Introduction

This lab is designed to provide you with basic skills when using the 3D modeling program SolidWorks. You will learn how to build parts, assemblies and drawings. You will be given a physical model and instructed to construct a CAD model using SolidWorks. Students with prior experience with SolidWorks should plan on attending the SolidWorks – II seminar to be taught later this semester. We will also help the students create/develop models for their design projects, so you should bring their lab notebooks.

Overview of Program

- Command Manager
- Feature Manager
- Graphics Area
- Toolbars/Tabs

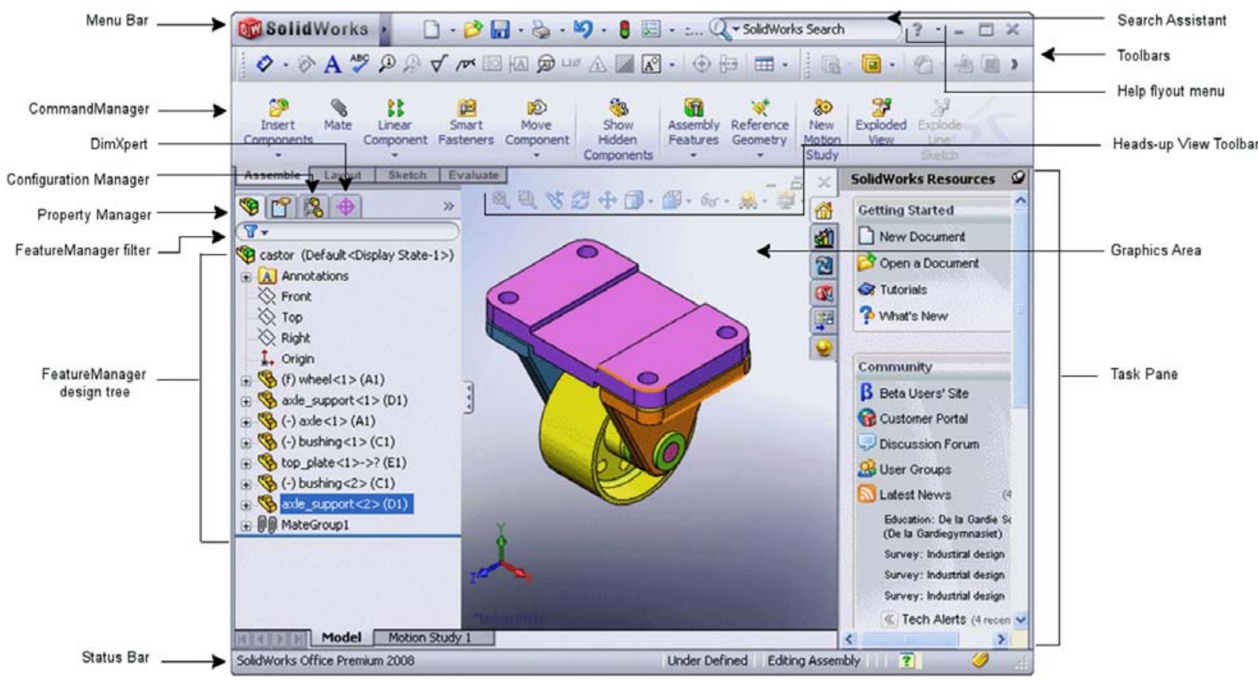

Figure 1 Work area in Solid Works

Tutorials!!! Students can do the tutorials on their own to expand their SolidWorks skills.

1. Click **SolidWorks Resources** icon in Task Pane (looks like a little house)
2. Click **Tutorials** icon in the Getting Started section
   Getting Started
   - Lesson 1 - Parts
   - Lesson 2 - Assemblies
   - Lesson 3 - Drawings

NOTE: Think…Draw in 2D, Extrude/Cut in 3D

a. Base

<u>Step 1</u>: Open "New Part"

1. Click **Part** square
2. Click **OK** button

<u>Step 2</u>: Extrude Boss/Base

1. Click **Extrude Boss/Base**
2. Make Sketch
    a. Select **TOP** Plane
    b. Draw rectangle
    a. Click on **Smart Dimension**
        i. Click on top of rectangle
        ii. Drag dimension to desired location
        iii. Assign dimension (length = 3.0" and width = 2.0")
    c. Click **Exit Sketch**
3. Extrude
    a. Assign thickness (1.00")
    b. Click *Switch Directions*
4. Click the **Green Arrow** icon in the *FeatureManager*

<u>Step 5</u>: Save Part

1. Click **Save** icon in the *Menu Bar*
2. Select folder to save part in
3. Name your project
4. Click **Save** button

<u>NOTE</u>: Demonstrate how to control/move the camera when viewing a part

- Rotate Part: Pressing Middle Mouse Button
- Zoom In/Out: Scroll Wheel
- View Orientation: Select pre-set views

<u>Step 3</u>: Extrude Cut(s)

1. Click **Extrude Cut**
2. Make Sketch
    a. Select **TOP** Plane
    b. Draw 2 circles (Center Point)
        i. Click to set position of center point
        ii. Click to set radius of circle (rough estimate)
    c. Dimension circles (r_1= 0.25" and r_2= 0.375")
    d. Position Circles
        i. Click on **Smart Dimension**
        ii. Click on center of circle
        iii. Click on edge of base

iv. Assign Dimension (d_{top}= 0.5" and d_{side}= 0.75")

e. Click **Exit Sketch**

3. Extrude Cut
   a. Assign depth
      i. Set to **Through All**
4. Click **Green Check Mark** in the *FeatureManager Area*

<u>Step 4</u>: Extrude Posts

1. Click **Extrude Boss/Base**
2. Make Sketch
   a. Select **TOP** Plane
   b. Draw 2 circles (Center Point)
      i. Click to set position of center point
      ii. Click to set radius of circle (rough estimate)
   c. Dimension circles (r_1= 0.25" and r_2= 0.372")
   d. Position Circles
      i. Click on **Smart Dimension**
      ii. Click on center of circle
      iii. Click on edge of base
      iv. Assign Dimension (d_{top}= 0.5" and d_{side}= 0.75")
   e. Click **Exit Sketch**
3. Extrude
   a. Assign thickness (0.5")
   b. Click ***Switch Directions***
4. Click the **Green Arrow** icon in the *FeatureManager*

<u>Step 5</u>: Save Part

1. Click **Save** icon in the *Menu Bar*
2. Select folder to save part in
3. Name your project
4. Click **Save** button

b. Fillets and Chamfers

<u>Step 1</u>: Create Fillets

1. Click **Fillets**
2. Select edges to Fillet (one long edge and one short edge)
3. Set radius to 0.1"
4. Select **Full Preview** icon
5. Click **Green Check Mark**

<u>Step 2</u>: Create Chamfers

1. Click on ***small arrow*** on **Fillet** button
2. Select edges to Chamfer
3. Set depth to 0.05"
4. Select ***"Full Preview" button***

5. Unselect ***"Tangent Propagation" button***
6. Click the **Green Arrow** icon in the *FeatureManager*

Step 3: Save Part
1. Click **Save** icon in the *Menu Bar*

c. Linear Patterns

Step 1: Create Feature to Pattern
1. Click **Extrude Cut**
2. Make Sketch
    a. Select **SIDE** Plane
    b. Draw 1 circle (Center Point)
    c. Dimension circles (r = 0.125")
    d. Position circle
        i. Click on **Smart Dimension**
        ii. Click on center of circle
        iii. Click on edge of base
        iv. Assign Dimension (d_{top}= 0.375" and d_{side}= 0.5")
    e. Click **Exit Sketch**

Step 2: Click **Linear Pattern**
1. Highlight first box in "Direction 1"
    a. Click top edge of base
    b. Set ***D1*** to 0.5
    c. Set ***# of Features*** to 5
2. Highlight first box in "Direction 2"
    a. Click side edge of base
    b. Set ***D2*** to 0.25
    c. Set ***# of Features*** to 2
3. Click the **Green Arrow** icon in the *FeatureManager*

Step 3: Save Part
1. Click **Save** icon in the *Menu Bar*

d. Assemblies

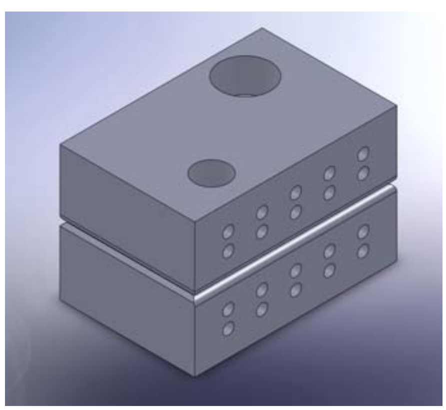

Figure 2. Assembly using two parts created in the above section

<u>Step 1</u>: Open new Assembly

1. Click **New Document** icon
2. Click **Assemblies** icon
3. Click **OK** button

<u>Step 2</u>: Insert Parts

1. Click "**Keep Visible**" Icon
2. Place 2 parts in the workspace (click twice)
3. Click the **Green Arrow** icon in the FeatureManager Area

<u>Step 3</u>: Mate Parts

1. Click **Mate** Icon
2. Create 3 Mates
    a. Mate #1
        i. Click the side of the Large Post on Part #1
        ii. Click the side of the Large Hole on Part #2
        iii. Click the **Flip Mate Alignment** Icon
        iv. Click the **Green Arrow** icon on pop-up toolbar.

        NOTE: Click on Part #2 and move out of view of Part #1
    b. Mate #2
        i. Click the side of the Small Post on Part #1
        ii. Click the side of the Small Hole on Part #2
        iii. Click the **Green Arrow** icon on pop-up toolbar.
    c. Mate #3
        i. Click on Top of Part #1
        ii. Click on Top of Part #2
        iii. Click the **Green Arrow** icon on pop-up toolbar.
3. Click the **Green Arrow** icon in the *FeatureManager*

<u>Step 4</u>: Save Assembly

1. Click **Save** icon in the *Menu Bar*
2. Select folder to save part in
3. Name your project
4. Click **Save** button

NOTE: Parts should line up without any errors. Linear patterns should line up.

d. Drawings

<u>Step 1</u>: Open a new Drawing

1. Click **New Document** icon
2. Click **Drawings** icon
3. Click **OK** button
4. Select **B-Landscape** from the scroll down menu in the *Sheet Format/Size* pop-up box
5. Click the OK button

<u>Step 2</u>: Choose part/assembly

1. Select part to be drawing Part/Assembly Insert box in the *PropertyManager*
2. Click the blue **Next Arrow** found at the top right corner of the *PropertyManager*

Step 3: Choose views to show

1. Select **Preview** found at the bottom of the Orientation box in the *PropertyManager* to display a preview in the Graphics Area
2. Select ***TOP** view icon from the *Standard Views* box
3. Move mouse cursor into the *Graphics Area* and place part in top/middle of the *Graphics Area*
4. Click the Right mouse button to place the **Top View**
5. Move the mouse cursor to the right and click to display the **Left View**
6. Move the mouse cursor to the left and click to display the **Right View**
7. Move the mouse cursor to the down and to the right to display the **Front View**
8. Click the **Green Arrow** icon in the *FeatureManager*

Step 4: Dimensioning

1. Dimension the modeled parts in same way as described in the **base** section above.

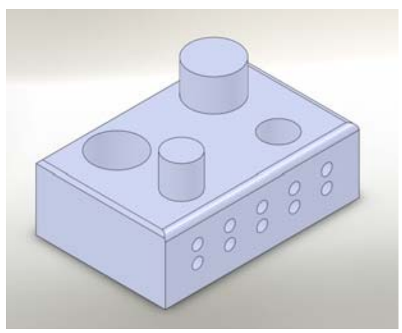

Figure 3. Part to be modeled