



Altair

HyperWorks

HM-4300: Setting Up an Abaqus Analysis in HyperMesh

In this tutorial, you will learn how to:

- Load the Abaqus user profile and a model
- Define a material and properties, and assign them to a component
- View a *SOLID SECTION for solid elements
- Define *SPRING properties and create a component collector for them
- Create a *SPRING1 element
- Assign a property to selected elements


Model Files

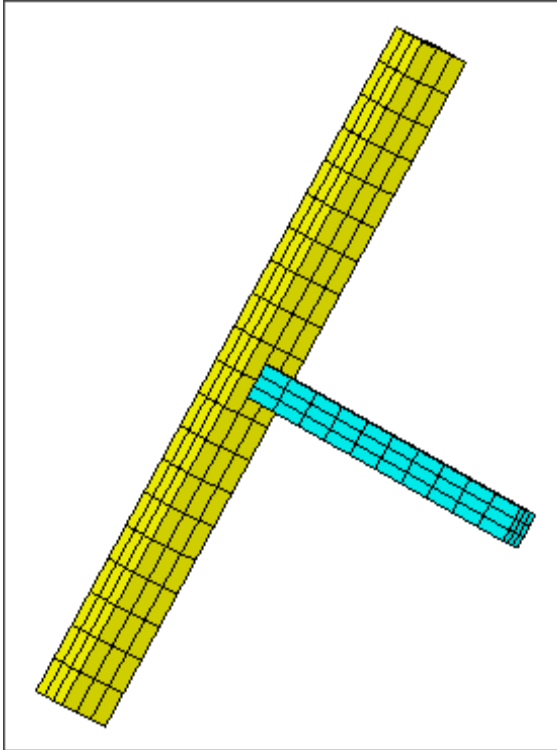
This exercise uses the `abaqus3_0tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Load the Abaqus user profile and model

A set of standard user profiles are included in the HyperMesh installation. They include: OptiStruct, RADIOSS, Abaqus, Actran, ANSYS, LS-DYNA, MADYMO, Nastran, PAM-CRASH, PERMAS, and CFD. When you load a user profile, HyperMesh opens the applicable utility menus, removes the unused panels, disables unneeded entities in the **Find**, **Mask**, **Card** and **Reorder** panels, and makes specific adaptations related to the Abaqus solver.

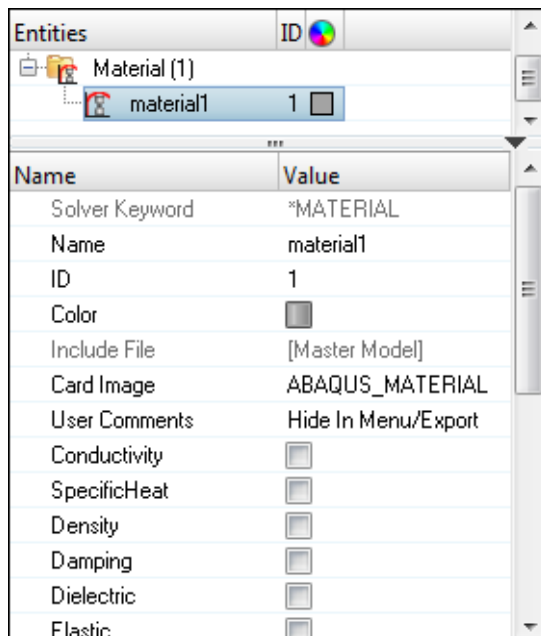
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard 3D**.
3. Open a model file by clicking **File > Open > Model** from the menu bar, or clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus3_0tutorial.hm` file. The model appears in the graphics area.




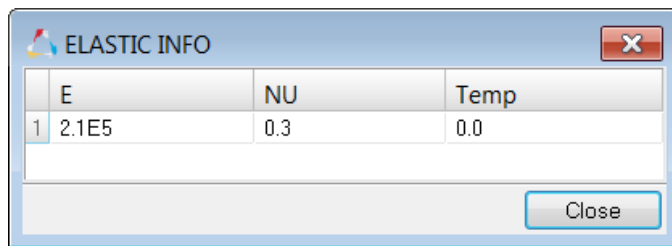
Step 2: Define the material properties

HyperMesh supports many different material models for Abaqus. In this step, you will create the basic *ELASTIC material model with no temperature variation. You will then assign the material to the property, which is assigned to a component collector.

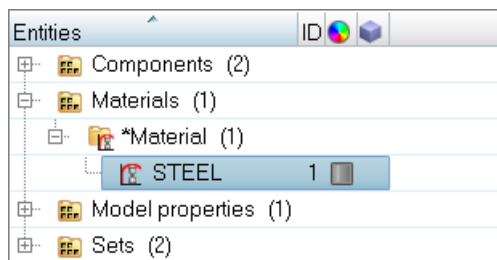
1. In the **Model** browser, right-click and select **Create > Material** from the context menu. HyperMesh creates and opens a material in the **Entity Editor**.



2. For **Name**, enter `STEEL`.
3. Optional: For **ID**, enter a new ID for the material.
Note: By default, HyperMesh sets the ID to 1. If you create a new material, HyperMesh will set the ID to $n+1$.
4. Optional: Click the **Color** icon, and select a color to display the material.
5. Select the **Elastic** checkbox. HyperMesh displays additional parameters. By default, the selected type is **ISOTROPIC**, and the **ELASTIC INFO** field value is 1.
6. In the **Data: E** field, click . The **ELASTIC INFO** dialog opens.
7. For **E(1)**, enter `2.1E5`.
8. For **NU(1)**, enter `0.3`.
9. Click **Close**.

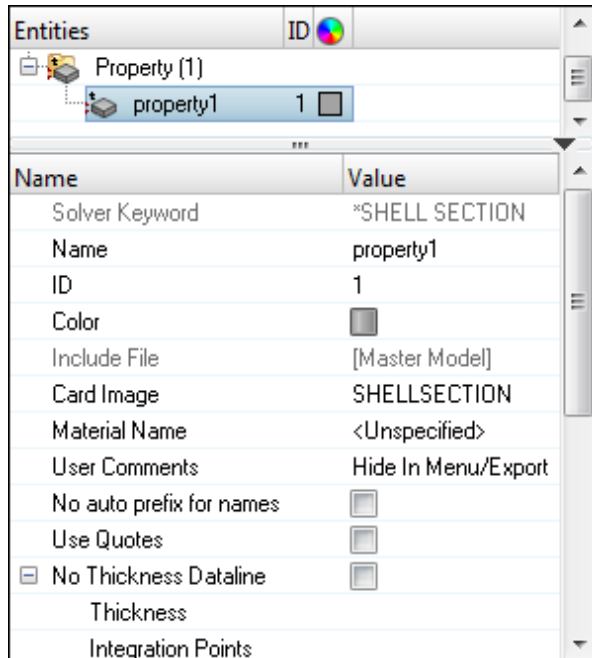


10. Open the **Solver** browser by clicking **View > Browsers > HyperMesh > Solver** from the menu bar.
11. In the **Solver** browser, review the material you just created.



Step 3: Define the *SOLID SECTION properties

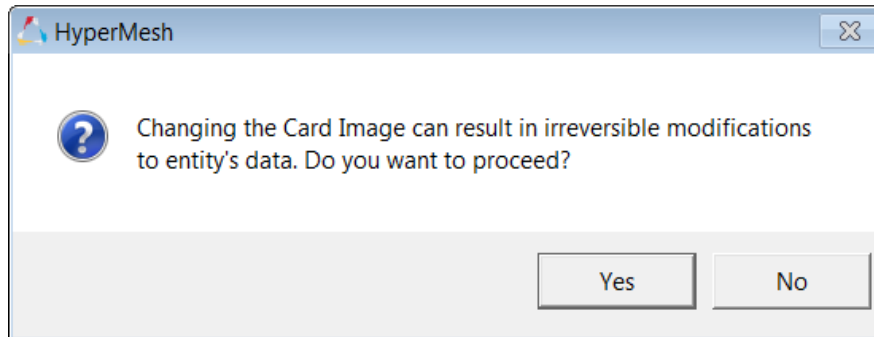
1. In the **Model** browser, right-click and select **Create > Property** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.



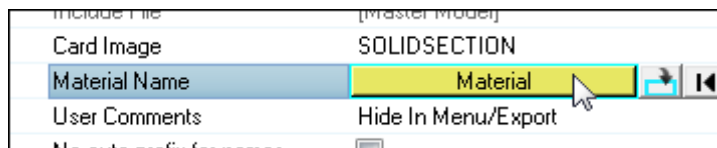
2. For **Name**, enter `Solid_Prop`.
3. Click the **Color** icon, and select a color to display the property.
4. For **Card Image**, select **SOLIDSECTION**.

Note: This ensures that sections pertaining only to solid elements are available as card image options.

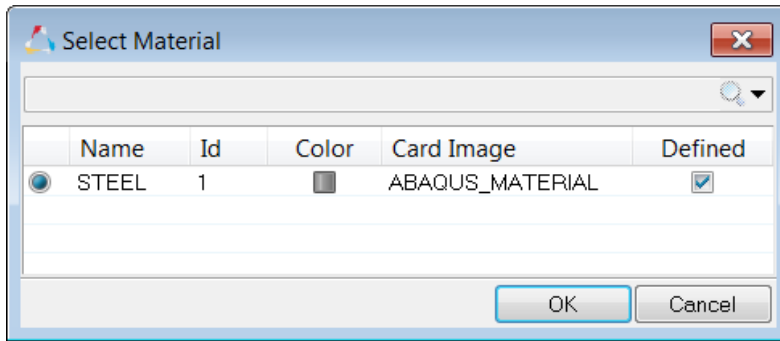
5. In the **HyperMesh** dialog, click **Yes** to proceed.



6. For **Material Name**, click **Unspecified** >> **Material**.



7. In the **Select Material** dialog, select **STEEL** and then click **OK**. HyperMesh assigns the material **STEEL** to the property **Solid_Prop**.

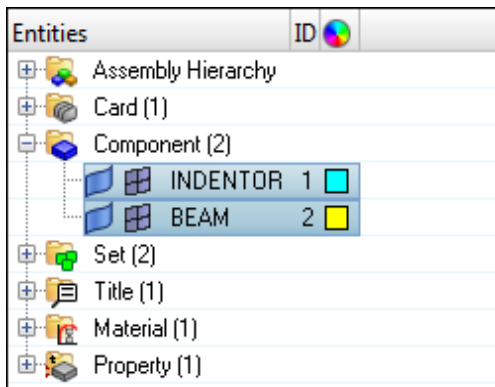


Step 4: Assign the property to the component

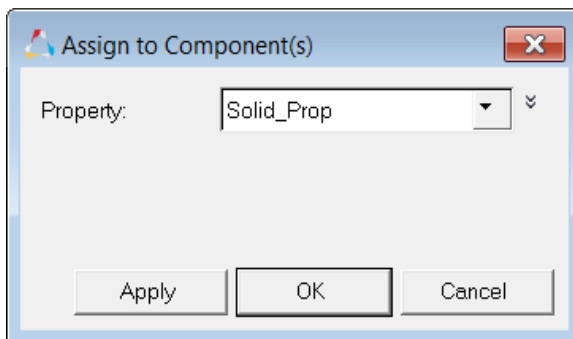
When a material is assigned to a property, when a property is assigned to a component, the material is automatically assigned as well.

1. In the **Model** browser, **Component** folder, select **BEAM** and **INDENTOR**.

Tip: Select multiple components by pressing **CTRL** while selecting components.



2. Right-click on the highlighted components and select **Assign** from the context menu.
3. In the **Assign to Component(s)** dialog, select **Solid_Prop** from the **Property** list.

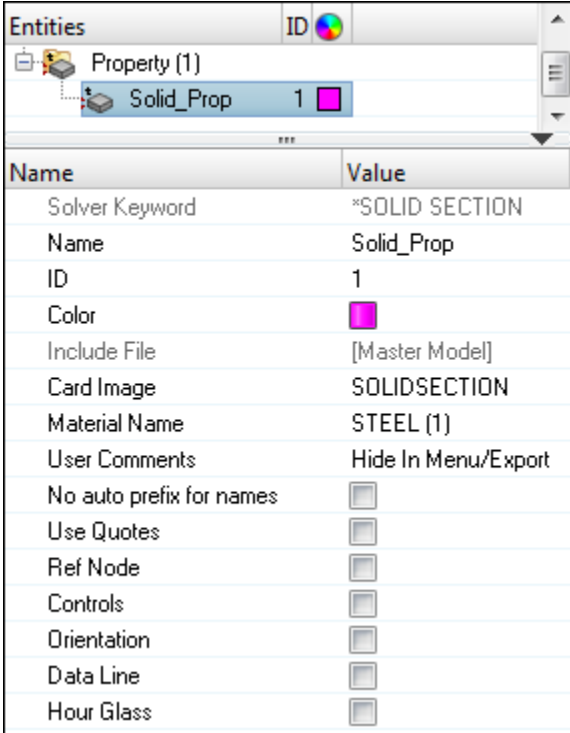



4. Click **OK**. HyperMesh assigns the property **Solid_Prop** to the components **BEAM** and **INDENTOR**.

Step 5: View the *SOLID SECTION for solid elements

HyperMesh supports sectional properties for all elements from the property collector. In this step you will view the *SOLID SECTION card for an existing component.

1. In the **Model** browser, **Property** folder, select **Solid_Prop**. HyperMesh opens the **Entity Editor**, and displays the property's corresponding data. The **Card Image** shows the keyword related to this property.




Name	Value
Solver Keyword	*SOLID SECTION
Name	Solid_Prop
ID	1
Color	
Include File	[Master Model]
Card Image	SOLIDSECTION
Material Name	STEEL (1)
User Comments	Hide In Menu/Export
No auto prefix for names	<input type="checkbox"/>
Use Quotes	<input type="checkbox"/>
Ref Node	<input type="checkbox"/>
Controls	<input type="checkbox"/>
Orientation	<input type="checkbox"/>
Data Line	<input type="checkbox"/>
Hour Glass	<input type="checkbox"/>

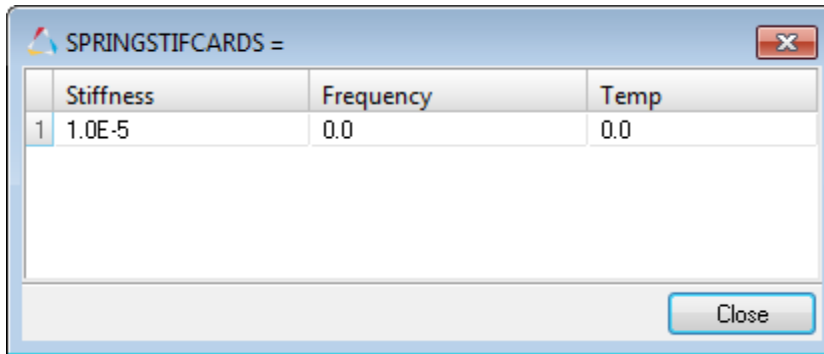
Step 6: Define the *SPRING properties

In Abaqus contact problems, it is common to use grounded springs to provide stability to the solution in the first loading step. In this step, you will create these springs and the *SPRING card.

1. In the **Model** browser, right-click and select **Create > Property** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.
2. For **Name**, enter Spring_Prop.
3. Click the **Color** icon, and select a color to display the property.
4. For **Card Image**, select **SPRING**.
5. In the **HyperMesh** dialog, click **Yes** to proceed.
6. For **dof1**, enter 3.

Note: The **dof2** parameter in the *SPRING card is ignored by Abaqus for SPRING1 elements.

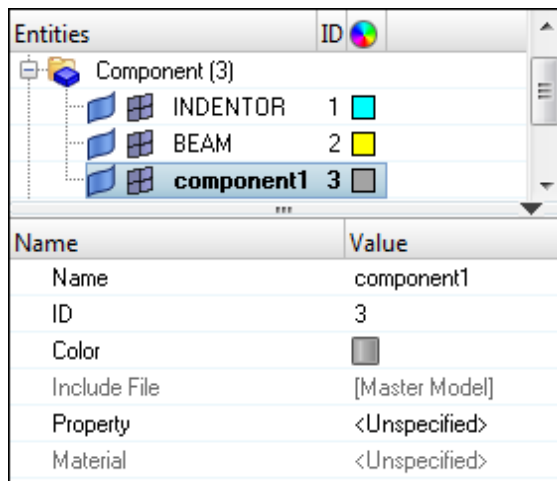
7. In the **Data: Stiffness** field, click . The **SPRINGSTIFCARDS=** dialog opens.
8. For **Stiffness(1)**, enter 1.0E-5.



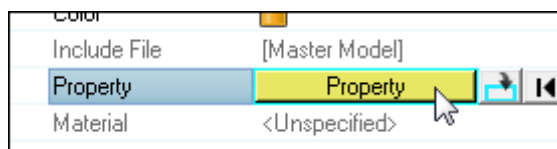
9. Click **Close**.

Step 7: Create a component collector for the *SPRING property

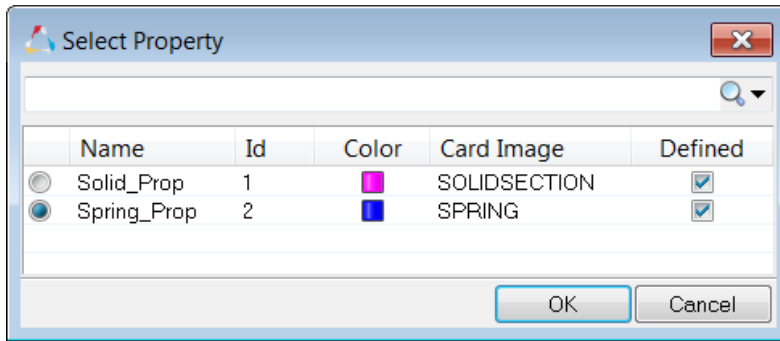
1. In the **Model** browser, right-click and select **Create** > **Component** from the context menu. HyperMesh creates and opens a component in the **Entity Editor**.



2. For **Name**, enter GROUNDED.
3. Click the **Color** icon, and select a color to display the component.
4. For **Property**, click **Unspecified** >> **Property**.

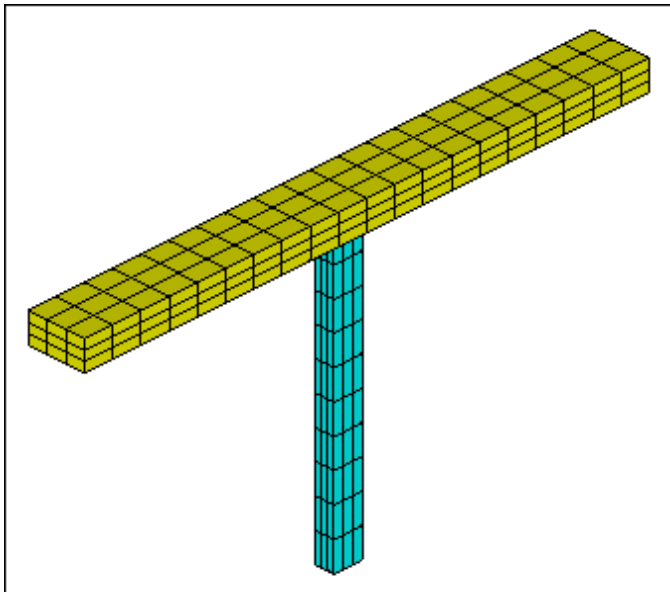


5. In the **Select Property** dialog, select **Spring_Prop** and then click **OK**. HyperMesh assigns the property **Spring_Prop** to the component **GROUNDED**.



Step 8: Reset the view for further processing

1. On the **Standard Views** toolbar, click .



Step 9: Create the SPRING1 element

1. Open the **Element Type** panel by clicking **Mesh > Assign > Element Type** from the menu bar.
2. Go to the **1D** subpanel.
3. Click **mass =**, and then select **SPRING1**.

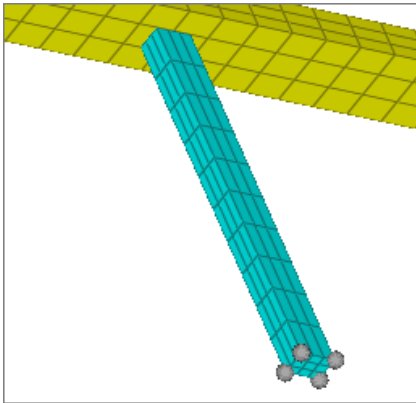
Note: In HyperMesh, grounded elements are created and stored as mass elements since they only have one node in the element connectivity.

4. Exit the panel by clicking **return**.
5. In the **Model** browser, **Component** folder, right-click on **GROUND** and select **Make Current** from the context menu.

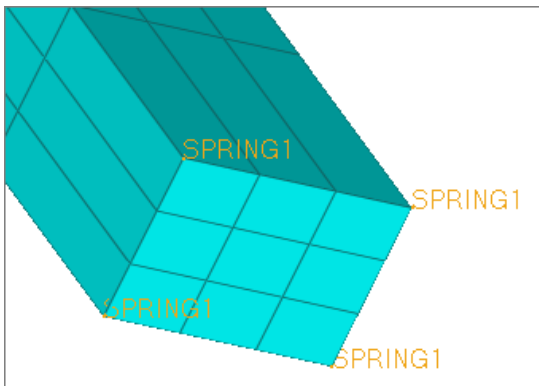
Note: As the spring elements are created, they will be placed in this component.

6. Open the **Masses** panel by clicking **Mesh > Create > Masses** from the menu bar.

- Click **nodes** >> **by id**.
- In the **id =** field, enter 451t460b3 and then press **Enter**. HyperMesh selects all of the nodes from 451 to 460 in increments of 3.



- Click **create**. HyperMesh creates SPRING1 elements.



- Click **return**.

Step 10-11: Assign a property to selected elements

In most cases, a property (for example, SHELL SECTION, SOLID SECTION) is assigned to a component collector. The elements belonging to the component collector assume the assigned property.

It is also possible in HyperMesh 11.0 to assign a property to individual elements without having to organize these elements in a separate component collector. A property can be assigned directly to selected elements. In this case, HyperMesh automatically creates an Abaqus element set (ELSET) for the selected elements and assigns the property to the ELSET.

The name of the ELSET follows the naming convention `HMprop_propertyname`, where `propertyname` is the name of the property collector that is assigned to the elements. In the example below, a property collector called `property1` is created and assigned to elements 1. This information is translated in the Abaqus `*.inp` file as:

```

**      Template:  ABAQUS/STANDARD 3D
**
**NOD
      1,  2.5          ,  0.0          ,  2.5
      2,  2.5          ,  0.0          , -2.5
      3, -2.5          ,  0.0          , -2.5
      4, -2.5          ,  0.0          ,  2.5
**HWCOLOR COMP          1      11
**ELEMENT,TYPE=S4R,ELSET=auto1
      1,          1,          2,          3,          4
**ELSET, ELSET=HMprop_property1
      1
**HM_set_by_property  11          22
**SHELL SECTION, ELSET=HMprop_property1, MATERIAL=

```

The HMprop_ prefix can be suppressed in the property's card image by select the **No_auto_prefix_for_names** check box.

The comment ****HM_set_by_property 11** is written by HyperMesh to the *.inp file so that the element property definition and assignment are read properly when the *.inp file is imported in HyperMesh. The number 11 refers to the color chosen for the property and 22 as the property ID.


Step 10: Create a property

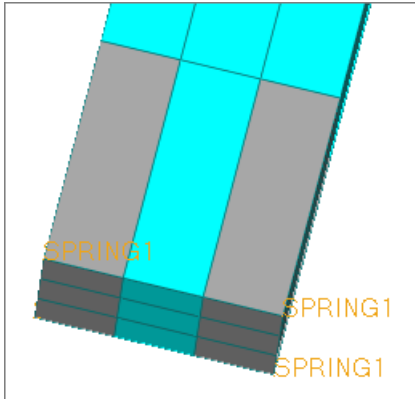
1. In the **Model** browser, right-click and select **Create > Property** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.
2. For **Name**, enter `ElemPrp`.
3. Click the **Color** icon, and select a color to display the property.
4. For **Card Image**, select **SOLIDSECTION**.

Note: This ensures that sections pertaining only to solid elements are available as card image options.

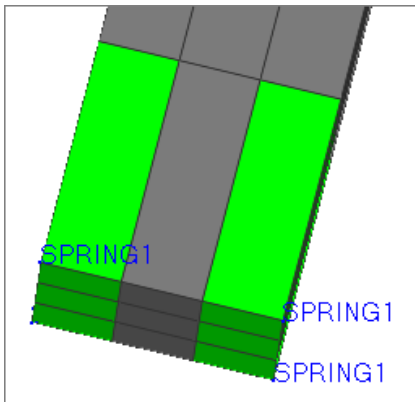
5. In the **HyperMesh** dialog, click **Yes** to proceed.

Step 11: Assign a property to individual elements

1. In the **Model** browser, **Property** folder, right-click on **ElemPrp** and select **Assign** from the context menu.
2. On the **Standard Views** toolbar, click .
3. In the graphics area, select the leftmost and rightmost layers of solid elements belonging to the horizontal BEAM component.



4. Click **proceed**. HyperMesh assigns the property.
5. In the **Mixed Property Warning** dialog, click **OK**.
6. On the **Visualization** toolbar, select **By Prop** from the **Element Color Mode** list. HyperMesh colors the elements by their property assignment.



When a property is assigned to a component collector, HyperMesh writes out the comment `**HM_comp_by_property` to distinguish the component property assignment from that of individual element.