



Altair

---

**HyperWorks**

## HM-4800: PERMAS Analysis

In this tutorial, you will go through the full set up of a PERMAS analysis. You will:

- Organize existing elements into HyperMesh component collectors
- Create materials and elemental properties and assign both to elements and sets
- Create beam and spring elements
- Apply loads and boundary conditions
- Organize the components, loads and boundary conditions into \$LOADING and \$CONSTRAINTS variants
- Specify contacts and contact properties
- Define a load history with the \$NNLOAD card

## Model Files

---

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

## Exercise

---

### Step 1: Load the PERMAS user profile and the model

Complete the steps below to load the PERMAS user profile and the model:

1. Open HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Permas**.
3. From the menu bar, click **File > Open > Model**.
4. In the **Open Model** dialog, open the `permastube_clamping.hm` file.

**Note:** This file contains pre-defined model data. Use this file in the following exercises.

### Step 2: Create a PERMAS \$COMPONENT and \$SYSTEM

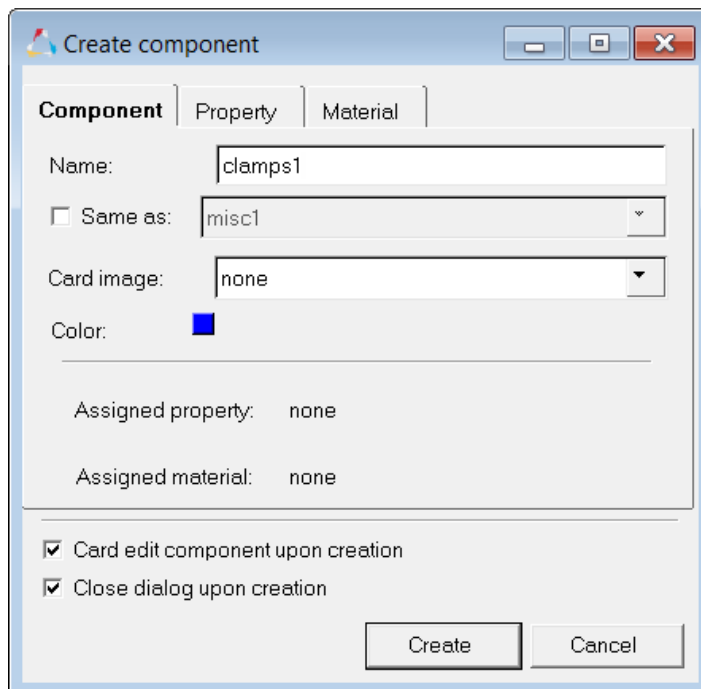
As a first recommended step for each model set up in the PERMAS user profile, you should create a component and a system card. If this is not done, a warning message is displayed in the message bar on export. Currently only one component and system per model is supported in HyperMesh.

1. From the menu bar, click **Setup > Create > Control Cards**.
2. In the **Card Image** dialog, click **COMPONENT**.
3. In the **Name** field, enter `comp1`.

4. Click **return** to create the card and close the card image.
5. Click **SYSTEM**.
6. In the **Name** field, enter `sys1`.
7. Click **return** to create the card and exit the card image.
8. Click **return** again to close the **Card Image** dialog.

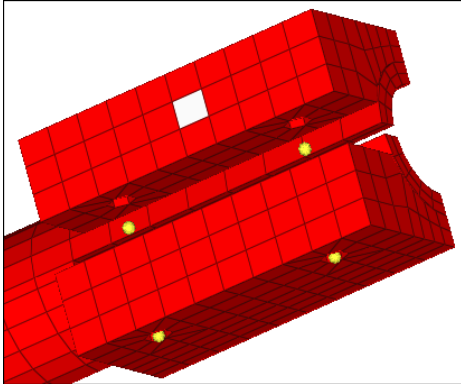
### Step 3: Create a component collector

1. In the **Model** browser, right-click and select **Create** > **Component** from the context menu.
2. In the **Create Component** dialog:
  - For **Name**, enter `clamps1`.
  - Select a color for the component.
  - Leave **Card image** set to **none**.
  - Click **Create**.

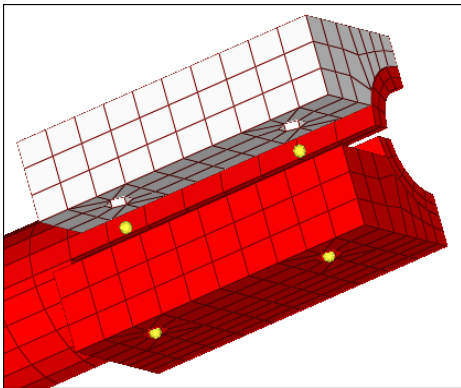


### Step 4: Organize the elements

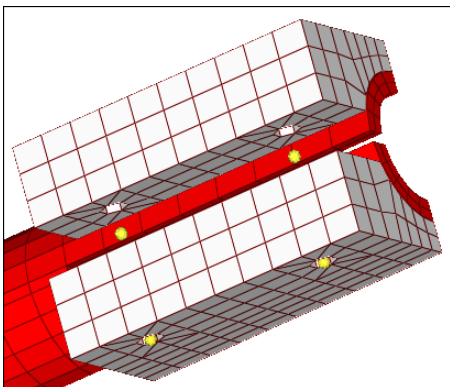
1. From the menu bar, click **Mesh > Organize > Elements > To Component**.
2. Select one element on one side of the clamp.



3. In the **Organize** panel, click **elems >> by attached**. The entire side of the clamp becomes selected.

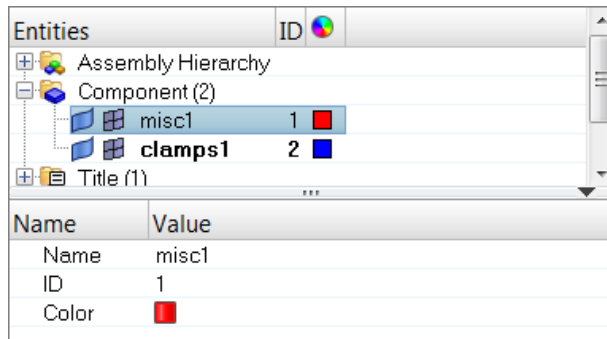


4. Select the elements on the other side of the clamp.

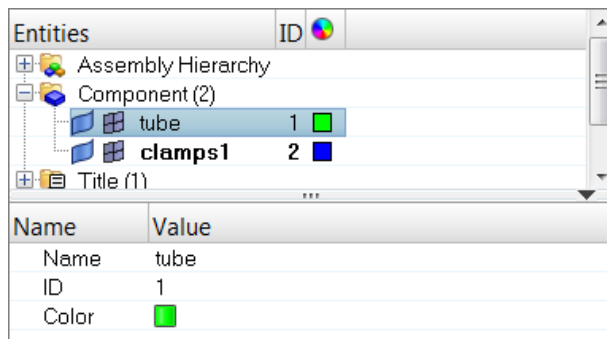


5. In the **dest component =** field, ensure that **clamps1** is entered.
6. Click **move**. HyperMesh organizes the selected elements into the clamps1 component.
7. Click **return** to exit the panel.

- In the **Model** browser, **Component** folder, click **misc1**. The **Entity Editor** opens and displays the component's card data.



- In the **Entity Editor**:
  - For **Name**, enter **tube**.
  - Click the color box, and select a new display color for the component.



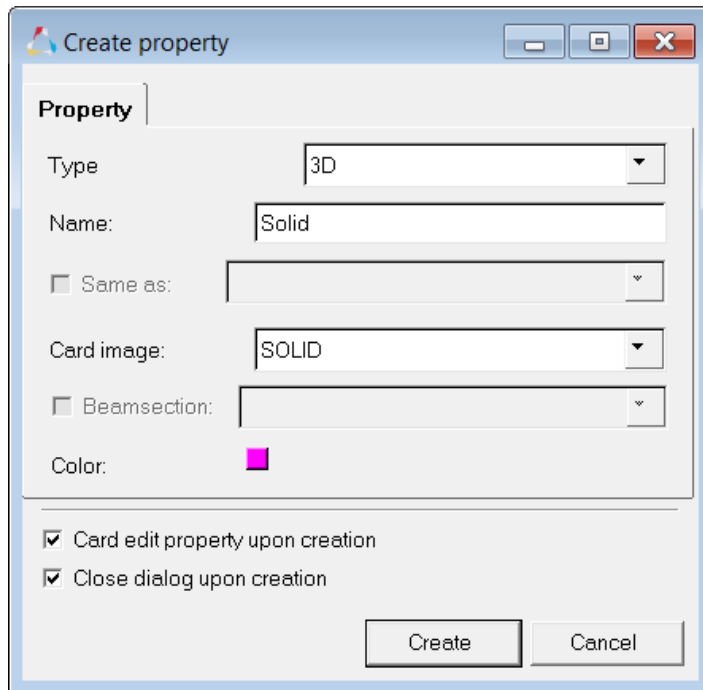
## Step 5: Creating material and elemental properties

In this step you will create two materials and a solid property for the already existing elements. You will also assign the property and the materials to the previously created components. Assigning a property/material to a component is equivalent to a property to set assignment in PERMAS. For 2D and 3D elements, it is currently the only way of property assignment.

- In the **Model** browser, right-click and select **Create > Property** from the context menu.
- In the **Create Property** dialog:
  - For **Name**, enter **Solid**.
  - Set **Type** to **3D**.
 

**Note:** The type selection is helpful when you review your model in the [Model browser](#) (property or material view). Even if you do not select a type at this point, it will still display as the correct type. Selecting the type only prefilters the card images for you in this dialog.
  - Set **Card image** to **SOLID**.
  - Select the **Card edit property upon creation** checkbox.

- Click **Create**. The card image for the property you just created opens.

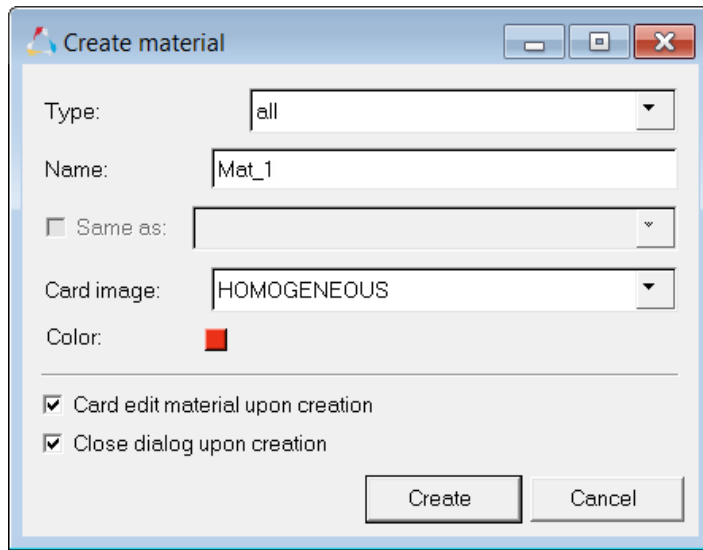


3. When finished, click **return** to close the card image.

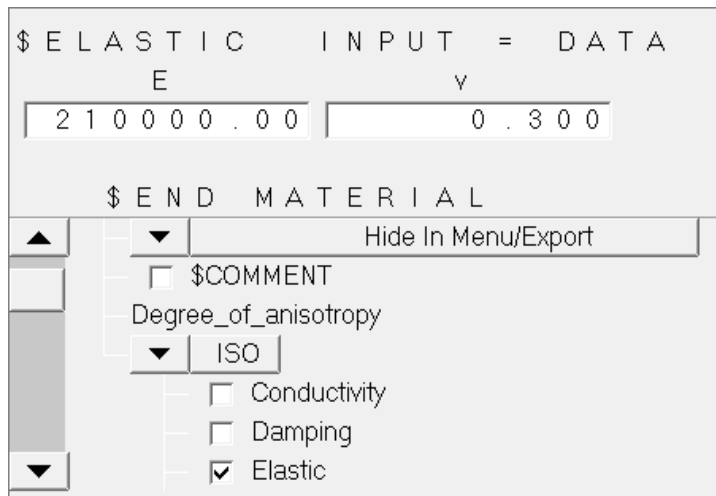


4. In the **Model** browser, right-click and select **Create > Material** from the context menu.
5. In the **Create material** dialog:
  - For **Name**, enter `Mat_1`.
  - Set **Card image** to **HOMOGENEOUS**.
  - Select the **Card edit material upon creation** checkbox.

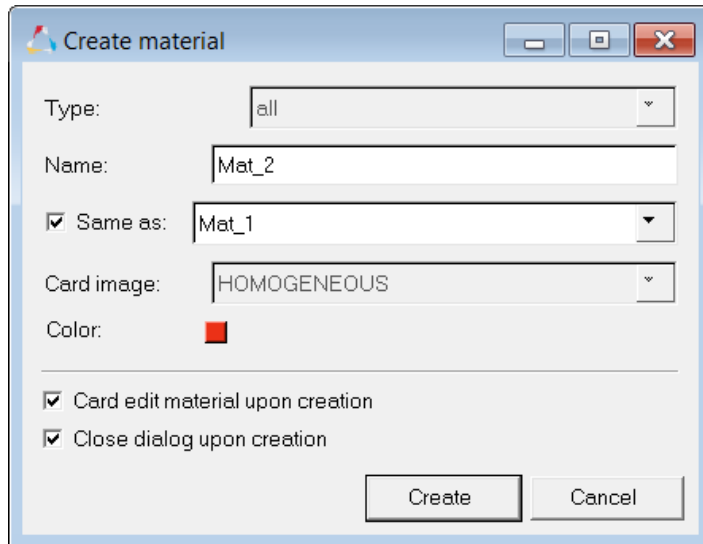
- Click **Create**. The card image for the material you just created opens.



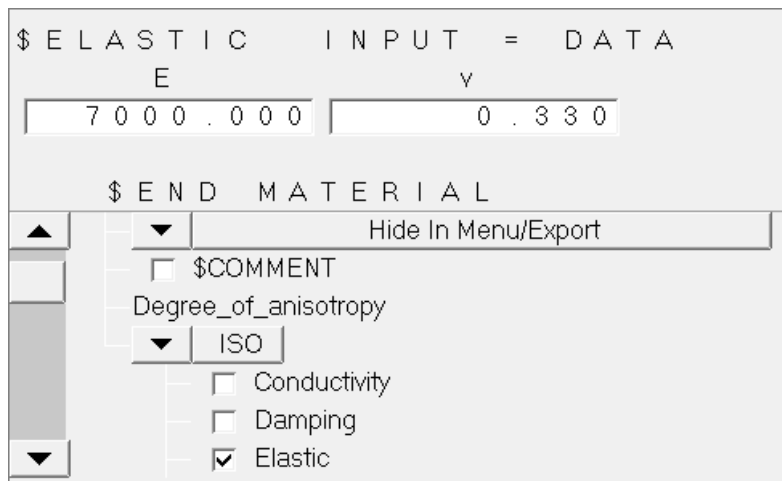
6. In the card image:
  - Select the **Elastic** checkbox.
  - In the **E** (elasticity modulus field) field, enter 210000.
  - In the **v** (poisson's ratio) field, enter 0.3.
  - Click **return** to close the card image.




7. In the **Model** browser, right-click and select **Create > Material** from the context menu.
8. In the **Create material** dialog:
  - For **Name**, enter Mat\_2.
  - Select the **Same as** checkbox to apply the same characteristics from the Mat\_1 material to the Mat\_2 material.
  - Click **Create**. The card image for the material you just created opens.



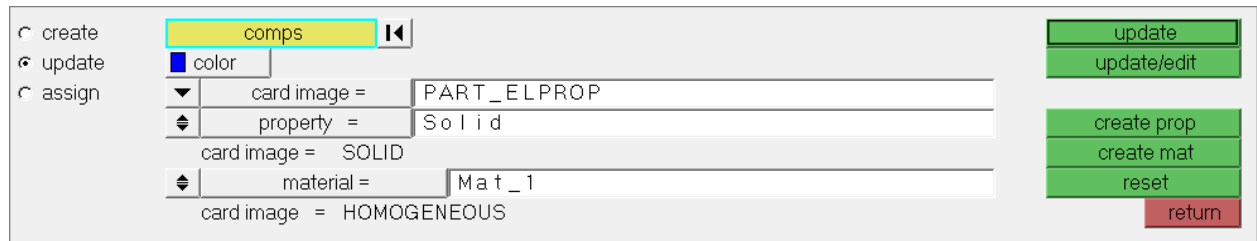
9. Because **Mat\_2** has the same characteristics as **Mat\_1**, the **Elastic** checkbox is already selected and the values should already be entered in the **E** and **v** fields. Change **E** to 7000 and **v** to 0.330.



10. Click **return** to close the card image.
11. On the **Collectors** toolbar, click . The **Components** panel opens.
12. Go to the **update** subpanel.
13. Click **comps**.
14. Select the component, **clamps1**.
15. Click **select**.
16. Click **Card image**, and select **PART\_ELPROP**.
17. Set the **property/no property** toggle to **property**.
18. Click **property=**, and select **solid**.
19. Set the **material/no material** toggle to **material**.



20. Click **material=**, and select **Mat\_1**.



21. Click **update**.
22. In the **Model** browser, **Component** folder, right-click on **clamps1** and select **Card Edit** from the context menu. The card image opens, and the name of the component, property, and material you assigned is shown.

\$ELPROP		
Name	GEODAT	MATERIAL
clamps1	Solid	Mat_1

23. Click **return** to close the card image.
24. Edit the **tube** component so that the card image is set to **Part\_ELPROP**, and the **Solid** property and **Mat\_2** material are assigned.
25. Click **Update** to complete the changes.

## Step 6: Create the Beam elements

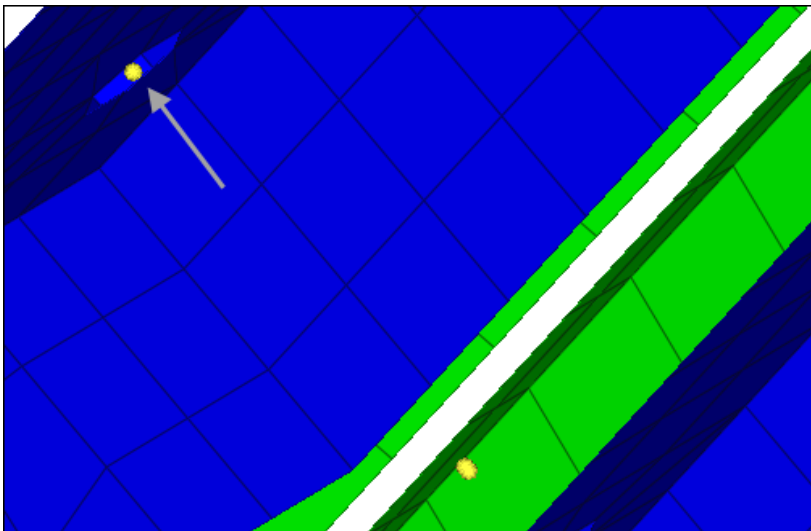
In this section of the tutorial you will learn how to create beam elements and assign a property to individual elements during creation. In comparison to 2D and 3D elements, you can assign mutually exclusive properties to elements or sets (components).

- In the **Model** browser, right-click and select **Create > Component** from the context menu.
- In the **Create component** dialog:
  - For **Name**, enter *Beam*.
  - Set **Card image** to *none*.
  - Clear the **Card edit component upon creation** checkbox.
  - Click **Create**.
- In the **Model** browser, right-click and select **Create > Property** from the context menu.
- In the **Create property** dialog:
  - For **Name**, enter *XBA*.
  - Set **Type** to **1D**.
  - Set **Card image** to **BEAM**.

- Select the **Card edit property upon creation** checkbox to edit the property's card data upon creation.
  - Click **Create**.
5. In the card image, enter the following parameters:
- **A-1** = 12.6
  - **Ixx-1** = 12.6
  - **Iyy-1** = 12.6
  - **J-1** = 25.1

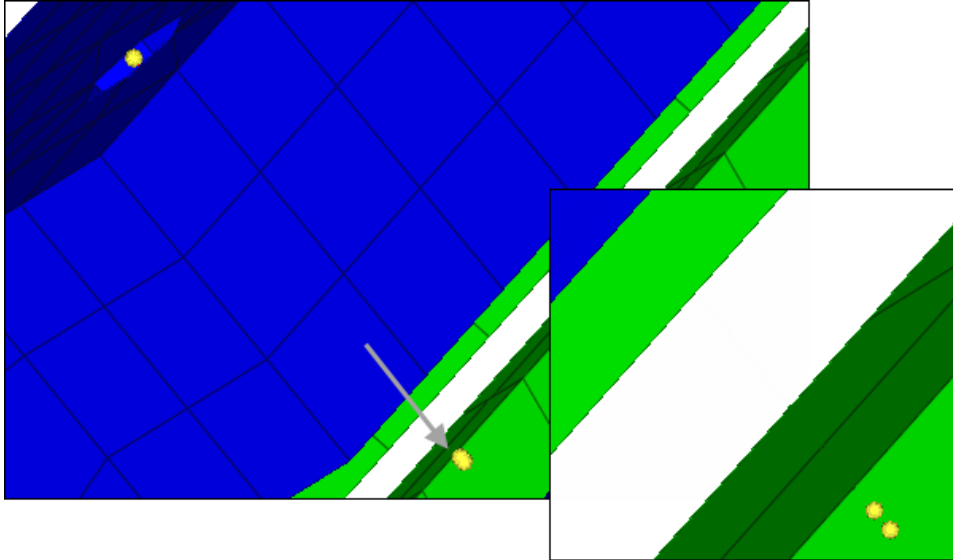
\$GEODAT BEAM CONT=AREA BEND TORS				
Name				
XBA				
A-1	Ixx-1	Iyy-1	Ixy-1	J-1
12.600	12.600	12.600	0.000	25.100

6. Click **return** to exit the panel.
7. Open the **Bars** panel by clicking **Mesh > Create > 1D Elements > Bars** from the menu bar.
8. Set **orientation** selector to **z-axis**.
- Note:** Because HyperMesh is a generic preprocessor, you have to provide an orientation vector to create the element, although this vector does not go anywhere in the PERMAS deck. For beams, use the node option in the same panel where a direction vector needs to be defined.
9. Click **property =**, and select **XBA**.
10. Using the **node A** selector, select the node on the top of one side of the clamp.

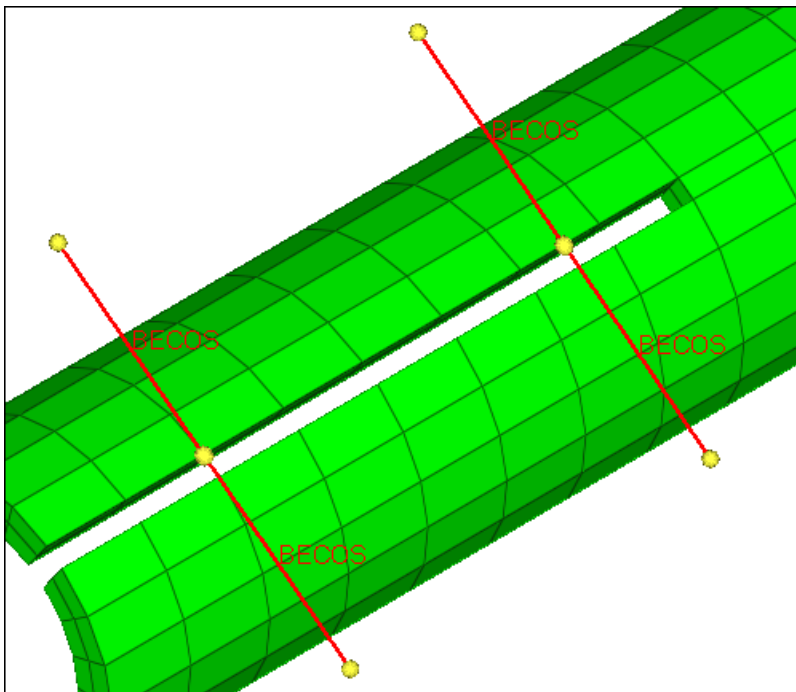



- Using the **node B** selector, select the node in the middle of the clamp that is closest to the node you selected as node A. The beam is created automatically.

Tip: You may need to zoom in on the model to properly select the the node in the middle of the clamp which is closest to the node you selected as node A.

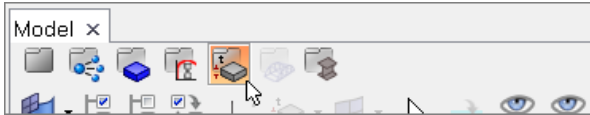



- Repeat steps 6.17 and 6.18 to connect the lower node in the middle of the clamp with the node on the opposite, outer side.
- Repeat steps 6.17 – 6.19 on the other side of the clamp.
- In the **Model** browser, **Component** folder, right-click on **clamps1** and select **Hide** from the context menu to see the four sets of bars connecting the six nodes.



15. In the **Model** browser, click  (property view) to display the elements in the color of the property that has been assigned to them.

**Note:** This is a good way to ensure that you have assigned the correct property to the elements.



16. In the **Model** browser, click  (model view) to return to the default display settings. If necessary, right-click on the **clamps1** component and select **Show** to display the clamps again.
17. Click **return** to exit the panel.

## Step 7: Create the Spring elements

In this step you will create a spring element similar to the way you created the beam elements, and you will change the element type used when elements are created. You will use the coincident picking mechanism to help you precisely select nodes, which is useful in cases where entities (for example nodes and elements) are close to each other.

1. In the **Model** browser, right-click and select **Create > Component** from the context menu.
2. In the **Create component** dialog:
  - For **Name**, enter `Springs`.
  - Set **Card image** to **PART\_ELPROP**.
  - Clear the **Card edit component upon creation** checkbox.
  - Click **Create**.

**Note:** Springs should now be your current component collector, which ensures that newly created elements are put into this collector.
3. Open the **Element Type** panel by clicking **Mesh > Assign > Element Type** from the menu bar.
4. Go to the **1D** subpanel.
5. Click **spring =**, and select **X2STIFF3**.
6. Click **return** to exit the panel.
7. From the menu bar, click **Preferences > Graphics**.
8. In the **Graphics** panel, select the **coincident picking** checkbox.
 

**Note:** Coincident picking enables you to select one or more nodes which reside very close to each other. During node selection, you will have more control. See the online help for the [graphics subpanel](#) for more information.
9. Click **return**.
10. In the **Model** browser, right-click and select **Create > Property** from the context menu.

11. In the **Create property** dialog:
  - For **Name**, enter **XST**.
  - Set **Type** to **OTHERS**.
  - Set **Card image** to **SCALAR**.
  - Select the **Card edit property upon creation** checkbox.
  - Click **Create** to create the property and edit the card.
12. In the card image:
  - Select the **DOF** checkbox to enable the degrees of freedom at the first and second node.
  - In the **coeff** field, enter 100.
  - In the **DOF1** and **DOF2** fields, enter 2.
  - Click **return**.

\$GEODAT SCALAR CONT = COEFF DOF			
Name	coeff	DOF1	DOF2
XST	100.000	2	2

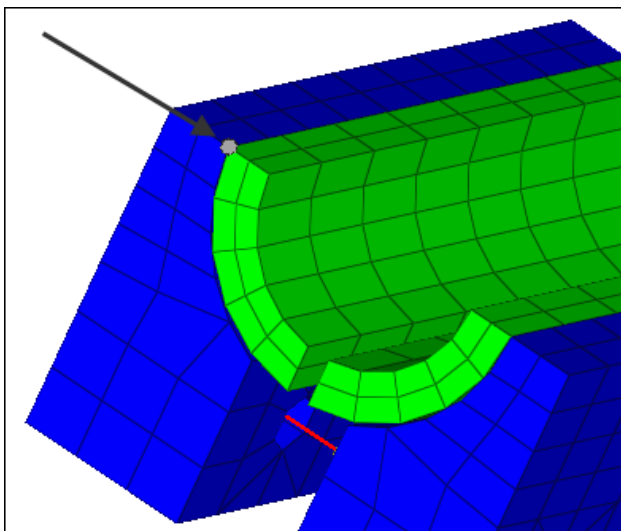
User Comments

▼ Hide In Menu/Export

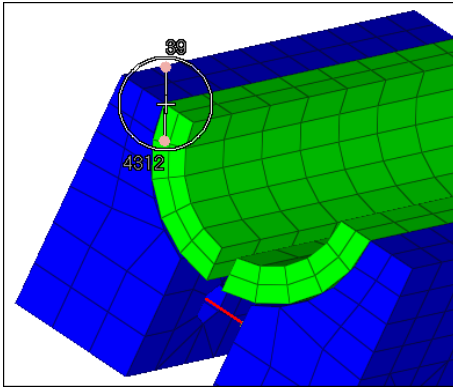
DOF

\$COMMENT

13. Open the **Springs** panel by clicking **Mesh > Create > 1D Elements > Springs** from the menu bar.
14. Click **property =**, and select **XST**.
15. Using the **node** selector, select the node shown in the image below. The coincident picking tool appears, and displays two nodes.

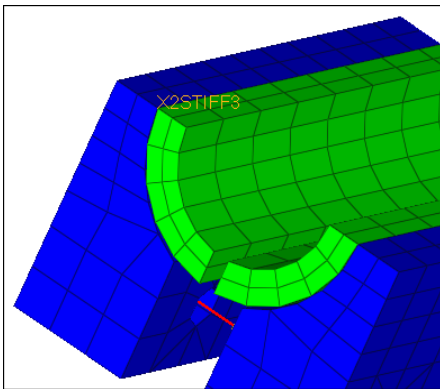


16. Click the top node.

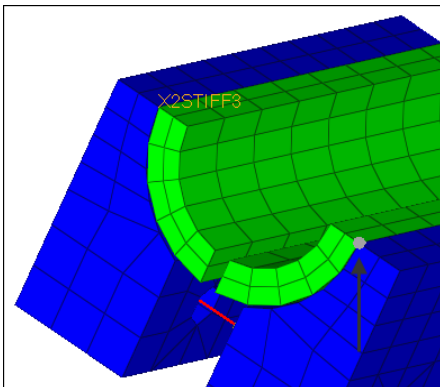


17. Using the **node** selector, click the same node.

18. Using the coincident picking tool, click the bottom node. HyperMesh creates a spring.



19. Repeat steps 7. 25 and 7.28 to select the same nodes on the other side of the model.



20. When finished, click **return** to exit the panel.

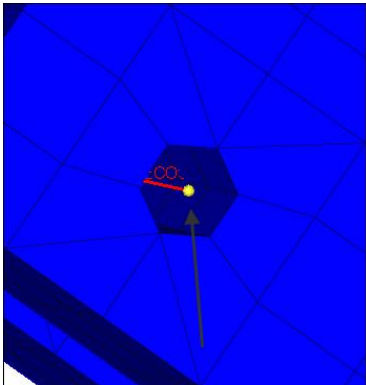
## Step 8: Create Multi-Point Constraints

In this step, you will create two different types of multi point constraints. In order to switch their display on and off separately, you will create a component collector for each type.

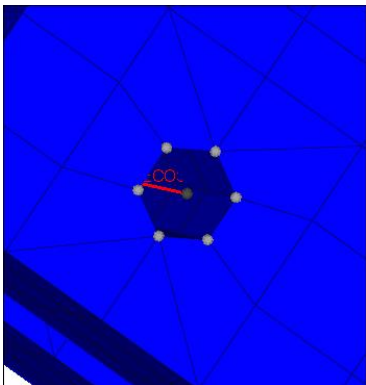
Because MPCs do not need a property assignment in HyperMesh, no card image (\$ELPROP) is needed for these components.

1. In the **Model** browser, right-click and select **Create > Component** from the context menu.
2. In the **Create component** dialog:
  - For **Name**, enter MPC\_Rigids.
  - Set **Card image** to **none**.
  - Clear the **Card edit component upon creation** checkbox.
  - Click **Create**.

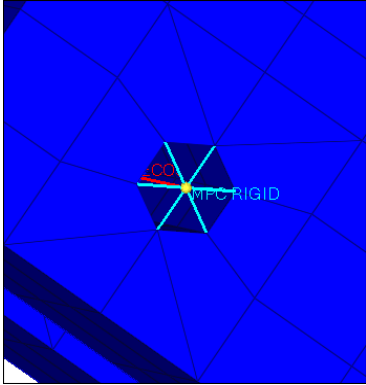
**Note:** This collector will be used to organize MPC elements created later in this step.
3. Open the **Rigids** panel by clicking **Mesh > Create > 1D Elements > Rigids** from the menu bar.
4. Set the **dependent** switch to **multiple nodes**.
5. Clear the **dof4**, **dof5** and **dof6** checkboxes.
6. Using the **independent node** selector, select one of the nodes at the end of the beam elements created in Step 6.



7. Using the **dependent node** selector, select the nodes surrounding the first node.



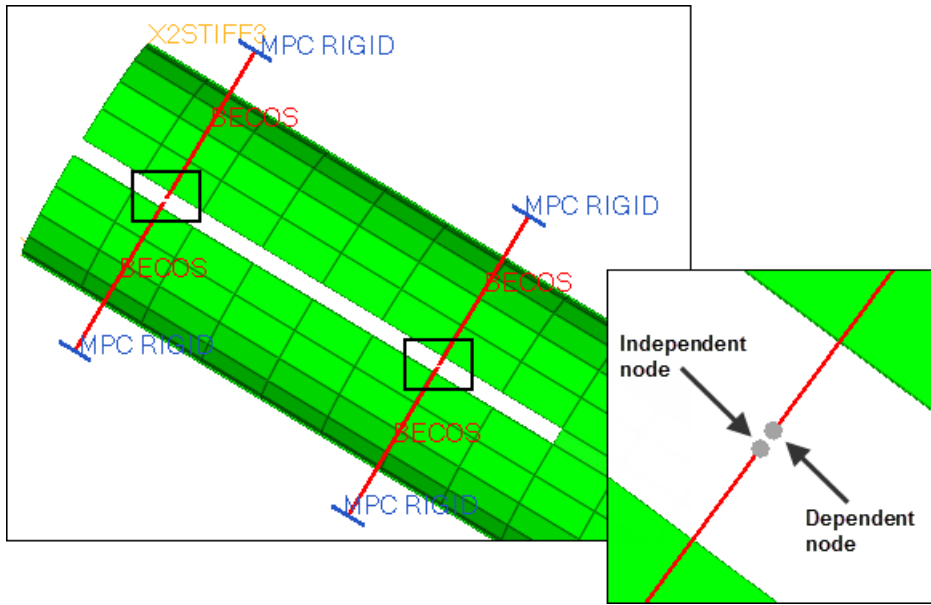
8. Click **create**.



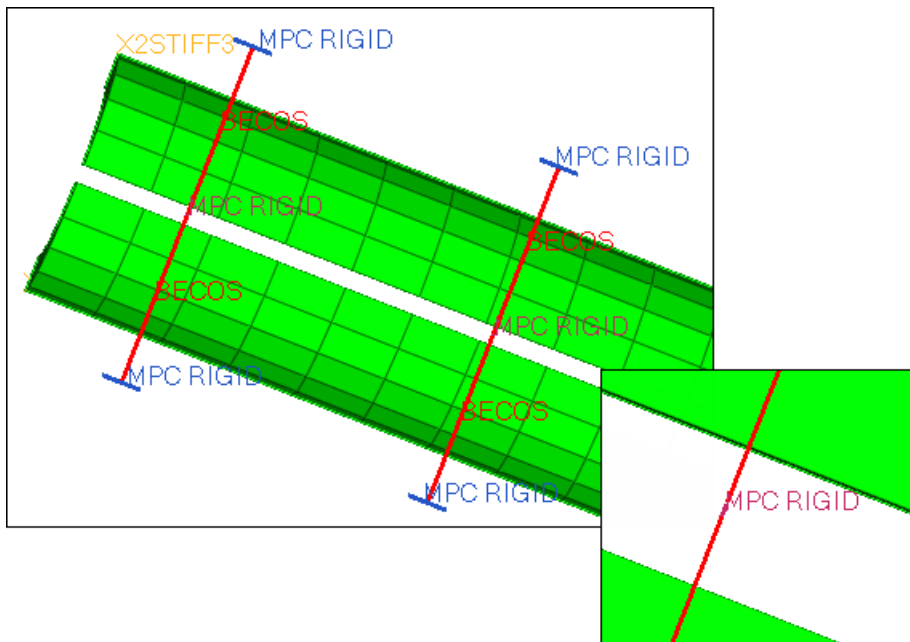
9. Repeat steps 8.9 - 8.11 to create elements for the other three ends of the beam elements.
10. When finished, click **return** to exit the panel.
11. Open the **Utility** menu by clicking **View > Browsers > HyperMesh > Utility** from the menu bar.
12. In the **Utility** menu, click **Disp**.
13. Click **Clear Temp Nodes**.
14. In the **Model** browser, right-click and select **Create > Component** from the context menu.
15. In the **Create component** dialog:
  - For **Name**, enter **MPC\_Same**.
  - Set **Card image** to **none**.
  - Click **Create**.
16. Open the **Element Type** panel.
17. Click **rigid =**, and select **MPC SAME**.
18. Click **return** to close the panel.

**Note:** MPC\_Same should now be your current component collector. If not, right-click on **MPC\_Same**, in the **Model** browser, and select **Make Current** from the context menu.
19. Open the **Rigids** panel. A **MPC Same** rigid will be used to connect the two beams created in Step 6, where there is a small gap between each vertical section. You will use the **Rigids** panel to connect these two sections with a **MPC Same** rigid.
20. Select all of the **DOF** checkboxes, except **dof2**. This one is set free for the pretension load which is applied later in the tutorial.
21. Set the **dependent** switch to **single node**.
22. Using the **independent** node selector, select the inner end of one of the beams.





23. Using the **dependent** node selector, select the inner end of the other beam. The **MPC Same rigid** is created.



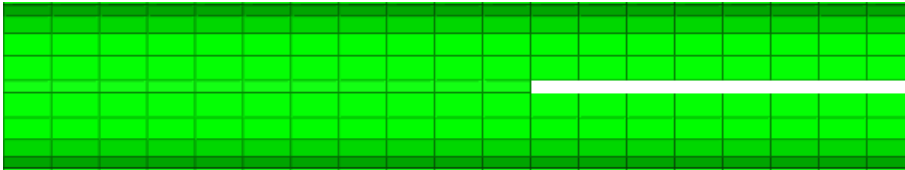
24. Click **return** to close the panel.

### Step 9: Set definition

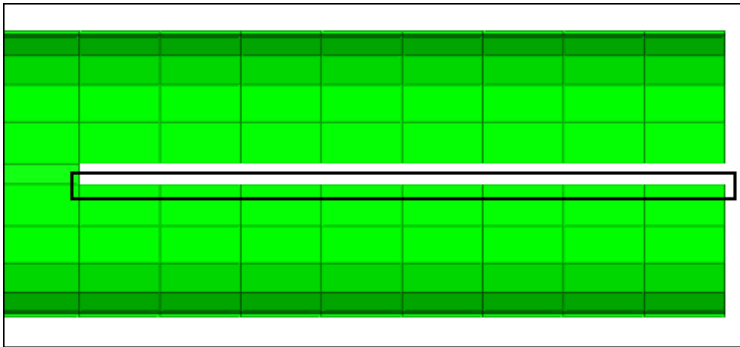
In this step you will create sets that will later be used for contact definition. The sets that need to be created include:

- SET\_1: lower edge of the split portion of the tube
- SET\_2: upper edge of the split portion of the tube

- SET\_11: lower node of 1st bolt connection
  - SET\_12: upper node of 1st bolt connection
  - SET\_21: lower node of 2nd bolt connection
  - SET\_22: upper node of 2nd bolt connection
  - Clamps: nodes of the contact surfaces at the clamps to the tube
1. In the **Model** browser, **View** folder, right-click on **set\_def** and select **Show** from the context menu.




2. From the menu bar, click **Tools > Create > Sets**. The **Sets** panel opens.
3. In the **name** field, enter SET\_1.
4. While pressing **SHIFT**, draw a box around the lower edge of the split portion of the gap. All nodes within this section highlight.

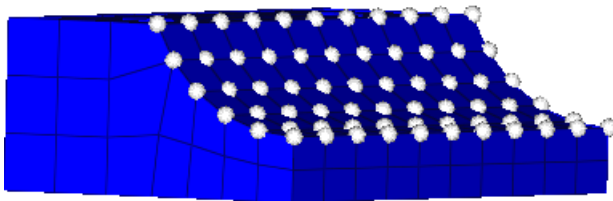
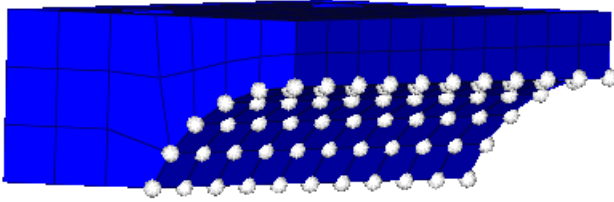


5. Click **create**. HyperMesh organizes all of the highlighted nodes into SET\_1.
6. In the **Model** browser, **Component** folder, turn on the display of **Beam** and **MPC Same**.
7. Repeat steps 9.3 – 9.5 to create SET\_2, SET\_11, SET\_12, SET\_21, and SET\_22 using the definitions above.

Use the following parameters to help you create the sets:

- **SET\_2** – Select the top set of nodes on the opening that was selected in step 4.
- **SET\_11** – Zoom in on the MPC SAME rigid, and select one node on the bottom of the first MPC SAME rigid created in Step 7.
- **SET\_12** – Zoom in on the MPC SAME rigid, and select one node on the top of the first MPC SAME rigid created in Step 7.
- **SET\_21** – Zoom in on the MPC SAME rigid, and select one node on the bottom of the second MPC SAME rigid created in Step 7.
- **SET\_22** – Zoom in on the MPC SAME rigid, and select one node on the top of the second MPC SAME rigid created in Step 7.

6. Click **return**.
7. In the **Model** browser, **Component** folder, right-click on **clamps1** and select **Isolate Only** from the context menu.
8. On the **Standard Views** toolbar, click  (**XY Top Plane View**).
9. From the menu bar, click **Tools > Create > Sets**.
10. In the **name** field, enter `Clamps`.
11. Click **nodes >> by window**.
12. Draw a box to select the nodes indicated in the image below.



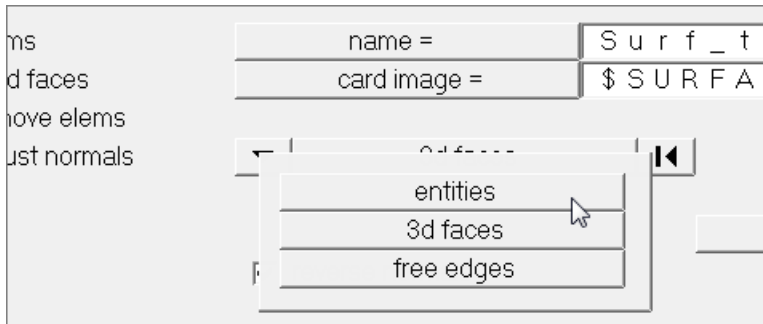
13. Click **create**. HyperMesh creates the set.
14. Click **return** to close the panel.

Tip: When you have a large amount of sets, use the **Set** browser to create sets.

## Step 10: Creating a surfaces from solid elements

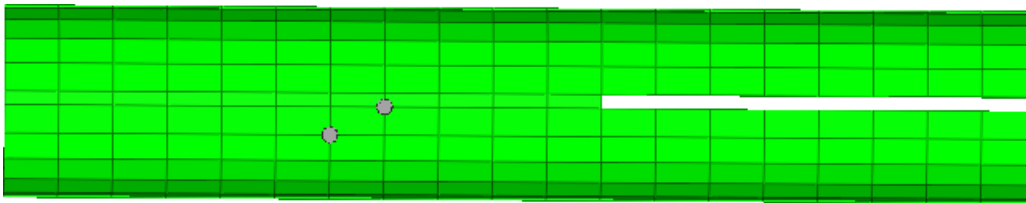
The surface definition and the sets from above will be used later for contact definition.

1. In the **Model** browser, **Component** folder, right-click on **Tube** and select **Isolate Only** from the context menu.
2. Open the **Contact Surfaces** panel by clicking **BCs > Create > Contact Surfaces** from the menu bar.
3. Go to the **solid faces** subpanel.
4. In the **name** field, enter `Surf_tube`.
5. Click **card image =**, and select **\$SURFACE**.
6. Set the switch to entities.



7. Click **elems** >> **by collector**.
8. Select the component, **tube**.
9. Click **select**.
10. Using the **nodes** selector, select two nodes on the **tube** component. These nodes should be on one solid element, diagonal from each other.

**Note:** These nodes will be used to specify the faces of the solids to be taken into account for this surface creation.



11. Click **create**. HyperMesh creates the contact surface.
12. Click **return** to exit the panel.

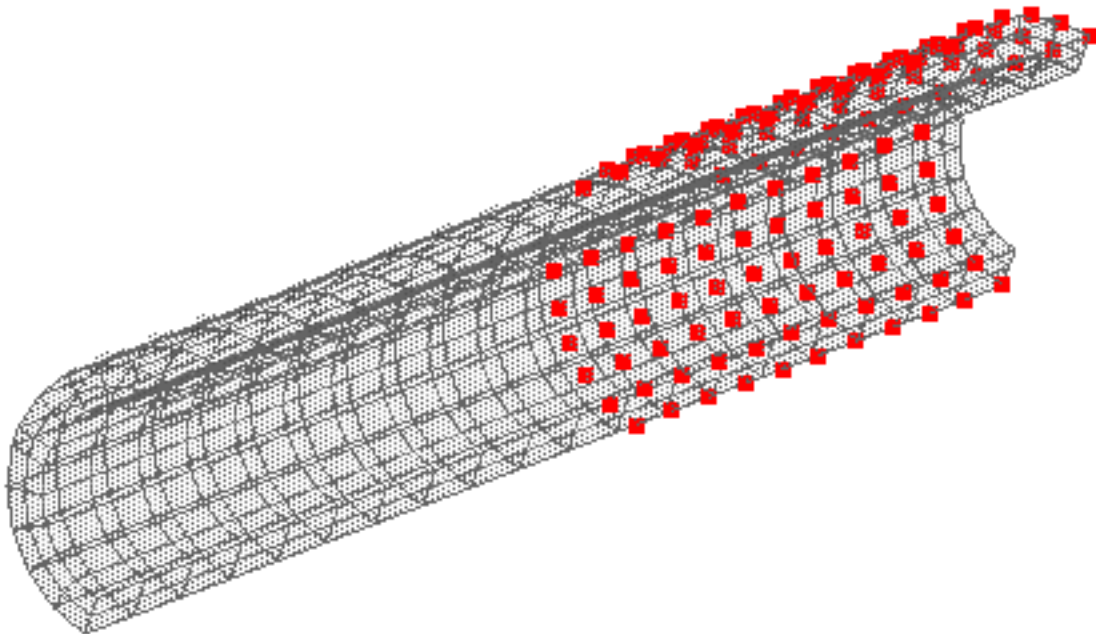
### Step 11: Define the contact partners using \$CONTACT

In this analysis, four contacts will be established.

Contact name	Type	Master	Slave
Bolt_1	Node to Node	SET_11	SET_12
Bolt_2	Node to Node	SET_21	SET_22
Gap	Node to Node	SET_1	SET_2
Tube_clamps	Surface to Node	Surf_Tube	Clamps

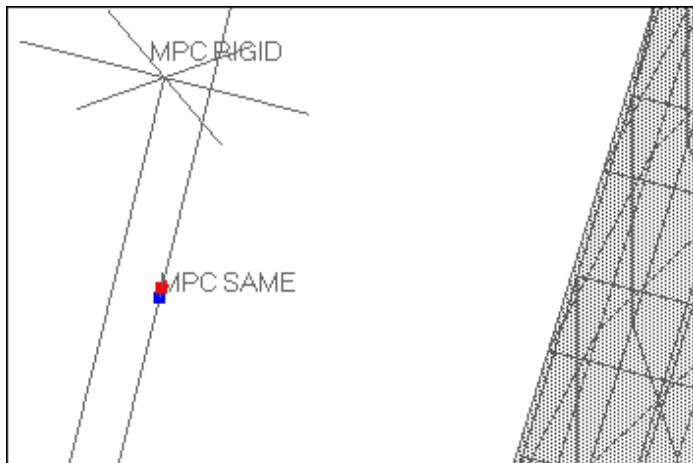
1. Open the **Interfaces** panel by clicking **BCs** > **Create** > **Interfaces** from the menu bar.
2. Go to the **create** subpanel.
3. In the **name** field, enter `tube_clamps`.

4. Click **type =**, and select **CONT\_SURFACE\_NODE** to create a surface to node contact
5. Choose a color for the interface.
6. Click **create**.
7. Go to the **add** subpanel.
8. Set the **master** switch to **csurfs**.
9. Using the **contactsurfs** selector, select **surf\_tube**.
10. Click **update**.
11. Set the **slave** switch to **sets**.
12. Using the **sets** selector, select **Clamps**.
13. Click **update**.
14. Click **review**. You should now be able to see if you have set up master and slave surfaces correctly for your contact definition. Master surfaces display in blue, and slaves display in red. If one (or both) do not display, then you might have missed clicking **update**. If this happens, select the missing entity (master or slave) again and click **update**.

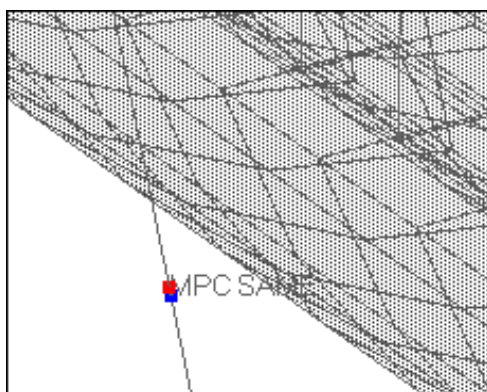


15. Click **return** to close the panel.
16. In the **Model** browser, **Component** folder, turn on the display of **Beam**, **MPC Rigid** and **MPC Same**.
17. From the menu bar, click **BCs > Create > Interfaces**.
18. Go to the **create** subpanel.
19. In the **name** field, enter `Bolt_1`.
20. Click **type =**, and select **CONT\_NODE\_NODE** for a node to node contact.

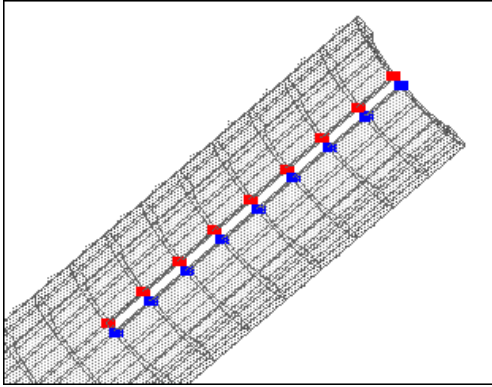
21. Choose a color for the interface.
22. Click **create**.
23. Go to the **add** subpanel.
24. Set the **master** switch to **sets**.
25. Using the **sets** selector, select **SET\_11**.
26. Click **update**.
27. Set the **slave** switch to **sets**.
28. Using the **sets** selector, select **SET\_12**.
29. Click **update**.
30. Click **review**.
31. Rotate the model to view the interface.



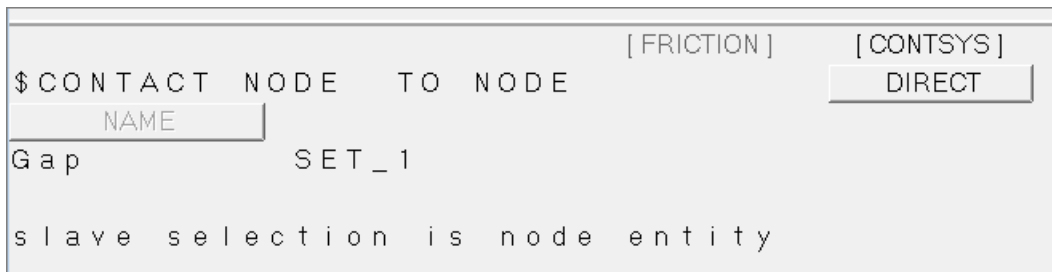
32. Click **return** to close the panel.
33. Create the **Bolt\_2** interface, with the following criteria:
  - Interface type – CONT\_NODE\_NODE
  - Master set – SET\_21
  - Slave set – SET\_22



34. Isolate the display of the **tube** component, and create the interface named **Gap**, with the following criteria:
- Interface type – CONT\_NODE\_NODE
  - Master set – SET\_1
  - Slave set – SET\_2



35. In the **Model** browser, **Group** folder, right-click on **Gap** and select **Card Edit** from the context menu.
36. In the card image of the \$CONTACT card, click **CONTSYS** and select **DIRECT**.




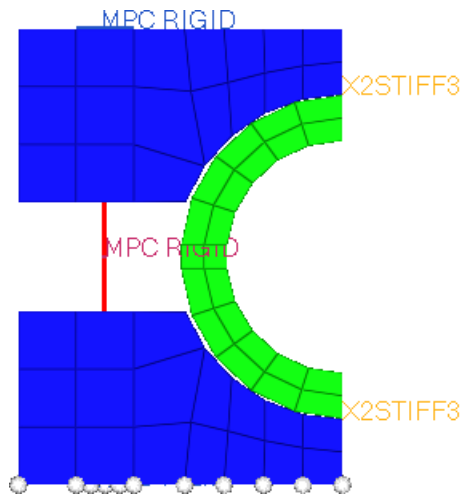
37. Click **return**.
38. Repeat steps 11.35 – 11.36 for the **Bolt\_1** and **Bolt\_2** interfaces.

## Step 12: Create the boundary conditions

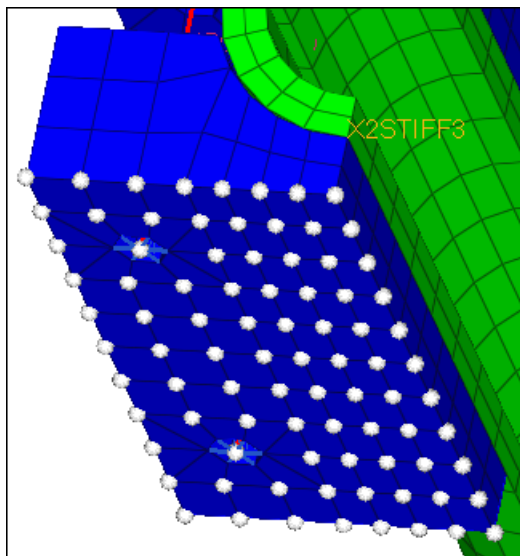
In this step you will create the constraints applied to this simulation, and group them into one load collector named **Boundary**. Later you will create a \$CONSTRAINTS variant (load step in HyperMesh) and attach the load collector with the previously created contact constraints.

1. In the **Model** browser, right-click and select **Create > Load Collector** from the context menu.
2. In the **Create Load Collector** dialog:
  - For **Name**, enter `Boundary`.
  - Select a color for the load collector.
  - Set the **Card image** to **SUPPRESS**.

- Click **Create**.
3. On the **Standard Views** toolbar, click  (**XY Top Plane View**).
  4. Open the **Constraints** panel by clicking **BCs > Create > Constraints** from the menu bar.
  5. Clear the **dof4**, **dof5** and **dof6** checkboxes.
  6. Verify that the **Clamps1** component is displayed.
  7. While pressing **SHIFT**, draw a box around the bottom row of elements to select the nodes.



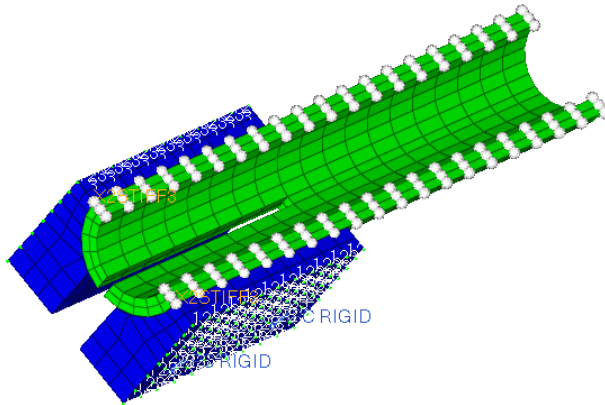
8. Rotate the model so that the underside of the bottom clamp is shown.
9. Right-click on the nodes around the MPC Rigids to deselect them.



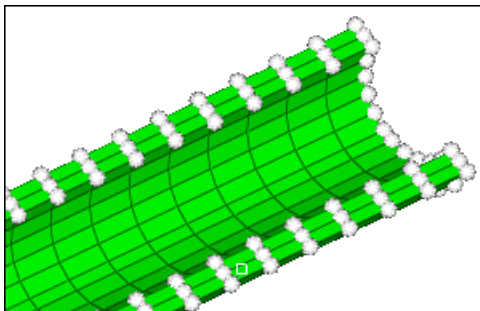
10. Deselect the nodes surrounding the other **MPC Rigid** on this side of the clamp as well.
11. Click **create**.
12. In the **Model** browser, **Component** folder, display **tube**.



13. In the **size** field, enter 1.0 to reduce the size of the constraints in the graphics area.
14. While pressing **SHIFT**, draw a box around the far end of the tube to select those nodes in order to include them in the constraint.



15. Click **nodes >> on plane**.
16. Select three nodes on the narrow edge of the tube, and then click **select entities**. All of the nodes on that plane highlight.

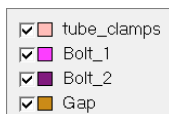


17. Click **create** to create the constraints.
18. Click **return** to exit the panel.
19. Turn on the display of all of the components again.

All of the constraints have been created. In the next step you will include the constraints in a **\$CONSTRAINTS** variant.

### Step 13: Assign the boundary conditions to a load step

1. From the menu bar, click **Setup > Create > Load Steps**.
2. In the **name** field, enter `Con_1`.
3. Using the **loadcols** selector, select **Boundary**.
4. Using the **groups** selector, select all four groups.



5. Click **create**.
6. Click **edit**.
7. In the card image, set **AnalysisProcedure** to **CONSTRAINTS**.

**Note:** If the **Analysis Procedure** field is not set to **constraints**, the load collectors are included, but will not be exported.

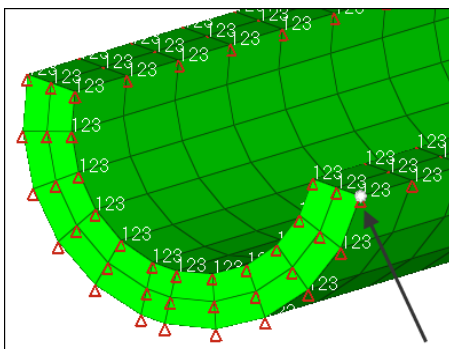
8. Click **return** twice to close the panels.

**Note:** You can only attach a load collector card image SUPPRESS to a load step with analysis procedure CONSTRAINTS and a load collector with card image LOADS in one with analysis procedure LOADING. If this is not followed correctly your loads and boundary conditions will not get exported. Besides that you will find a warning in the card image of the load step as well as a comment in the exported deck.

### Step 14: Define the load and contact property

In this step, you will create one load pattern (LPAT) for a force applied to the tip of the tube as well as the contact properties for the contacts created in the earlier part of this tutorial. In HyperMesh a load pattern is represented by a load collector with card image LOADS. Finally these load patterns will be included into a \$LOADING variant. The LOADING variant is mapped to a load step in HyperMesh.

1. In the **Model** browser, right-click and select **Create > Load Collector** from the context menu.
2. In the **Create Load Collector** dialog:
  - For **Name**, enter `Single_Load`.
  - Select a color for the load collector.
  - Set **Card image** to **LOADS**.
  - Click **Create**.
3. Open the **Forces** panel by clicking **BCs > Create > Forces** from the menu bar.
4. Using the **nodes** selector, select the node shown in the image below.



5. Set the orientation selector to **y-axis**.
6. In the **magnitude=** field, enter `-12000`.
7. Click **create**.

8. In the **Model** browser, right-click and select **Create > Load Collector** from the context menu.
9. In the **Create Load Collector** dialog:
  - For **Name**, enter `Pretension_Bolt1`.
  - Select a color for the load collector.
  - Set **Card image** to **LOADS**.
  - Select the **Card edit loadcollector upon creation** checkbox.
  - Click **Create**.
10. In the card image, select the **CONTVAL** checkbox.
11. Under **No\_of\_Contacts**, enter 1.
12. Double-click **contid(1)**, and select **Bolt\_1**.
13. Click **GAPWIDTH**, and select **ABS**.
14. Click **NORMAL**, and select **FORCE\_LOCK**.
15. In the **ContactForce(1)** field, enter 10000.

NAME	LPAT	[ GAPWIDTH ]	[ NORMAL ]	[ FRICTION ]	ApplyContact_
▲ Pretension_Bolt1	3				
\$CONTVAL ( 1 )	3	ABS	FORCE_LOCK		ALL
No_Of_Contacts					
	1				
contid(1)	gapWidth(1)	ContactForce(1)			
▼	2	0.000	10000.000		
<input type="checkbox"/> PREVAL <input checked="" type="checkbox"/> CONTVAL No_Of_Keywords = <input type="text" value="1"/> <input type="checkbox"/> PRETENSION_LOAD					

16. Click **return** to close the card image.
17. In the **Model** browser, right-click and select **Create > Load Collector** from the context menu.
18. In the **Create Load Collector** dialog:
  - For **Name**, enter `Pretension_Bolt2`.
  - Select a color for the load collector.
  - Set **Same as** to **Pretension\_Bolt1**.
  - Click **Create**.
19. In the card image, double-click **contid(1)** and select **Bolt\_2**.
20. Click **return** to close the card image.
21. In the **Model** browser, right-click and select **Create > Load Collector** from the context menu.

22. In the **Create Load Collector** dialog:
  - For **Name**, enter `Gap_Tube_Clamps`.
  - Select a color for the load collector.
  - Set **Card image** to **LOADS**.
  - Select the **Card edit loadcollector upon creation** checkbox.
  - Click **Create**.
23. In the card image, select the **CONTVAL** checkbox.
24. Under **No\_of\_Contacts**, enter 2.
25. Double-click **contid(1)**, and select **Gap**.
26. Double-click **contid(2)**, and select **tube\_clamps**.
27. Click **GAPWIDTH**, and select **ABS**.
28. Click **NORMAL**, and select **CALCULATE**.
29. Click **return** to close the card image.

## Step 15: Define the \$NLLOAD cards

In this step you will create a \$NLLOAD card to specify how individual load patterns are acting over an artificial time.

1. Open the **Load Steps** panel by clicking **Setup > Create > Load Steps** from the menu bar.
2. In the **name** field, enter `NLLOAD`.
3. Using the **loadcols** selector, select the following load collectors: **Single\_Load**, **Pretension\_Bolt1**, **Pretension\_Bolt2**, and **Gap\_Tube\_Clamps**.
4. Click **create**.
5. Click **edit** to open the card image.
6. Select the **NLLOAD** checkbox.
7. In the **TimeSteps** field, enter 4.
8. In the **NoOfLPATS** field, enter 4.
9. Using the **LPAT** selectors, select load collectors as follows:
  - LPAT(1) - Gap\_Tube\_Clamps
  - LPAT(2) - Pretension\_Bolt1
  - LPAT(3) - Pretension\_Bolt2
  - LPAT(4) - Single\_Load

10. Enter the following information for each of the load patterns as shown in the image below:

▲	LPAT(1)	T(0,0)	T(0,1)	T(0,2)	T(0,3)
	5	1 . 0 0 0	- 9 9 9 . 0 0 0	- 9 9 9 . 0 0 0	- 9 9 9 . 0 0 0
	LPAT(2)	T(1,0)	T(1,1)	T(1,2)	T(1,3)
	3	0 . 0 0 0	1 . 0 0 0	1 . 0 0 0	- 9 9 9 . 0 0 0
	LPAT(3)	T(2,0)	T(2,1)	T(2,2)	T(2,3)
	4	0 . 0 0 0	0 . 0 0 0	1 . 0 0 0	- 9 9 9 . 0 0 0
	LPAT(4)	T(3,0)	T(3,1)	T(3,2)	T(3,3)
▼	2	0 . 0 0 0	0 . 0 0 0	0 . 0 0 0	1 . 0 0 0

11. Click **return** twice to close the panels.

The NLLOAD card has been defined. Use the **Plot NNLOAD** tool, located on the **Utility** menu, to review the contents of the card. You can see the load history of each load pattern plotted as a graph. You also can edit values within the table of the tool. See the topic [Creating an NLLOAD Card](#) in the HyperMesh online help for more information.

**Note:** The entry fields for the NLLOAD entries require real numbers, and cannot understand hyphens, which are used commonly by PERMAS users. However, to define a hyphen for export you can write -999 which will display as '-' in the exported deck. This applies to imported files, too. A hyphen will convert to '-999' on import.


## Step 16: Define the \$SITUATION card

In this step, you will build a \$SITUATION card which specifies which loads, boundary conditions, and system definitions are combined for this analysis.

1. Open the **Load Steps** panel.
2. In the **name** field, enter `SIT_NLLOAD`.
3. Click **create**.
4. Click **edit** to open the card image.
5. Set **AnalysisProcedure** to **SITUATION**.
6. Using the **CONSTRAINTS** selector, select **Con\_1**.
7. Using the **LOADING** selector, select **NLLOAD**.
8. Click **return** twice to close the card image and then the panel.

## Step 17: Export the deck to a .dat file

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

3. Next to **Export Options**, click .
4. Set **Export** to **All**.
5. Click **Export**.

### **Acknowledgements**

Altair thanks INTES for their assistance and support during the creation of this tutorial.