Planar Truss

Find the nodal displacement developed in the planar truss when a vertically downward load of 1,000 N is applied at node 4.

Solution by ANSYS Workbench

Prepare an Analysis Model

1. Start the application package ANSYS Workbench.
2. In the left toolbox, select Static Structure from the Analysis Systems by double clicking. Name it as Planar Truss.
Create a Geometric Model

1. Doubly click A3 Geometry to enter the Design Modeler.
2. In the dialog box “Select desired length unit”, pick up Centimeter and press OK.

3. To create the base points, select the icon Point in the top toolbox.
4. In the left Details View, import a text file containing the coordinates of four points as listed below.

```
1 1 0 0 0
1 2 100 0 0
1 3 50 50 0
1 4 200 100 0
```

5. Press Generate to create a group of four points as shown below.

6. In the pull-down menu, press Concept → Lines From Points. Pick up points 1 and 3 and then press Generate to create a line body.

7. Pick up points 3 and 2 to create another line body. In the Details of Line 2, change the Operation from “Add Material” to “Add Frozen”.

```
Details of Line2

Lines From Points: Line2
Point Segments: 1
Operation: Add Frozen
```
8. Similarly, pick up points 3 and 4; points 2 and 4, respectively, to create the other two line bodies. It looks like the one below.

9. Press Concept → Cross Section → Rectangular. In the Details of Rect1, modify the value of B as 2 cm so that the cross sectional area is 2 cm².

10. Create another cross section Rect2 with an area of 1 cm².

11. In the model tree, expand the 4 line bodies. In the Details, let the first two bodies have cross section Rect1 and the last two bodies have Rect2.

12. In the pull-down menu, press View and check Cross Section Solids.
13. In the model tree, select the first Line Body, press the right button and pick Rename to change it as “Line Body1”.

14. Similarly, rename the other 3 Line Bodies as “Line Body2”, “Line Body3” and “Line Body4”.
15. Return to the Project Schematic.

Plug-In Line Bodies

1. In the Project Schematic, select A3 Geometry, press the right button and pick Properties.
2. In the Basic Geometry Options, check Line Bodies.
Assign Material Properties

1. In the Project Schematic, select A2 Engineering Data, press the right button and pick Edit.

2. In the Engineering Data, duplicate the default “Structural Steel” and then rename the copy as “Metal”.

3. Change the value of Young’s modulus from \(2 \times 10^{11}\) to \(2 \times 10^{10}\) Pa \(= 2 \times 10^{10} \text{ N/m}^2 = 2 \times 10^6 \text{ N/cm}^2\).

4. In the top toolbox, press Return to Project.
**Develop Model**

1. In the Project Schematic, doubly click A4 Model, or press the right button and pick Edit… to enter the Mechanical analysis.
2. In the Model tree, expand Geometry; select the 4 Line Bodies.

   ![Model Tree Diagram]

   **Material**
   - Assignment: Metal
   - Nonlinear Effects: Yes
   - Thermal Stain Effects: Yes

3. In the bottom Details, change the Material Assignment from “Structural Steel” to “Metal”.
4. Select Mesh, press the right button and pick Generate Mesh.

**Set up the Connections**

1. In the Model tree, select the Connections.
2. In the top toolbox, select the Revolute from the Body-Body connection.

   ![Body-Body Connection]

3. Set the selecting filter of geometry to vertex, select the vertexes connecting Bodies 1 and 2.
4. Similarly, set up the connections between Bodies 1 and 3, Bodies 2 and 3, Bodies 3 and 4 as Revolute.

**Set up the Constraints and Loads**

1. In the Model tree, select Static Structure.
2. In the top toolbox, press Supports and pick up Simply Supported. Select the lower left node and press Apply in the Details.
3. Press Supports again, and pick up Simply Supported for the lower right node.
5. Select the top right node; press Apply in the Details for Geometry Selection. Define by Components and set Y Component as -1000 N.
Set up Solution Items

1. In the Model tree, select Solution to set up the desired quantities.
2. In the top toolbox, press Deformation and pick Total.
3. Press Deformation and pick Directional twice. For the latter, set the Orientation as Y Axis.
4. Press Tools and pick Beam Tool to obtain stresses.
5. To find out the force reaction, press \text{Probe} and pick \text{Force Reaction}.

6. In the Details, set the Boundary Condition as the Simply Supported.

7. Similarly, press \text{Probe} and pick \text{Force Reaction} again. Set the Boundary Condition as the Simply Supported 2.

\textbf{Solution}

1. In the top toolbox, press \text{Solve}. After a few seconds, the solving process is completed.

\textbf{Review the Results}

1. Select the preset \text{T}otal \text{D}eformation to view the deformed body.
2. Select the preset Directional Deformation in X and Y axes.

A: Planar Truss
Directional Deformation
Type: Directional Deformation (X Axis)
Unit: mm
Global Coordinate System
Time: 1

```
3.485 Max
3.0977
2.7104
2.323
1.9357
1.5483
1.161
0.77365
0.3863
0.0010392 Min
```

3. Select Direct Stress to see the trusses in tension or compression.

A: Planar Truss
Direct Stress
Type: Direct Stress
Unit: MPa
Time: 1

```
15.811 Max
1.698
1.5837
1.4699
-0.64399
-4.7578
-8.8717
-12.986
-17.099
-21.213 Min
```
4. Select Force Reaction to find out the reactions at the supports.