## Tutorial 1

## **OBJECTIVE**:

Use SolidWorks/COSMOS to model a **flat plate with a hole through the center** in order to determine maximum stress.

Step-by-Step process to drawing and testing a plate in COSMOSWorks.

- 1. Begin by opening COSMOS Works 5.0 from the Start Menu
- 2. Click on the **New Drawing** Icon from the tool bar.
- 3. The New SolidWorks Document window will appear. From that window select the icon labeled **Part**. Then click **OK**.
- 4. Change units to English. (Reference figure 1.1)
  - Go to the **Tools/Options...** menu.
  - Click on the **Document Properties** tab
  - Select Units.
  - Change the Linear Units to Inches.
  - Click **OK**.



Figure 1.1

- 5. Begin to draw the object.
  - Select **Plane 1**, at the left of the screen, by clicking on it.
  - Click on the **Sketch** icon located on the right-hand side of the screen. (See figure 1.1 for icon locations).
- 6. Draw a rectangle
  - Click on the **Rectangle** button on the left center of the screen (See Figure 1.2).
  - In the drawing screen click and hold the left mouse button at the origin and drag to create a rectangle. Release the mouse button.





- 7. Dimension the rectangle
  - Click the **Dimension** icon (See figure 1.2).
  - Click on a side of the rectangle once then click at the location where the dimensions is to be placed, COSMOS Works automatically draws the dimension.
  - Repeat this for the other side of the rectangle. Note: the sketch is now fully defined.
- 8. Modify Dimensions
  - Deselect the **Dimension** icon

- Select the **Select** icon (Figure 1.2).
- Double click on the side dimension. This allows you to modify the dimension value.
- Enter **5**.
- Double click on the top dimension. Change the top dimension to 2.5.
- Press the **F** key to fit the drawing to the screen; your drawing should now resemble Figure 1.3.
- 9. Extrude the Base Feature
  - Click the **Extruded Boss/Base** icon (See Figure 1.3). The Extrude Feature dialog box will appear (See Figure 1.4).
  - Enter **0.125** as the depth.
  - Click **OK**. The Rectangle has now become a 3D Box.





10. Draw the Circle.

- Click the front face of the box. It should now be highlighted in green.
- Click the **Normal To** icon (See Fig 1.4) on the view toolbar.
- Click the **Sketch** icon.
- Click the **Circle** icon (See Fig 1.3).
- Click and drag to draw a circle in the middle of the rectangle face.





11. Dimension the Circle.

- Click the **Dimension** icon.
- Click on the outer edge of the circle
- Click to place the dimension for the circle diameter.
- Click on the center of the circle and release, click on upper edge of rectangle to dimension the vertical placement of the circle within the rectangle.
- Click again to place the dimension.
- Click on the center of the circle and release, click on side edge of rectangle to dimension the horizontal placement of the circle within the rectangle.
- Click again to place the dimension.
- Click the **Select** icon and change the dimension of the circle to **0.5**", and modify the dimensions so that the circle is centered within the rectangle. The center to top distance should be **2.5**", and the center to side distance should be **1.25**".
- 12. Cut a hole in the plate.
  - Click the **Select** Icon.
  - Click on the edge of the circle.

- Click the **Extruded Cut** icon (See Fig 1.5). The Extruded Cut Feature dialog box will appear.
- Select **Through All** from the **Type** pull down menu.
- Click **OK**. Notice that a hole has been cut in the rectangular box.

Modeling of the solid is now complete.

Next we will set up and run the Finite Element Analysis program to analyze the stresses on the part.

- 13. Define a study. A study is a set of loading conditions. Studies are utilized to analyze one part under different circumstances.
  - From the **FEM** menu select **Study...** The Study dialog box will appear.
  - Select the Add... button. Enter new study name.
  - Select **Static** for analysis type.
  - At this point the mesh type can be selected as "Solid" or "Shell." Select **Solid** for this study.
  - Click **OK** to close the New Study dialog box.
  - Click **OK** to close the Study dialog box.
- 14. Specify a material type. COSMOS Works 5.0 has a wide variety of different materials that can be used for a part.
  - From the **FEM** menu select **Material...** The Material dialog box will appear.
  - For this study use Alloy Steel, which is already selected.
  - Click **OK** to close the dialog box. Note: If other materials were to be used this is where the material would be changed.

			Bottom View
SelidWorks Educational License - Inst P	ructional Use Daly - [Part1] Viritize Halp X ==> - 🔋 🏞 🄝 🎯 📽 🌹 🗟 🤤	Top View	19 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
Part3 X Tutorial Part3	Funce     Type     C doply Force/Mameri     C doply Force/Mameri     C doply Nomel Force     C rough     C doply Nomel Force     C rough     Parce     Force     C rough     Force     C rough     Force     C rough     Force     C rough     C rough     Force     C rough     C rough<	Selected Environ	Isometric
			یا القطیمی
Brat Mr. Barbart	Marrison Witness Schursters Witness	end Gebel Weater - Miry	19 10 Charles Charles Charles Charles

Figure 1.6

- 15. Insert forces on the plate to be analyzed. Boundary conditions must be specified in order to analyze a part in COSMOSWorks (Reference Fig 1.6).
  - Click the **Top View** button to view the top of the plate.
  - Click the **top plane** of the plate.
  - From the **FEM** menu select **Insert/Force...**. The Force dialog box will appear.
  - Select Apply Normal Force from the section labeled "Type."
  - In the Distributions Section change units to English.
  - Enter a Force of -300
  - Click OK.

16. Insert restraints on the plate to be analyzed.

- Click the **Bottom View** button to view the bottom of the plate (See Fig 1.6).
- Click the **bottom plane** of the plate.
- From the **FEM** menu select **Insert/Restraints...**. The Restraint dialog box will appear.
- Select **Fixed** from the section labeled "Type."
- Click **OK**.

- 17. Click the **Isometric** button from the view toolbar. (See Fig 1.6)
- 18. Create Mesh of plate. A mesh is a conglomeration of the actual finite elements used to analyze a part.
  - From the **FEM** menu select **Mesh/Create...** The Mesh dialog box will appear. The mesh dialog box controls the "Fineness" of the mesh. A course mesh uses larger elements and a fine mesh uses smaller elements. Complex geometries require a fine mesh to properly simulate actual conditions.
  - The pre-selected mesh is sufficient for this model. Click **OK**.

19. Analyze the Part.

- From the **FEM** menu select **Analyze...**.
- Select the study name from the Analyze dialog box
- Click **OK**.

There are several methods to view the results. Plots of Stress, Strains and displacements can be viewed. Also numerical values can be listed.

- 20. View Stress Plots.
  - From the **FEM** menu select **Plot Results/Stress...**. The Stress Plot dialog box will appear.
  - Change the units to "psi."
  - Click **OK**. Notice the stress plot appears on the screen. Red colors indicate high stress areas (See Fig 1.7). (Note: similar plots can be generated for strain and displacement)



## Figure 1.7

21. View the list of results.

- From the **FEM** menu select **List Results/Stress...**. Then the List Stress dialog box will appear.
- Change the units to **psi**.
- Several different types of stresses can be viewed. **Von Mises** stresses are selected as default. For this exercise do not change the stress component.
- Click **OK**.
- The maximum value should be near 3120 psi.

Then next segment of this exercise will cut the plate into various dimensions and repeat analysis to see how it compares to the previous result.

22. Return to original drawing.

- Click the FEM tab on the bottom left of the screen. (See Fig. 1.8).
- Click the **FeatureManager Design Tree** tab on the bottom left of the screen (See fig 1.8). The original drawing should appear.
- View the front face of the plate so that the hole is visible. This can be done by clicking the front view icon.

23. Cut the part in half.

- Select the **front face** by clicking on it,
- Click the **Sketch Icon**.
- Click the **Line** icon.
- Draw a line directly across the center of the plate in the horizontal direction. Bee sure that the line is directly in the center of the plate. Note

as long as the mouse pointer is close to the center, the grid snap will automatically place the line in the center. The line will turn black if it is positioned correctly, and blue if it is not at the mid point of the plate.

- Click the **Extruded Cut** icon. The Extruded Cut dialog box will appear.
- Select type as **Through All**.
- Click **OK**. Notice the plate is now cut in half. (See Fig 1.8)



Figure 1.8

24. Under the **FEM** Menu create a **new study**.

25. Restrain the plate

- Place a -300 lb normal force on the uncut bottom edge. (Refer to step 15 for assistance).
- On each of the two top faces place a restraint. (Refer to step 16 to find the dialog box). Do not select fixed for these faces. In the restraints dialog box select **On Flat Face** as the type.
- Check the **Normal to Face** box below.
- Select the Top Right Edge by clicking on it (See Fig. 1.8). Note the cursor will change to an "edge" when the mouse is properly positioned to select the edge.
- Place a restraint on the edge (Refer to step 16 to find the dialog box).

- Choose Use Reference Plane of Axis as the type.
- Check the boxes next to Along Plane Dir 1 and Normal to Plane.

26. Create a new mesh for the half plate.

- Define the material as alloy steel from the **FEM** menu.
- Reanalyze the part. The new maximum Von Mises stress should be near 3055 psi.
- 27. Cut the plate once more, this time so that the plate is <sup>1</sup>/<sub>4</sub> of the original (See Fig 1.9). This cut is performed in the same manor as in step 24.



- 28. Under the **FEM** Menu create a **new study**. Also define material as alloy steel.
- 29. This part will require several restraints to produce the correct results. Figure 1.9 shows the part with the proper restraints. The following steps refer to the references made in Figure 1.9 in regard to restraint location.
- 30. First add the force to the Top of the plate. Place a -150 lb force on the Top surface as was done in step 16.
- 31. Restrain the bottom of the <sup>1</sup>/<sub>4</sub> plate.

- Select the Bottom surface of the plate.
- From the **FEM** menu select **Insert/Restraints...**. The restraints dialog box will appear as shown in Fig 1.10.
- Select Use Reference Plane or Axis as the Type.
- Check the box by Along Plane Dir 2: leaving the Displacement equal to 0.
- Click **OK**.

32. Restrain the side surface of the  $\frac{1}{4}$  plate.

- Select the Side of the plate. See Fig 1.10
- From the **FEM** menu select **Insert/Restraints...**. The restraints dialog box will appear as shown in Fig 1.10.
- Select Use Reference Plane or Axis as the Type.
- Check the box by **Along Plane Dir 1:** leaving the Displacement equal to 0.
- Click **OK**.

33. Restrain the Top Hole Edge of the <sup>1</sup>/<sub>4</sub> plate.

- Select the Top Hole Edge of the plate. (See Fig 1.10)
- From the **FEM** menu select **Insert/Restraints...**. The restraints dialog box will appear as shown in Fig 1.10.
- Select Use Reference Plane or Axis as the Type.
- Check the box by **Along Plane Dir 1:** and also check the box by **Normal to Plane:** in both leave the displacement equal to 0.
- Click **OK**.

34. Restrain the Bottom Hole Edge of the <sup>1</sup>/<sub>4</sub> plate.

- Select the Bottom Hole Edge of the plate. See Fig 1.10
- From the **FEM** menu select **Insert/Restraints...**. The restraints dialog box will appear as shown in Fig 1.10.
- Select Use Reference Plane or Axis as the Type.
- Check the box by **Along Plane Dir 1:** and also check the box by **Along Plane Dir 2:** in both leave the displacement equal to 0.
- Click **OK**.

- 35. Create Mesh of plate.
  - From the **FEM** menu select **Mesh/Create...** The Mesh dialog box will appear. The pre-selected mesh is sufficient for this model. Click **OK**.

36. Analyze the Part.

- From the **FEM** menu select **Analyze...**.
- Select the study name from the Analyze dialog box.
- Click **OK**. The maximum stress on this part should be near 3414 psi.

E FEA Amign L quarter n FEA Amign L quarter n		₩] <b>₽₩®₽₽₩®</b> ® + ₩]
Image: Second	Ford   Fand     Final   Face     Final   Face     On Flat Face   Face     Outplacement Lints:   mn     Outplacement   Face     Atomy plane Dia 2   mn     OK   Cancel     Halp   Face	
Ready		Salara Fort

Figure: 1.10