Altair

HyperWorks

HM-4850: Introduction to Samcef

In this tutorial, you will learn how to:

- Load the SAMCEF user profile
- Retrieve the HyperMesh model files for this tutorial
- Define the hypothesis on the elements
- Create an isotropic material
- Create a shell property
- Assign the property and the material to the elements
- Create loads and the load cases associated with them
- Export the model in SAMCEF format

Model Files

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

Exercise

Step 1: Load the SAMCEF user profile

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

Step 2: Retrieve the model file

1. Open a model file by clicking *File* > *Open* > *Model* from the menu bar.
2. In the **Open Model** dialog, open the file `Hook.hm`.
3. On the **Visualization** toolbar, click 🧊 to shade the models elements and mesh lines, if they are not already shaded.
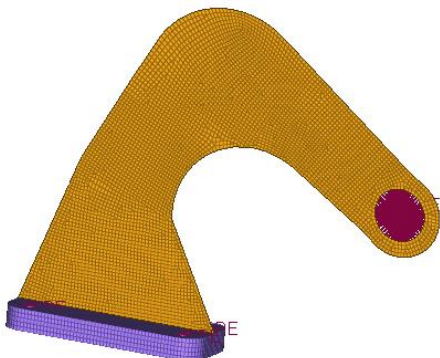
Step 3: Check the hypothesis

1. Open the **Element Types** panel by clicking ***Mesh > Assign > Element Types*** from the menu bar.

2. Go to the **2D & 3D** subpanel.

3. In this panel, you have the link between the keyword of HyperMesh (tria3, quad4, ...) and the hypothesis of SAMCEF. You can change the hypothesis for each element configuration and also update each element with a different hypothesis.
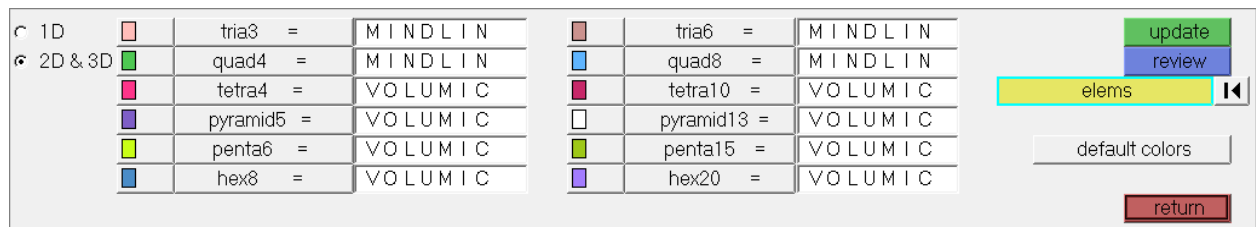
4. Click ***return*** to close the panel.

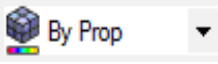## Step 4: Create the material

1. In the **Model** browser, right-click and select ***Create* > *Material*** from the context menu.

2. In the **Create Material** dialog:

    *   For **Name**, enter `steel`.

    *   Set **Card image** to ***ISOTROPIC***.

        Note: You can create different types of elastic material (anisotropic, isotropic or orthotropic).

    *   Select a color for the material.

    *   Select the ***Card edit material upon creation*** checkbox.

    *   Select ***Create***. The isotropic card image displays.

        Note: If a material property in brackets does not have a value below it, this indicates that it is turned off. Edit these material properties by clicking on the property in the brackets you wish to edit. In the text field that appears below it, enter a value.

3. For **YT**, enter `2.1e5`.

4. For **NT**, enter `0.3`.

5. For **M**, enter `7.9e-9`.

6. Click ***return***.

    A new material named steel has been created. At any time, the card image for this material can be modified using the card editor.

Step 5: Create the properties and assign it to the elements

1. In the **Model** browser, right-click and select *Create* > *Property* from the context menu.

2. In the **Create Property** dialog:

    - For **Name**, enter `shell`.

    - Set **Type** to *2d*.

    - Set **Card image** to *SHELLPHP*.

    - Select a color for the property.

    - Click the *Material* tab.

    - Select the *Assign material* checkbox.

    - Set **Name** to `steel`.

    - Click *Create*.

3. In the **SHELLPHP** card image, select the **THICK** checkbox.

4. In the **THICK** field, enter `1.5`.

5. Click *return*. A new 2D property has been created.

6. In the **Model** browser, **Component** folder, right-click on *shell* and select *Assign* from the context menu.

7. In the **Assign to Component(s)** dialog:

    - Set **Property** to *shell*.

    - Set **Material** to *steel*.

    - Click *OK*. The property is now assigned to the elements.

8. Create a property named **solid**, that is of type **3D**, and has a **SOLIDMAT** card image. You can skip editing the card image for this property, because this 3D property is a dummy property defined to assign material and an ATT number to the elements.

9. Assign the elements of the component *solid* to the new 3D property *solid*.

10. Check that the properties are correctly assigned to the elements using the *By Prop* icon ![By Prop icon] . The elements should have the same color as the properties.

Altair
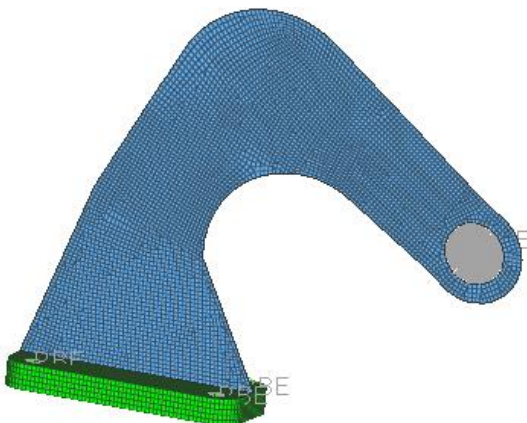
Step 6: Create the load collectors

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 `spc`.

    • Select a color for the load collector.

    • Click *Create*.

3. Create a second load collector named **force**.

Step 7: Create the constraints

1. In the **Model** browser, **Load Collector** folder, right-click on *spc* and select *Make Current* from the context menu. This sets the **spc** load collector as the current load collector.

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

3. Go to the **create** subpanel.

4. Using the **nodes** selector, select the master nodes of the rigid elements, which are defined in the holes of the component **solid**.
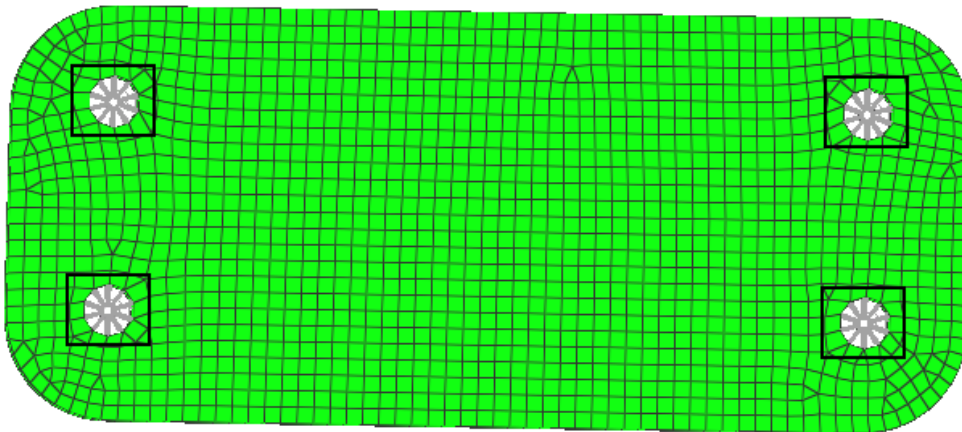
5. Constrain dof1, dof2, dof3, dof4, dof5 and dof6 by checking all of their checkboxes. Set all of them to a value of `0.0`

    DOFs with a check will be constrained, while DOFs without a check will be free.

    DOFs 1, 2 and 3 are x, y, and z translational degrees of freedom.

    DOFs 4, 5 and 6 are x, y an z rotational degrees of freedom.

6. Click *create* to apply these constraints to the selected nodes.

7. Click *return* to close the panel.

Step 8: Create the forces

1. In the **Model** browser, **Load Collector** folder, right-click on *force* and select *Make Current* from the context menu. This sets the **force** load collector as the current load collector.

2. Open the **Forces** panel by clicking *BCs > Create > Forces* from the menu bar.

3. Go to the **create** subpanel.

4. Using the **nodes** selector, select the master nodes of the rigid element.
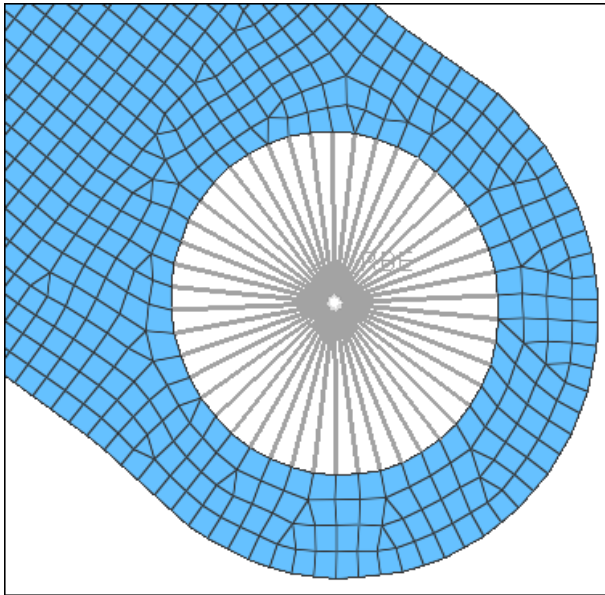
5. Set the **local system** / **global system** toggle to *global system*.

6. Set the **vector definition** switch to *constant vector*.

7. In the **magnitude =** field, enter `250`.

8. Set the orientation selector to *z-axis*.

9. Click *create*.

10. Click *return*.

Step 9: Create a SAMCEF database

1. Open the **Load Steps** panel by clicking *Setup > Create > Load Steps* from the menu bar.

2. In the **name** field, enter `z-force`.

3. Using the *loadcols* selector, select the load collectors *force* and *spc.*

4. Click *create*.

5. Click *return* to close the panel.

△ Altair

Step 10: Export the file

1. From the menu bar, click *File* > *Export* > *Solver Deck*. The **Export - Solver Deck** tab opens.

2. In the **File** field, enter the name of the file as `hook.dat`.

3. Set **template** to **Samcef**.

4. Click *Export*.