FINITE ELEMENT ANALYSIS OF A PLANAR TRUSS

Instructor: Professor James Sherwood

Revised: Michael Schraiber, Dimitri Soteropoulos, Sanjay Nainani

Programs Utilized: HyperMesh Desktop v2017.2, OptiStruct, HyperView

This tutorial explains how to build a planar truss. The pre-processing program used is Hypermesh, and Optistruct is used for the analysis.

Figure 1. Truss Dimensions and Boundary Conditions

The following exercises are included:

- Setting up the problem in HyperMesh
- Applying Loads and Boundary Conditions
- Submitting the job
- Viewing the results

Step 1: Launch HyperMesh and set the Optistruct User Profile

1. Launch Hypermesh. The User Profiles dialog appears.
2. Select Optistruct and click OK. This loads the User Profile. It includes the appropriate template, macro menu, and import reader, paring down the functionality of HyperMesh to what is relevant for generating models for Optistruct.
Step 2: Create basic geometry.

1. Right-click in the Model browser and select Create > Component.

2. Set the value for Name to “Truss1,” select a color, and click Create.

3. Create the nodes for the truss by selecting Geometry > Create > Nodes > XYZ

4. Create each node from the following table:

5. Change the view to the XY plane from the standard toolbar.
<table>
<thead>
<tr>
<th>Node</th>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>15</td>
<td>10</td>
</tr>
<tr>
<td>6</td>
<td>30</td>
<td>20</td>
</tr>
<tr>
<td>7</td>
<td>45</td>
<td>10</td>
</tr>
</tbody>
</table>

Table 1. Node coordinates

Figure 4. View of Seven Nodes

6. Create the lines for the truss by selecting **Geometry > Create > Lines > Linear Nodes**.
7. Create the truss element-by-element, selecting two nodes at a time and selecting **create**. The final geometry should resemble the structure below.
8. Click **create**.
9. Delete all temporary nodes by selecting **Geometry > Delete > Nodes**; click the **clear all** button. Click **return** to get back to the main menu
Step 3: Create Steel material.

1. Right-click in the Model browser and select Create > Material. This will create a Material in the Model Browser.
2. Enter name Steel for it.
3. The card edit panel for the material will appear in the bottom

![Figure 6: Material](image)

4. Make sure Card Image is set to MAT 1
5. Right-click on the Material and choose the Card Edit option.
6. Set the value for \( E \) (Young’s Modulus) to \( 3.0 \times 10^7 \) psi.
7. Set the value for \( NU \) (Poisson’s Ratio) to 0.3. Accept defaults for the rest and click return.

![Figure 7. Edit Material Card Edit](image)

**Step 4: Create the section properties for the bar elements (OptiStruct CROD by using HyperBeam)**

1. To open the HyperBeam panel, click Properties > Hyperbeam from the menu bar.
2. Go to the standard section subpanel.
3. Set the standard section library to OPTISTRUCT.
4. Set the standard section type to Rod.

![Figure 8. HyperBeam Panel](image)

5. Click create. HyperMesh invokes the HyperBeam module.

   Note: The solid, green circle represents the cross section. Under the local coordinate system you should see the number 10.000, which is the radius of the cross-section.

6. Under Parameter Definition, click the Value field next to Radius \( (r) \) and change the value from 10.0 to 2.0. HyperMesh updates the value in the Data pane to reflect the circle’s new radius.
7. In the Model browser, right-click on the section circle_section.1 and select Rename from the context sensitive menu.
8. In the editable field, rename the section “XC1.”
9. To close the HyperBeam module and return to your HyperMesh session, click **File > Exit** from the menu bar.

   To return to the main menu, click **return**

   **Step 5: Create cross-sectional property.**

   1. Right-click in the **Model** browser and select **Create > Property**.
   2. This will create a property name it **Rod_property**.
   3. In the entity editor:
Enter the above shown values.

4. Set the value for Card Image to PROD.

5. Select the checkbox for Beamsection and set its value to “XC1.”

6. Click the Material and select Steel.

7. The Area, A, and Moment of Inertia, J, should automatically calculate to the values below.

8. Right-click on the “Truss1” component from the model menu and select Assign from the pulldown menu.

9. Set the value for Property to “Rod_Property” and click Ok.

Step 6: Create a 1-D Line Mesh for the truss.

1. From the main menu, select Mesh > Create > Line Mesh.
2. Set the element selector to lines; click to select and choose all to select all 11 lines.
3. Ensure that segment is whole line property is active
4. Ensure that element config: is set to “rod.”
5. Set element size= to 20.0
6. Set the value for property to “Rod_Property.”
7. Click mesh.
8. Click return twice to return to the main menu.
9. To make the mesh elements CROD elements, select Mesh > Create > 1D Elements > Rods.
10. Click the update radio button, and select all elements.
11. Set the property= value to “Rod_Property” and set the elem types= value to “CROD.”
12. (Optional) To create a better visual representation of the truss, change the 1D Element Representation from the visualization panel to “1D Detailed Element Representation.”

Step 7: Create a load collector for the applied force.

1. In the model browser, right-click and select Create > Load Collectors.
2. In the window that opens, set the value for Name to “Force1.”
3. From the main menu, select BCs > Create > Forces.
4. Make sure you are in the create subpanel.
5. Set uniform size= to 10.0.
6. Set magnitude= to 1000.
7. Set the orientation to x-axis.
8. Make sure load types= is set to FORCE.
9. Make sure the nodes selector is active, and select the node at the top of the truss.
10. Click create, then return.

Step 8: Create a load collector for the roller and pin constraints

1. In the model browser, right-click and select Create > Load Collectors.
2. In the window that opens, set the value for Name to “Constraints.”
3. From the main menu, select BCs > Create > Constraints.
4. Ensure that load types= is set to SPC.
5. Make sure the nodes selector is active, and select the bottom-left node of the truss; this will be the roller.
6. In the DOF side panel, only check dof2 and dof3.
7. Click create.
8. Make sure the nodes selector is active, and select the bottom-right node of the truss; this will be the pinned connection.
9. In the DOF side panel, only check dof1, dof2, and dof3.
10. Click create.
11. Click return to return to the main menu.

Step 9: Create a Load Step to prepare for analysis.

1. Select Setup > Create > LoadSteps to enter the LoadStep creation panel.
2. In the LoadStep panel, set the value for Name to “Static Truss Analysis.”
3. Set the value for Analysis Type to Linear Static.
4. Set the value for SPC to “Constraints.”
5. Similarly, set the value for LOAD to “Force1.” The panel should look like the one below:

![Figure 12. Creating the Static Truss Analysis LoadStep](image1.png)

6. Click Create, then Return to get back to the Main Menu.

Step 9: Construct an OptiStruct Analysis

1. From the main panel, select Analysis.
2. Choose OptiStruct from the Analysis panel.
3. Under export options: choose all.
5. Under memory options: choose memory default.
6. Select a destination for the file in the input box by selecting save.
7. Click OptiStruct to perform the analysis.

![Figure 13. Fields in the OptiStruct Panel](image2.png)

Step 10: View a Contour Plot of Stresses
8. Choose **Results** from the window that opens to view the results of the job.

![The HyperWorks Solver View window](image1.png)

**Figure 14.** The HyperWorks Solver View window

9. To view the axial stress in the model, click the **Contour** icon in the visualization panel.
10. Ensure that **Result type** is set to **Element Stresses (1D)** (s) and select **CROS Axial Stress** from the drop down menu below. Click **Apply** to view the results:

![The Axial Stress plot for the given subcase](image2.png)

**Figure 15.** The Axial Stress plot for the given subcase
11. To view the deformed shape of the model, click the **Deformed** icon in the visualization panel.

12. Change the **Scale** type to **Model Percent**; in the **Value** field, enter **10.0**.

13. In the visualization panel, drag the **Current Time** selector to the end to view the final deformed shape of the truss:

![Figure 16. Deformed plot of the truss](image)

14. To view the individual stresses in each element, select the **Query** toolbar icon.

15. Change the selector button to **Elements** and select the three elements shown below to view their stress values. Ensure that they match the values shown below.
Step 10 (Optional): Save the .mvw file

Select File > Save to save the results of the OptiStruct analysis to be viewed at a later time.