Course in Ansys

Example 0150

Example – Truss 2D

Objective:
Compute the maximum deflection

Tasks:
Display the deflection figure?

Topics:
Topics: Start of analysis, Element type, 
Real constants, Material, modelling, element size for beam models, saving/restoring

\[ E = 210000 \text{N/mm}^2 \]

\[ \nu = 0.3 \]

\[ L = 100 \text{mm} \]

\[ H = 120 \text{mm} \]

\[ a = b = 20 \text{mm} \]

\[ c = d = 10 \text{mm} \]

\[ F = 100 \text{N} \]
Step by Step guide for Ansys 13

1. Preprocessor > Modeling > Create > Keypoints In Active CS

3 Points (click apply for 1 and 2 and ok for 3)

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>100</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>120</td>
<td>0</td>
</tr>
</tbody>
</table>
2. Preprocessor > Modeling > Create > Lines > Lines>Straight Line

Click on point 1, ok, point 2, ok

Create lines between points 1-2; points 2-3;

Alternatively type the point numbers into the popup box and press enter to create lines

3. Preprocessor>Sections>Beam>Common Sections

Section 1: B=20 H=20

Section 2: B=10 H=10
4. **Preprocessor>Materials>Material Props>Material Model**  
   Inside the resulting window
   
   **Structural>Linear>Elastic>Isotropic**
   
   \[ E = 210000 \]
   
   \[ PRXY = 0.3 \]

5. Close popup window
6. Preprocessor>Element Type>Add/Edit/Delete

Click Add

Select Beam> 2 node 188

Click OK, Click Close
Select the line between points 1 and 2 (horizontal line) > Click OK

Under Element Type select 1

Select the Line between points 2 and 3 (diagonal line) > click OK

Under Element Type change to 2
8. Preprocessor>Meshing>Mesh>lines
   a. Pick All

9. Solution > Analysis Type> New Analysis
   a. Select static > click OK
10. Solution > Define Loads > Apply > Structural > Displacement > On Keypoints
   a. Select keypoint 3 (or type 3 into the box) > click ok

b. Select All DOF and type in 0 for Value > click OK
11. Solution > Define Loads > Apply> Structural> Displacement> On Keypoints
   a. Select Keypoint 1 > click OK

   ![Image of ANSYS interface showing displacement on keypoints]

   b. In the next box select UX and UZ and for Value select 0 (this creates a roller fixture which allows movement in the Y direction ONLY) > Click OK

12. Solution > Define Loads > Apply> Structural> Force/Moment> On Keypoints
a. Select keypoint 2 (or type the number 2) > OK

b. Select FY for direction, Constant Value, and enter -100 for the VALUE box (the force is directed down, hence the negative)> click OK
13. Solution > Solve > Current LS

a. Click OK
14. General Postproc>Plot Results>Deformed Shape
   a. Select Def + undeformed> OK

15. General Postproc>Plot Results>Contour plot>Nodal Solu
   a. In Popup window Nodal Solution>Stress>con Mises stress
   b. Click OK