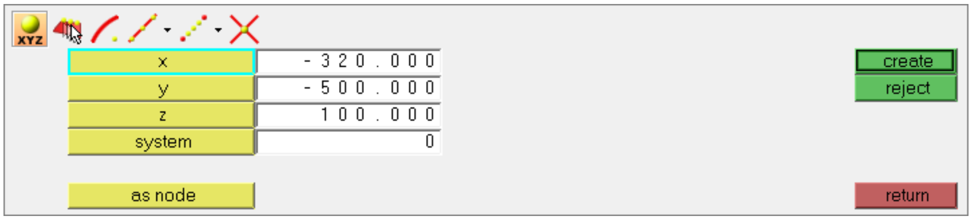
Step 15: Define the location and size of the section's plane

In this step the plane's origin (the tail of the normal vector) is defined by a base node.

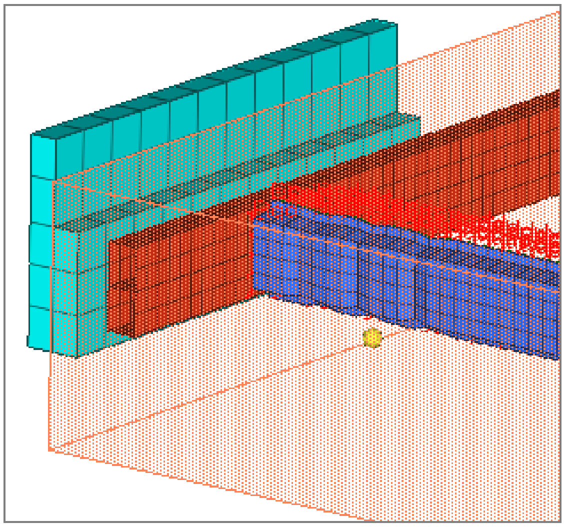
The **Entity Editor** should still be open for the CrossSection_Plane cross section.

1. Create a base node.
    a. Open the **Create Nodes** panel by clicking ***Geometry*** > ***Create*** > ***Nodes*** > ***XYZ*** from the menu bar, or by pressing ***F8***.
    b. In the **x** field, enter `-320`.
    c. In the **y** field, enter `-500`.

d. In the **z** field, enter `100`.



e. Click ***create***. A new node displays.



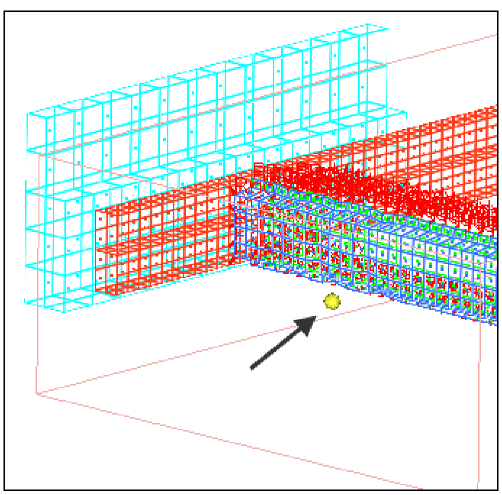
f. Click ***return***.

2. In the **Entity Editor**, define the **XTAIL, YTAIL, ZTAIL** (base node) for the section.

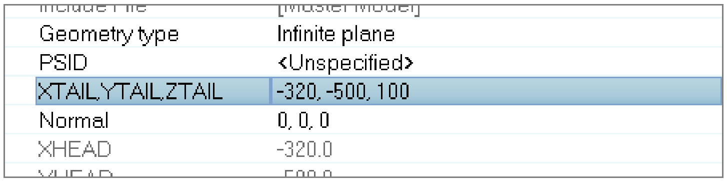
a. Click **XTAIL, YTAIL, ZTAIL** (base node), and then click .

b. In the graphics area, select the base node you just created.

Tip: If the base node is not visible, click on the **Visualization** toolbar to display elements as a wireframe (skin only).

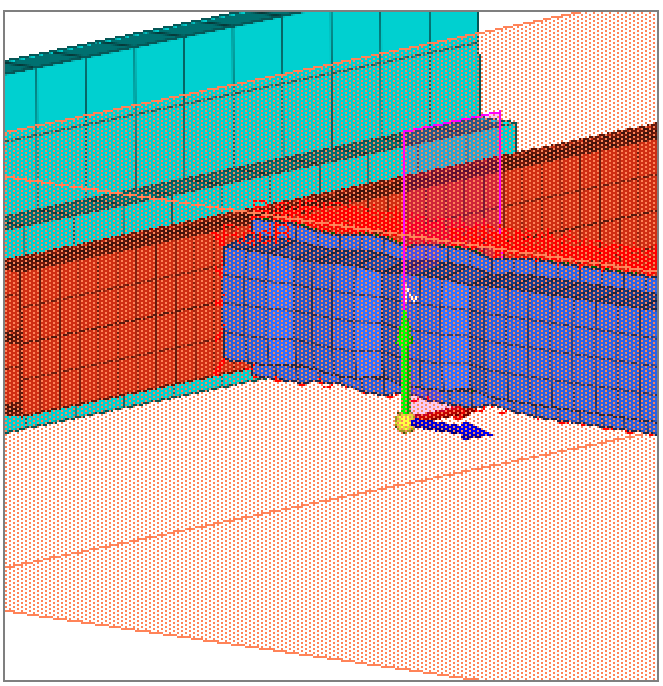


c. Click ***proceed***. The **Entity Editor** displays the coordinates of the base node in the **XTAIL, YTAIL, ZTAIL** field.



Geometry type	Infinite plane
PSID	<Unspecified>
XTAIL,YTAIL,ZTAIL	-320, -500, 100
Normal	0, 0, 0
XHEAD	-320.0

3. Set **Geometry type** to ***Finite plane***.
4. Define the normal vector.
   a. Click **Normal**, and then click .
   b. In the panel area, set the orientation selector to ***x-axis***.
   c. Click ***proceed***.
5. Define the edge vector
   a. Click ***Edge***, and then click .
   b. In the panel area, set the orientation selector to ***y-axis***.
   c. Click ***proceed***. The **Entity Editor** displays the coordinates of the edge vector L in the **Normal** field.
6. For **LENL** (length of edge a, in the L direction), enter `100`.
7. For **LENM** (length of edge b, in the M direction), enter `200`.



Tip: If you know the coordinates of the base node, edge, and normal, you can manually enter them in the Entity Editor.

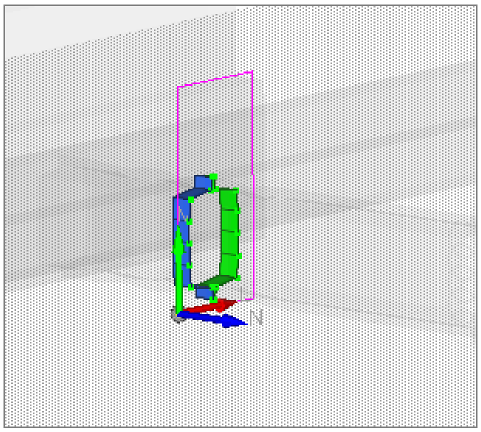
Step 16: Specify the parts slave to the cross section

In this step the **Entity Editor** should still be open for the CrossSection_Plane cross section.

1. For **PSID**, click ***Unspecified*** >> ***Set***.
2. In the **Select Set** dialog, select ***CrossSectionPlane-Parts*** and then click ***OK***.

Step 17: View the entities slave to the rigid wall

1. In the **Solver** browser, right-click on ***CrossSection_Plane*** and select ***Review*** (press ***Q***) from the context menu. The slave entities and rigid wall highlight. All of the other entities temporarily display grey.

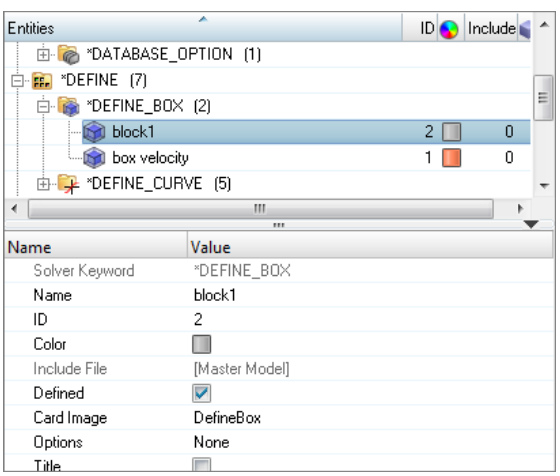


2. Return all of the entities to their original display color by right-clicking on ***CrossSection_Plane*** and selecting ***Review*** (press ***Q***) from the context menu.

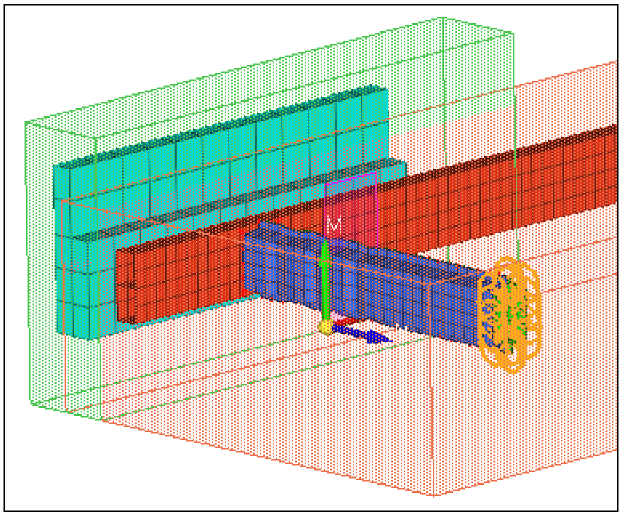
Step 18: Create a *DEFINE_BOX containing the nodes making up the barrier and bumper's left side.

These nodes will be slave to the rigid wall.

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



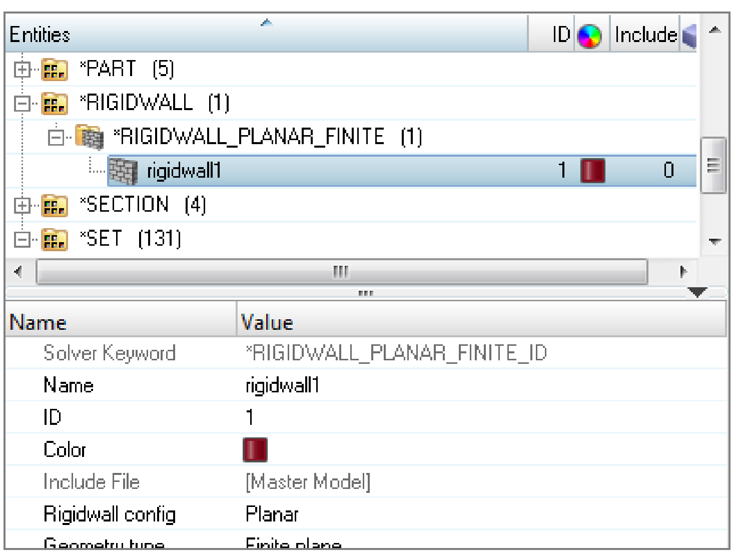
2. In the **Entity Editor**, define the block.
   a. For **Name**, enter `half model`.
   b. Optional. Click the ***Color*** icon and select a color to display the block.
   c. For **Xmin Ymin Zmin**, enter `-600`, `-800`, `0`.
   d. For **Xmax Ymax Zmax**, enter `-460`, `0`, `400`.



Step 19: Define a HyperMesh group by creating *RIGIDWALL_PLANAR_FINITE

*RIGIDWALL are created from the Solver browser or Model browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

1. In the **Solver** browser, right-click and select ***Create*** > ****RIGIDWALL*** > ****RIGIDWALL_PLANAR_FINITE*** from the context menu. A new rigid wall opens in the **Entity Editor**.



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

Step 20: Define the location and size of the rigid wall

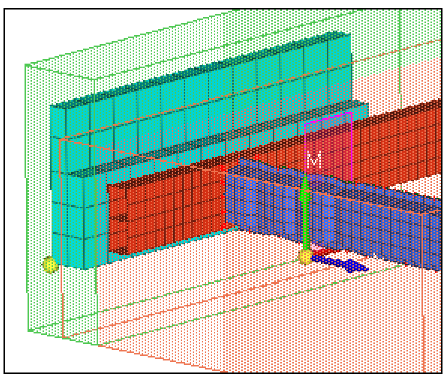
In the **Create Nodes** panel, **XYZ** sub-panel, the rigid wall's origin (the tail of the normal vector) is defined by a base node. In this step, you will create a node from the create nodes panel and then select it for the base node.

In this step the **Entity Editor** should still be open for the rigid wall.

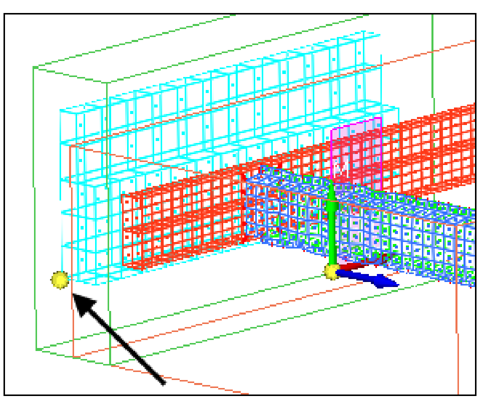
1. Create a base node.
   a. Open the **Create Nodes** panel by pressing ***F8***.
   b. Go to the **XYZ** subpanel, click 

.
   c. In the **x** field, enter `-600`.
   d. In the **y** field, enter `-750`.
   e. In the **z** field, enter `90`.
   f. Click ***create***.

   Tip: If the base node is not visible, click  on the **Visualization** toolbar to display elements as a wireframe (skin only).

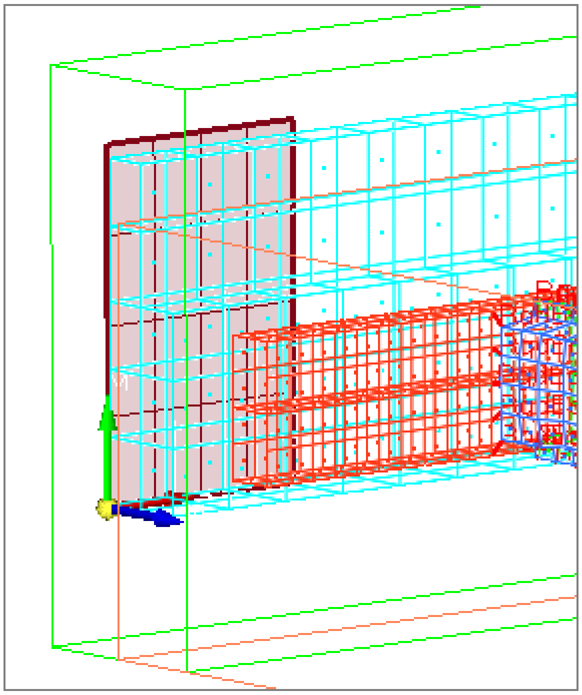


   g. Click ***return***.
2. In the **Entity Editor**, enter values for XT, YT, ZT, or select the above node for the rigid wall base from graphics area.



3. Set **Geometry type** to ***Finite plane***.
4. Define the normal vector.
   a. Click **Normal**, and then click .
   b. In the panel area, set the orientation selector to ***x-axis***.
   c. Click ***proceed***.
5. Define the edge vector.
   a. Click ***Edge***, and then click .
   b. In the panel area, set the orientation selector to ***y-axis***.
   c. Click ***proceed***.
6. For **Length LENL**, enter `165`.
7. For **Length LENM**, enter `250`.

   Note: The input values for LENL and LENM are the length of the edges a and b in the L and M directions, respectively. These values define the extent of the rigid wall.



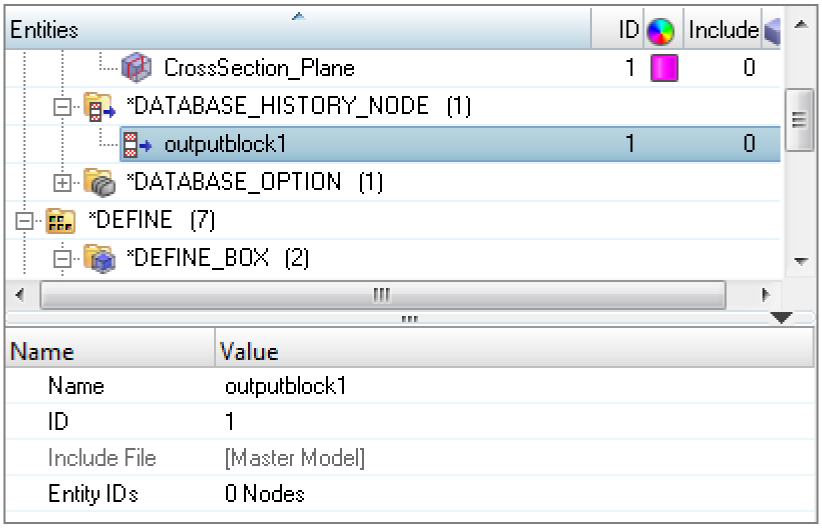
Step 21: Use the Entity Editor for the rigid wall to specify the nodes in the *DEFINE_BOX half model as slave to the rigid wall

In this step the **Entity Editor** should still be open for the rigid wall.

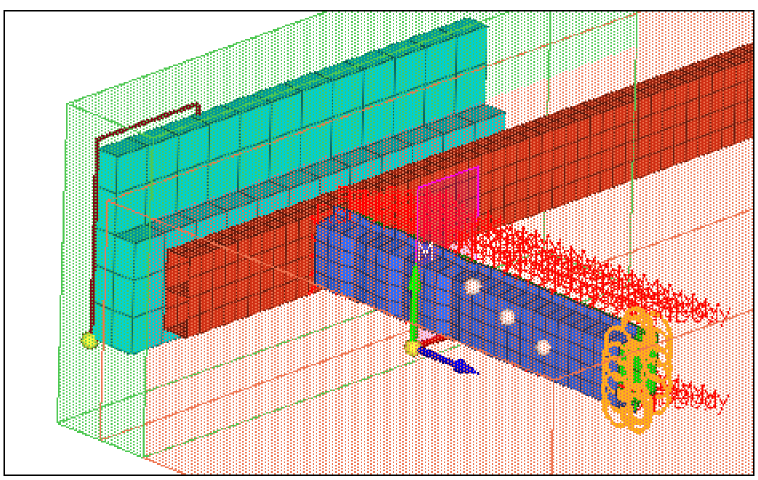
1. Click **BOXID** >> ***Block***.
2. In the **Select Block** dialog, select ***half*** and then click ***OK***.
3. For **FRIC** (Interface friction), enter `1.0`.

Step 22: Specify some nodes to be output to the ASCII NODOUT file with *DATABASE_HISTORY_NODE

1. In the **Solver** browser, right-click and select ***Create*** > ****DATABASE*** > ****DATABASE_HISTORY_NODE*** from the context menu. A new output block opens in the **Entity Editor**.



2. In the **Entity Editor**, define the output block.
   a. For **Name**, enter `nodeth`.
   b. For **Entity IDs**, click ***0 Nodes*** >> ***Nodes***.
   c. In the graphics area, select a few nodes of interest.



   d. Click ***proceed***.

Step 23: Export the model to an LS-DYNA 971_R# formatted input file

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***. The **Export - Solver Deck** tab opens
2. Set **File type** to ***LsDyna***.
3. In the **File** field, navigate to your working directory and save the file as `Bumper_complete.key`.
4. Click ***Export***.

Step 24 (Optional): Submit the LS-DYNA input file to LS-DYNA 970 solver

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 `bumper_complete.key`.
4. Start the analysis by clicking ***OK***.

Step 25 (Optional): View the results in HyperView

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