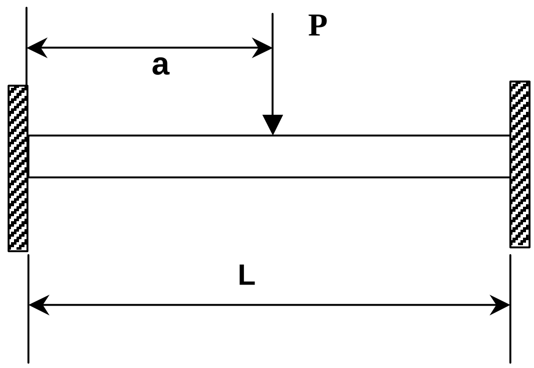
TUTORIAL 3

<u>OBJECTIVE</u>:

Use SolidWorks/COSMOS to model the statically indeterminate dual cantilevered beam shown below. The beam is exposed to a point load of 300 lb..

Material is steel (E = 30E 6 lb/in2)



thickness= .75"
height = .75

Let L = 12 inches
Let a = 8 inches
Let P = 300 lb.

Find the <u>maximum deflection</u> of the beam using the analytical expression.

Compare your analytical findings with findings determine through FEA analysis using COSMOS.

Analytical expression for maximum deflection is given below.

$$d_{\max} = \frac{-2F(L-a)^2 a^3}{3EI(L+2a)^2}$$

GETTING STARTED:

1. **Open a new part file as described in tutorial 1.**

2. **Change the part units to English.**
   - Left click on **Tools, Options**
   - Click on the **Document Properties** tab
   - Select **Units** (Change linear untits to inches and leave others as default)

SKETCH THE BEAM:

The beam is simple and very easy to sketch. Basically, we will just draw an end view of the beam in the sketching window (see Figure 1), and then we will extrude the beam to its full length (see Figure 2).

1. **Draw the face of the beam**
   - Left click on the **Sketch Button** (see Figure 1) located on the toolbar
   - Left click on the **Rectangle tool** (see Figure 1)
   - Left click on any point on the screen and continue to hold down the button
   - Drag the cursor to draw a rectangle (The size does not matter right now)

2. **Dimension the square**
   - Click the **Dimension Icon** (see Figure 1)
   - Click on any side of the square (a dimension leader and text should be displayed)
   - Drag the dimension line to the outside of the box
   - Repeat this step for one of the sides perpendicular to the first side

3. **Change the dimensions**
   - Select the cursor (select) icon (see Figure 1)
   - Double click on one of the dimensions (A box should appear)
   - Enter the correct value for the dimension (.75)
   - Click OK (The screen should now look like figure 1

4. **Extrude the beam**
   - Select the **Extrude Boss/Base** icon (see Figure 1)
   - Select Blind for the extrusion type
   - Change the depth to 12 inches
   - Click Ok

5. **The basic beam should now be complete (see Figure 2)**

COSMOS does not allow us to apply an actual point force as is drawn on the basic model. This is because a true point force would be distributed over an area of zero, leading to infinite stress. The solution to this is to place a distributed load over a very small protrusion that we will create on the surfaceof the beam.

CREATE THE STRIP

Refer to figure 3 and figure 4 as you create the strip.

1. **Selecting the sketching surface**
    - Select the top surface of the beam by left clicking on the surface (the surface should turn green)
    - Click the **Sketch** icon (see Figure 1)
    - Click the **Normal To** view (see Figure 1) on the views toolbar
    - Rotate the beam using the arrow keys while holding <alt>.
    - The beam should now be oriented as shown in Figure 3.

2. **Drawing the strip**
    - Select the **Rectangle** Tool (see Figure 1)
    - Draw the rectangle on the beam as shown in Figure 3 (Placement does not have to be exact)

3. **Define the location of the strip on the beam**
    - Select the **Dimension** Tool
    - Select the right top corner of the strip
    - Select the right top corner of the beam (a dimension line should appear)
    - Place the dimension to the right of the beam
    - Select the **cursor** icon (select) (see Figure 1)
    - Double click on the Dimension
    - Enter the Correct Value (3.95) (see Figure 3)
    - Click OK

4. **Define the thickness of the strip**
    - Select the **Dimension** Tool
    - Select the right side of the rectangle previously drawn (a dimension line should appear)
    - Place the dimension to the right of the beam
    - Select the cursor icon (select) (see Figure 1)
    - Double click on the Dimension
    - Enter the Correct Value (0.1) (see Figure 3)
    - Click OK

5. **Extrude the strip**
   - Click on the **Extrude Boss/Base** icon (see Figure 1)
   - Select Blind for the extrusion type
   - Enter 0.01 inches for the height
   - Click Ok

6. **Change the color of the strip**
   - Right click on the face of the strip
   - Choose **Base Properties**
   - Choose **Change Color**
   - Select a color of your choice

7. **The model is now complete (see Figure 4.)**

<u>SET UP THE FINITE ELEMENT ANALYSIS USING COSMOS</u>:

Since we want this beam to act as a statically indeterminate dual cantilevered beam, we must constrain each end to be held in place. Refer to Figure 5.

1. **Create the Study**
   - Click on **FEM** on the Top Menu
   - Select **Study** (a pop up box should appear)
   - Click **Add** from the pop up menu
   - Give the study a name (deflection)
   - Click OK

2. **Choose the Material**
   - Click on **FEM** on the Top Menu
   - Select **Materials** (a pop up box should appear)
   - Choose Steel Allow (default setting)
   - Click OK

3. **Constrain the beam**
   - Select the **cursor** (select) icon (see Figure 1)
   - Select one end of the beam (it should turn green)
   - Click on **FEM** on the Top Menu
   - Select **Insert**
   - Select **Restraints** (a pop up box should appear)
   - Choose **Fixed** in the type box
   - Click Ok
   - Repeat for the other end of the beam

4. **Apply the Force to the Beam**
   - Select the **Zoom Window** function (see Figure 1)
   - Zoom in on the strip (draw a box around it with the mouse)
   - Select **FEM** from the Top Menu
   - Select **Insert**
   - Select **Force** (A pop up box should appear)
   - Check mark the box that says **Normal To**
   - Enter the value of the Force (-300 lb)
   - Click OK

5. **Create the Mesh**
   - Select **FEM** from the Top Menu
   - Select **Mesh**
   - Select **Create** (a pop up box should appear)
   - Click OK (Different mesh sizes can be experimented with later)
   - The screen should now look like Figure 5.

<u>ANALYZE THE BEAM</u>

1. **Analyze**
   - Select **FEM** from the Top Menu
   - Select **Analyze** (a pop up box should appear)
   - Click OK (COSMOS will take a few moments to run the analysis.)

2. **Plot the Results**
   - Select **FEM** from the Top Menu
   - Select **Plot Results**
   - Select **Displacement** (a pop up box should appear)
   - Select inches as the displacement units and pounds as the reaction force units
   - Click Ok
   - The Finite element model should now be displayed in color that describes the relative displacement (see Figure 6)

3. **List Results**
   - Select **FEM** from the Top Menu
   - Select **List Results**
   - Select **Displacement** (a pop up box should appear)
   - Choose results within 5%
   - Click OK
   - The results will now be listed by node from greatest to least displacement

CHECKING THE FEA SOLUTION:

As a check on our work, the equation below can also be used to find an analytical solution for this simple beam problem. If we solve for $d_{\max}$ using this equation, the answer should closely match the maximum displacement determined using COSMOS FEA.

$$d_{\max} = \frac{-2F(L-a)^2 a^3}{3EI(L+2a)^2}$$

$$d_{\max} = \frac{-2*300lb.*(12in.-8in.)^2*(8in.)^3}{3*(30*10^6 Psi)*1/12*.75in.*(0.75in.)^3*(12in.+2*8in.)^2}$$

$$d_{\max} = -.00264in.$$

Check this value with the maximum value that was obtained using COSMOS FEA

FURTHER ANALYSIS:

Stresses and strains can also be analyzed for this beam as was done for the other tutorials.

4. **Reaction Forces**
   - Select **FEM** from the Top Menu
   - Select Result Tools
   - Select Reaction Force (a pop up box should appear)
   - Select either end of the beam
   - Click Update (Note the Reaction Force at that end)
   - Repeat for the other end of the beam

As expected, if the y-direction forces are added for each face, they will equal the load total and the moments on the beam will also sum to zero.

Plots of the stresses and displacements can be printed out by right clicking on the plot icon on the left FEA tree, and then selecting print.

Pictures