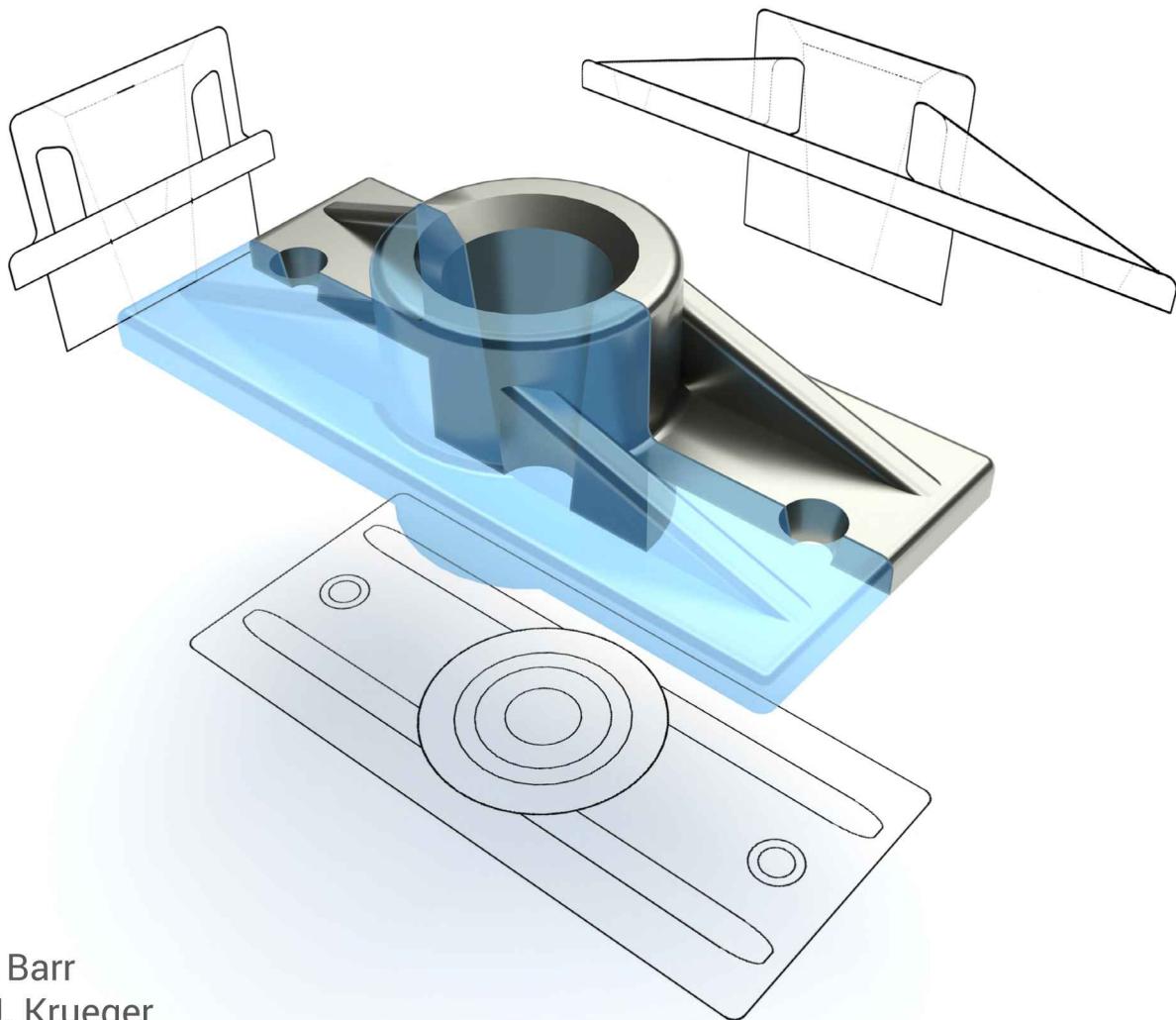


New
Fully revised
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Design Workbook Using **SOLIDWORKS® 2020**

Design, Detailing, Assembly & Analysis Basics



Ronald E. Barr
Thomas J. Krueger
Davor Juricic
Alejandro Reyes MSME, CSWE, CSWI



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DESIGN WORKBOOK

Using SOLIDWORKS 2020

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APPENDIX A – Drawing Sheet Template

NOTES:

Design Workbook Lab 1:

Basic 2D Sketching

INTRODUCTION TO SOLIDWORKS

SOLIDWORKS© is a 3D parametric solid modeling software used for mechanical design. Most designs start with a 2D sketch, where dimensions and geometric relations (or constraints) are applied between sketch elements, to fully define its geometry. The sketch is then turned into a 3D model using the Extrude, Revolve, Sweep or Loft command. A part is modeled by adding additional 2D sketch and 3D features, including bosses, cuts, fillets, chamfers, etc. Each new 3D feature either adds or removes material from the part, until the component's design is complete. This process is called Feature Based modeling.

If a part needs to be changed, its design can be modified at any time by editing any 2D sketch or 3D feature, where you can change dimensions, sketch geometry, etc.

In this first chapter, you will learn different 2D sketching techniques to create simple extruded or revolved parts, using four different exercises.

SOLIDWORKS TOOLBARS SETUP

When you first launch SOLIDWORKS the screen will appear as shown in **Figure 1-1**. To get started, touch the arrow next to SOLIDWORKS in the upper left corner to expand the menu bar, and click in the Push pin on the right to keep the menu visible. De-selecting it will cause the menu bar to automatically hide.

SOLIDWORKS has three different work environments: Part modeling, Assembly and Detail Drawing. Command toolbars and menu options are automatically activated. If needed, you are also able to change toolbars' visibility as needed to fit your individual needs.

To change a toolbar's visibility, select the menu View, Toolbars, and select/de-select the desired toolbar, or you can right click on any toolbar to display the same visibility options, as shown in **Figure 1-2**.

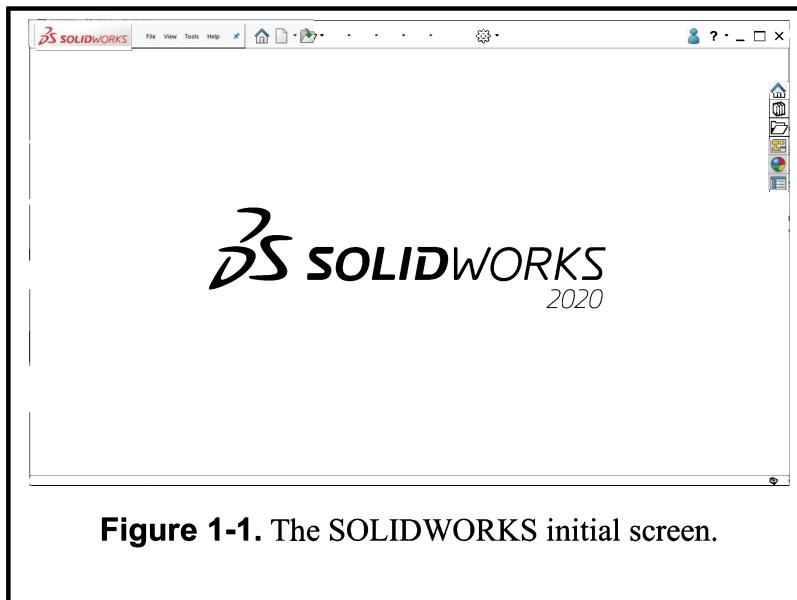


Figure 1-1. The SOLIDWORKS initial screen.

Design Workbook Using SOLIDWORKS 2020



Figure 1-2. Change toolbar visibility on or off.

STARTING A NEW PART

To start a new part in SOLIDWORKS, select the menu **File**, **New...** or click in the main toolbar in the **New** document command. The New Document dialog will be shown where you can start a new Part, Assembly, or Drawing. See **Figure 1-3**. If you cannot see the three options shown, click on the Novice button on the lower left corner.

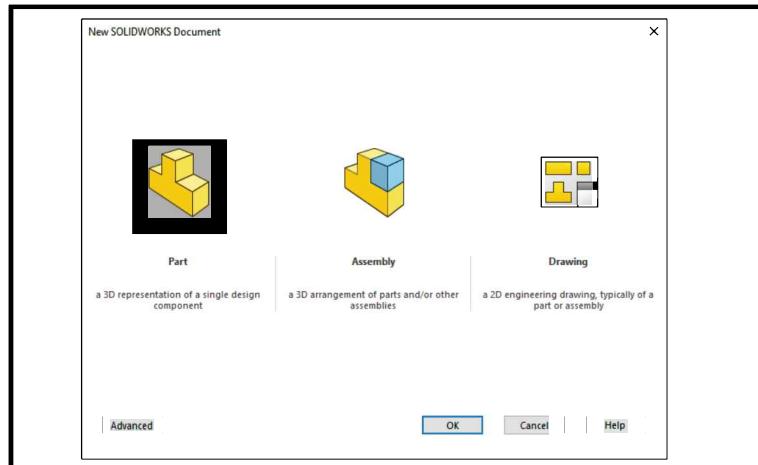


Figure 1-3. Starting a New Part in SOLIDWORKS.

THE SOLIDWORKS SCREEN LAYOUT

After selecting **Part**, click **OK**. A new part file is created, and the part modeling interface is shown. The **CommandManager** bar is enabled by default; it combines multiple toolbars in a single location using tabs for a more efficient use of screen space as shown in **Figure 1-4**.

If needed or desired, toolbars can be shown or hidden separate from the CommandManager as shown in **Figure 1-2**. A toolbar's visibility is remembered in each of the Part, Assembly and Drawing environments.

The center of the screen is the sketching area for your design work. Study **Figure 1-4** to become familiar with this screen layout. Each of the menus and toolbars will be described in the subsequent paragraphs.

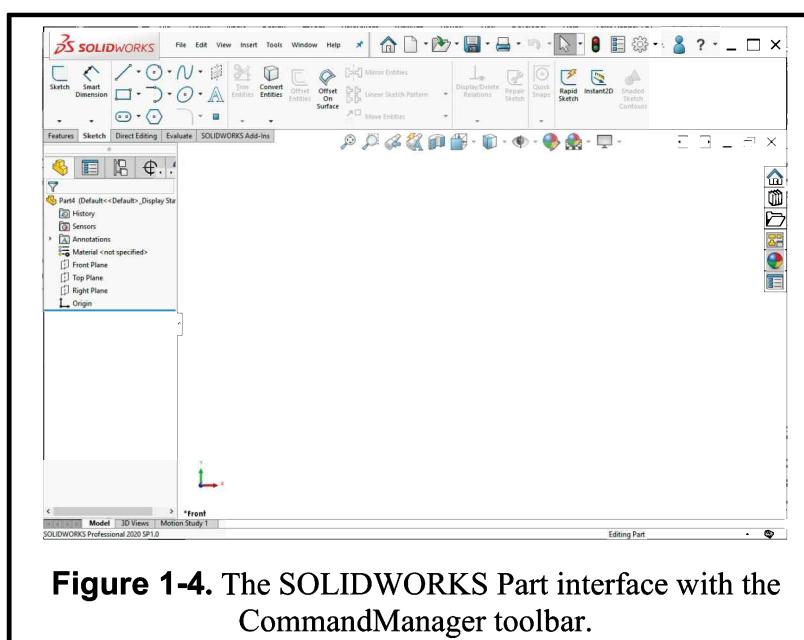


Figure 1-4. The SOLIDWORKS Part interface with the CommandManager toolbar.

MAIN MENU – to prevent this menu from automatically hiding, select the Push pin on the right of the menu bar.

File is where you can create New files and Open, Save, Print Preview, and Print existing files.

Edit is where you can find commands to modify existing features, Undo the last command, Cut, Copy, and Paste Entities. It also has the Rebuild command to update the model when you make changes.

View is where you can change the model's Display parameters, change the View Orientation, and select view manipulation commands like Zoom, Rotate, and Pan.

Insert is where you find most feature creation commands, including Extrusions, Cuts, Sweep, Loft, Fillets, etc.

Tools include multiple commands to help you create, modify, analyze and change document and system options.

Window is where you can select between multiple open documents and arrange them to tile them vertically or horizontally on the screen.

Help is where you can access SOLIDWORKS' user help and online tutorials.

FEATUREMANAGER TREE

The SOLIDWORKS **FeatureManager** Tree is the pane on the left side of the screen used to keep track of the solid modeling process and operations. It lists every feature and their sketch in a chronological tree diagram, where you can edit every feature if needed, and is included in all SOLIDWORKS documents. **Figure 1-5** shows an example of the FeatureManager tree.

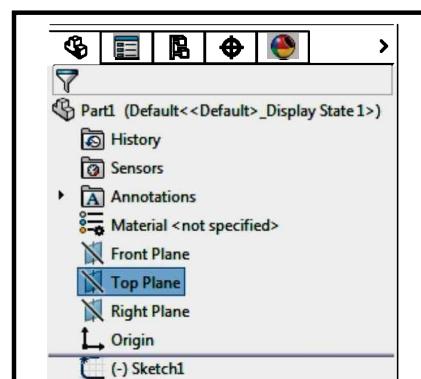


Figure 1-5.
FeatureManager Tree.

THE REBUILD BUTTON

The “**Rebuild Button**” is located on the Standard Toolbar and looks like a red/green traffic light, as shown in **Figure 1-6**. When you select it, the model is rebuilt and updates any changes that have been made. When a model is rebuilt, the current command is terminated.

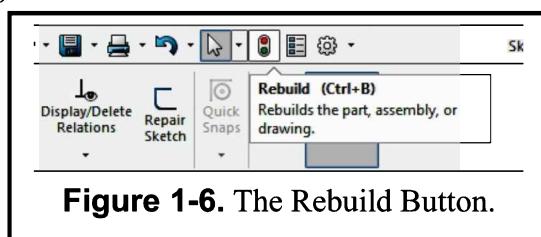


Figure 1-6. The Rebuild Button.

TOOLBARS

Toolbars include the commands available to create and modify models grouped by functionality. To see all the available Toolbars, right click on any toolbar, as shown in **Figure 1-2**, or select the menu **View, Toolbars**.

The Heads-Up **View Toolbar** is shown in **Figure 1-7** and includes the commands to manipulate the view including Zoom, Pan (move) and Rotate. Using the drop down command options you can change the model's display style to Hidden Lines Removed, Hidden Lines Visible, Shaded, etc.; options to change the model's orientation to Front, Back, Left, Right, Top, Bottom, Isometric and perpendicular to a selected plane or face; and visibility controls to hide or show auxiliary geometry.

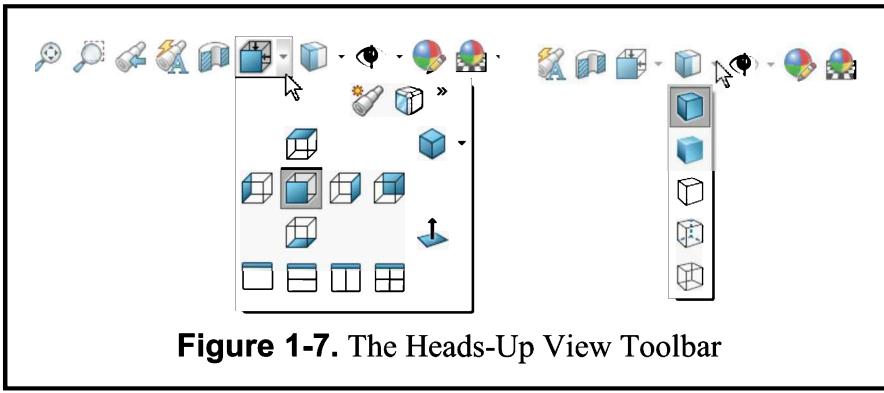


Figure 1-7. The Heads-Up View Toolbar

SKETCH TOOLBAR:

The sketch toolbar, shown in **Figure 1-8**, includes the tools to create sketch geometry. The Dimensions and Relations toolbar have the tools to add relations and dimension sketch entities. To start, you click on the **Sketch** icon.

As you create 2D sketch entities, you can use **Dimension** and **Add Relation** buttons to define the geometric parameters of your design. The sketch tools available include numerous 2D commands including **Line**, **Arc**, **Circle**, and **Rectangle**. To edit your sketch, several common commands like **Trim**, **Mirror**, **Fillet**, and **Offset** are available.



Figure 1-8. Sketch Toolbar Options.

MODEL PLANES

Before you start a 2D sketch in SOLIDWORKS, you must select a sketch plane. The three default orthogonal planes are the three standard orthographic views: **Front**, **Top**, and **Right**, as shown in **Figure 1-9**. After a part is created, you can also use any plane or flat surface of the model. The surface does not have to be parallel to one of the three principal planes to be used for a sketch.

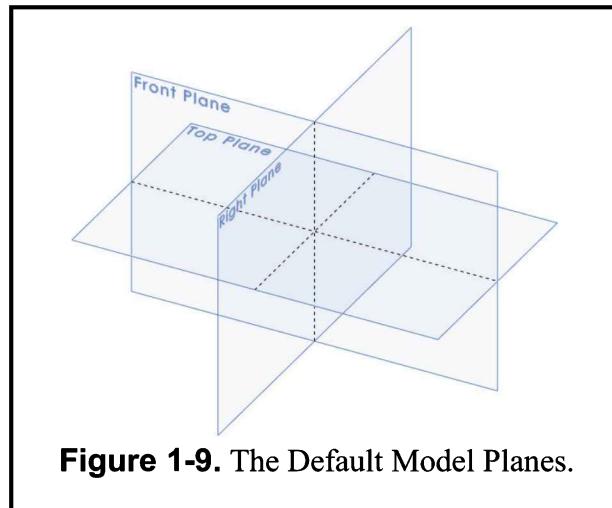


Figure 1-9. The Default Model Planes.

SKETCH LINE COLORS

A SOLIDWORKS sketch can be in any of many different states, and you can identify them by the sketch element's color. When sketching 2D profiles, the geometric elements are colored as follows:

- **Cyan** indicates the sketch elements are **Selected**.
- **Blue** indicates the sketch elements are **Under Defined**, which means they don't have the necessary dimensions and/or geometric relations needed to define them (undesirable).
- **Black** indicates the sketch elements are **Fully Defined** and have the necessary dimensions and/or geometric relations needed to define them (preferred).
- **Yellow or Red** indicates the sketch elements have conflicting dimensions and/or geometric relations and are **Over Defined** (undesirable).
- **Gray** indicates these entities are not in the active sketch.

STANDARD TEMPLATES

Before you start any exercises in SOLIDWORKS, it is a good idea to establish some standard 3D Part and 2D Drawing templates. One set will use the American National Standards Institute (ANSI) standard in Inches and the other for Metric dimensions.

To start, go to the menu **File, New**, select **Part** and click **OK**. Go to the menu **Tools, Options**, select the **Document Properties** tab, make sure the **Dimensioning Standard** is set to **ANSI** (see **Figure 1-10**). Next, select **Units** in the left-hand column of the dialog and activate **IPS (Inch, Pound, Second)**. You should also set the Decimal Places to three (3) as indicated in **Figure 1-11**. Click **OK** to close the “Document Properties”.

Now go to the menu **File, Save As**, change the **Save as Type** selection to **Part Templates (*.prtdot)**, select the **FOLDER** where you wish to save this template, and name the template **ANSI-INCHEES**. Make sure that you are saving this into your personal folder.

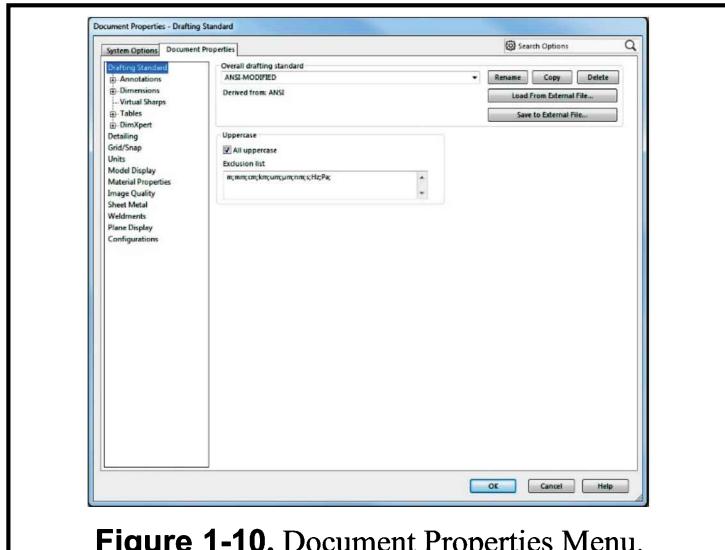


Figure 1-10. Document Properties Menu.

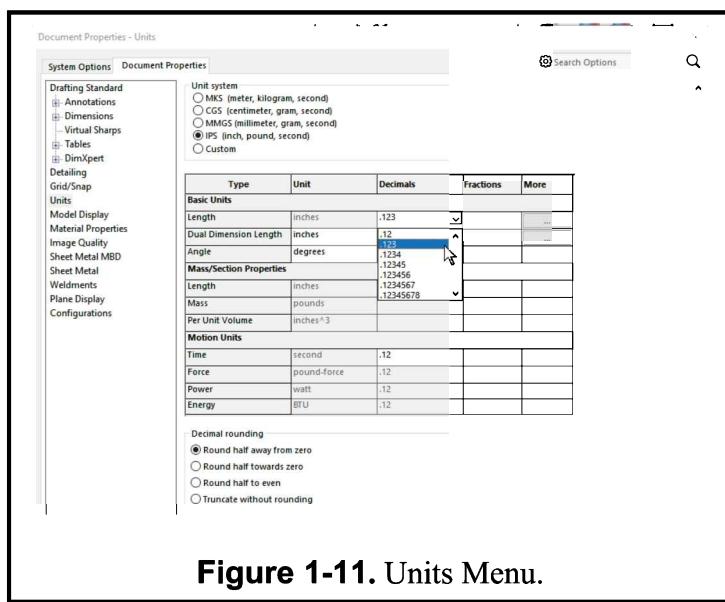


Figure 1-11. Units Menu.

Repeat the above process, only this time change the **Units** to **MMGS (Millimeter, Gram, Second)**, and two decimal places. This should be sufficient for metric measurements. Save this template to your **FOLDER**. Remember to change the **Save as Type** option to **Part Templates (*.prtddot)**, and set the name as **ANSI-METRIC**. **Make sure the template is saved to your personal folder.**

STANDARD DRAWING SHEET WITH TITLE BLOCK

The next process is to set up a 2D Drawing with a Title Block to document the two dimensioning standards. You may have to draw a title block (see **Figure 1-12**) if one has not been provided. An example of a drawing with a title block, with suggested dimensions and style, is available in **Appendix A** and included with the book's files.

- a) If a 2D Drawing template has not been provided, select the menu **File**, **New**, select the **Drawing** and click **OK**. Select the size **A-(ANSI) Landscape**. Turn off the **Display Sheet Format** option and click **OK**. If needed, click on the red **X** to cancel the **Model View** command.
- b) If a 2D Drawing template with Title Block has been provided, go to the menu **File** and **Open** that drawing file.

Go to the menu **Tools, Options**. In the **Document Properties** tab make sure the **Dimensioning Standard** is set for **ANSI**. Next, select **Units**, activate **IPS (Inch, Pound, Second)**, and set to three **(3)** decimal places; this will have enough accuracy unless you do a high tolerance drawing.

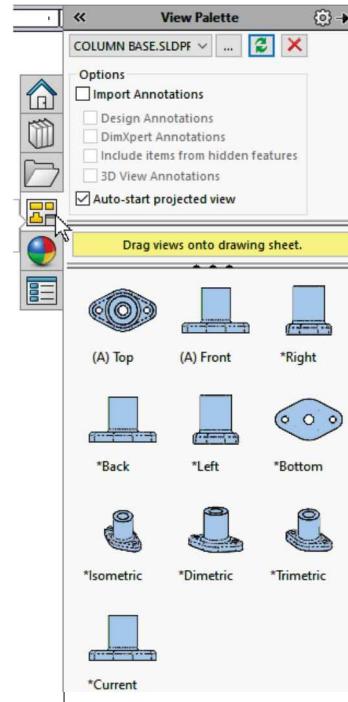
To draw the Title Block, right click on **Sheet1** in the **FeatureManager** and select **Edit Sheet Format**. Use the Sketch and Annotations toolbar tools to draw the title block. Select the menu **Insert, Annotations, Note** to add your personal information, including **NAME**, **DESK NUMBER**, and **SECTION NUMBER**. When finished, in the **FeatureManager Tree** right click on **Sheet1** and select **Edit Sheet**. With the title block complete, select the menu **File, Save As**. From the **Save as Type** drop down list select **Drawing Template (*.drwdot)**, select the folder where you wish to save this template, and save your template as **TITLEBLOCK-INCHES**. **Make sure that you are saving it to your personal folder.**

Repeat the above process, only this time, in the **Units** section select **MMGS (Millimeter, Gram, Second)**. Two **(2)** decimals should be sufficient for metric measurements. Add your personal information such as **NAME**, **DESK NUMBER**, and **SECTION NUMBER**. In the **FeatureManager Tree**, right click on **Sheet 1** and select **Edit Sheet**. Select the menu **File, Save As**. From the **Save as Type** drop down list select **Drawing Template (*.drwdot)**, select the folder where you wish to save this template, and save your template as **TITLEBLOCK-METRIC**. **Make sure that you are saving it to your personal folder.**

ADDING A DRAWING VIEW OF A PART (3D MODEL) TO A 2D DRAWING SHEET

For each of the assignments you will be asked to submit a printed copy of the part that you built in SOLIDWORKS. The following will be the procedure to accomplish this requirement.

- A. Build the solid model according to instructions and save that part in your folder. A part's file extension is **.sldprt**.
- B. Open the drawing template to be used (see **Figure 1-12** for example). Use the inches or metric drawing template as needed. If you choose the incorrect template, remember you can change the units of measurement later in the menu **Tools, Options, Document Properties, Units**.
- C. From the **Task Pane** toolbar, click on the **View Palette**, select the part you want to add to the drawing from the open files list, or browse to open a new part.
- D. Click and drag the Isometric drawing view from the list onto the sheet. In unit nine and ten you will learn how to make a complete detail drawing using multiple orthogonal views. If you do not see the model view after dragging it to the sheet, it means you are editing the **Sheet Format**. In this case, right click on **Sheet1** in the **FeatureManager Tree** and select **Edit Sheet**.
- E. To change the scale of a drawing view, select the view, in the **Scale** section of the **PropertyManager** pane select **Use Custom Scale** to change the scale of the view as needed. In the **Display State** section select **Shaded with Edges** for a shaded view.
- F. In the Title Block area of the drawing sheet, add the necessary notes using the menu **Insert, Annotations, Note**. Provide the exercise number in the upper right box. Your Title should be placed in an open area of the Title Block using a larger font size, i.e., **20 to 24 PT or .25"**. Your name, desk, Sec and exercise number should use a font size of **12pt or .125"**.



HOW TO USE A TEMPLATE

To use the previously created templates, there are two options, depending on the permissions allowed by the system administrator: You can add the files to a SOLIDWORKS templates folder (preferred option) or open the template files individually.

1. *To use the SOLIDWORKS templates functionality, you need access to modify SOLIDWORKS system settings.* Create a folder named "**My SW Templates**" in your personal folder and copy your templates to this folder. In SOLIDWORKS go to the menu **Tools, Options**, and in the **System Options** tab select **File Locations**. In the drop-down list select **Document Templates**, click **Add** and browse to the "**My SW Templates**" folder.

Design Workbook Using SOLIDWORKS 2020

To use the templates, click on the **New** command or the menu **File, New**. If you cannot see the “**My SW Templates**” tab, click on the **Advanced** button in the lower left. Select “**My SW Templates**” tab, click on the template you want to use and click **OK**.

2. If you do not have permission to modify system settings, the best option is to open your templates folder, right click on each template and select **Properties** from the context menu. In the file’s properties turn on the “**Read Only**” attribute and click **OK**. Using this option will prevent you from accidentally modifying your templates.

To use a part or drawing template, open it in SOLIDWORKS; you will see the name of the template followed by **[Read Only]** in the title. At this time, you can start working on your new part or drawing. When you click on **Save**, SOLIDWORKS will automatically change the file type to part (*.sldprt) or drawing (*.slddrw), without modifying the original template.

NOTE: When using this option, if the file’s **Read Only** attribute is not set, any changes made will be saved to the template.

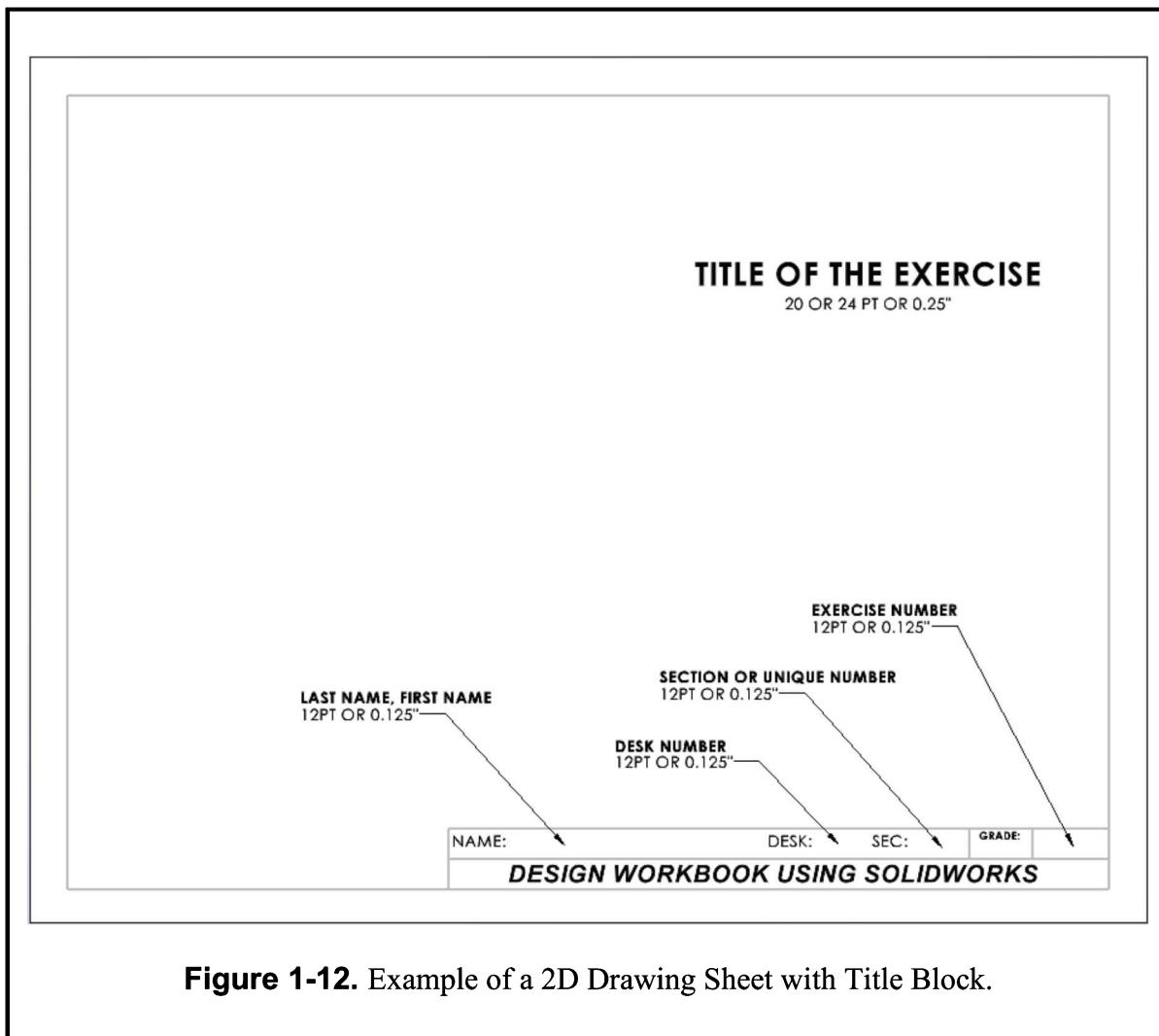


Figure 1-12. Example of a 2D Drawing Sheet with Title Block.

Exercise 1.1: CARBON FIBER GASKET

In this first exercise, you will build a Carbon Fiber Gasket part. To complete this part, you will create a 2D sketch and extrude to create a 3D part. In this exercise you will use many of the 2D sketching tools available.

If you can use the SOLIDWORKS templates folder option, click on the **New** command, select **ANSI-INCHES** from the “**My SW Templates**” tab and click **OK** as shown in **Figure 1-13**. Otherwise, click on the **Open** command, browse to your templates folder, and open the **ANSI-INCHES.prtdot** file. Using either approach, click on **Save** and name your part **CARBON FIBER GASKET**.

The new part’s units are already set to inches with 3 decimal places, as our template. If you need to change them, select the menu **Tools**, **Options**, select the **Document Properties** tab, and click on **Units**. Change the units of measure if needed and click **OK** to finish. The 2D sketch for the gasket will be created in the **Front Plane**. In the FeatureManager tree select the **Front Plane** (it will be highlighted in the screen).



Select the **Sketch Tab** in the **CommandManager** and click on the **Sketch** icon.

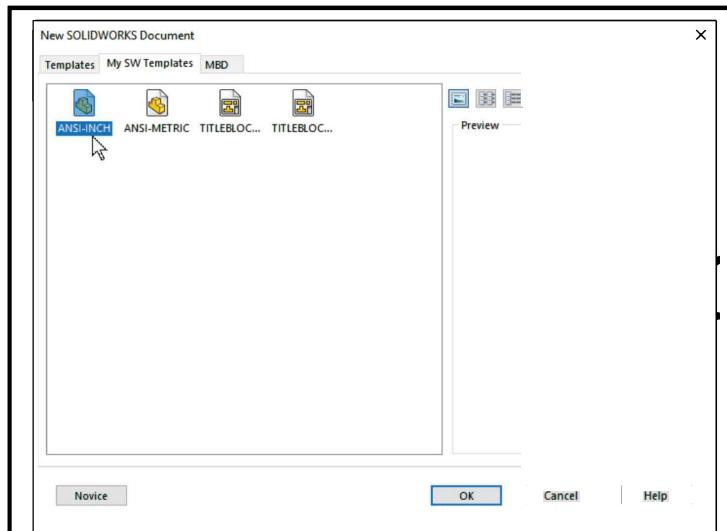


Figure 1-13. Setting Grids and Units for the Sketch.

After the sketch is created, your part will be automatically reoriented to a Front view, showing the **Front Plane** parallel to the screen. If needed, you can change the view orientation using the Heads-Up View toolbar, as shown in **Figure 1-7**.

- Select the **Line** tool from the sketch toolbar and draw the profile shown in **Figure 1-14** starting at the origin. Draw the profile approximately as shown. When drawing either vertical or horizontal lines, SOLIDWORKS automatically shows guides to let you know a horizontal or vertical geometric relation will be added to the line, constraining its orientation.

When finished press the **ESC** key to turn off the **Line** tool. At this time, you can use the left mouse button to drag the blue geometric elements, because their size has not been defined yet. To fully define the sketch geometry, you will now add geometric relations and dimensions.



Next, select the **Smart Dimension** tool to add the dimensions shown. Select the endpoints to dimension, locate the dimension on the screen, and enter the value indicated. If you need to change a dimension's value double click on it, enter the new value and click OK.

To fully define the sketch geometry up to this point, you need to add a geometric relation to make the two vertical lines equal. Hold down

the **Ctrl** key and select both vertical lines, release the **Ctrl** key and click on the **Equal** relation in the **Properties** pane. By making both lines the same length, the sketch is now fully defined, meaning that all necessary dimensions and geometric relations have been entered, and the sketch changed from blue to black, letting you know the sketch is fully defined.

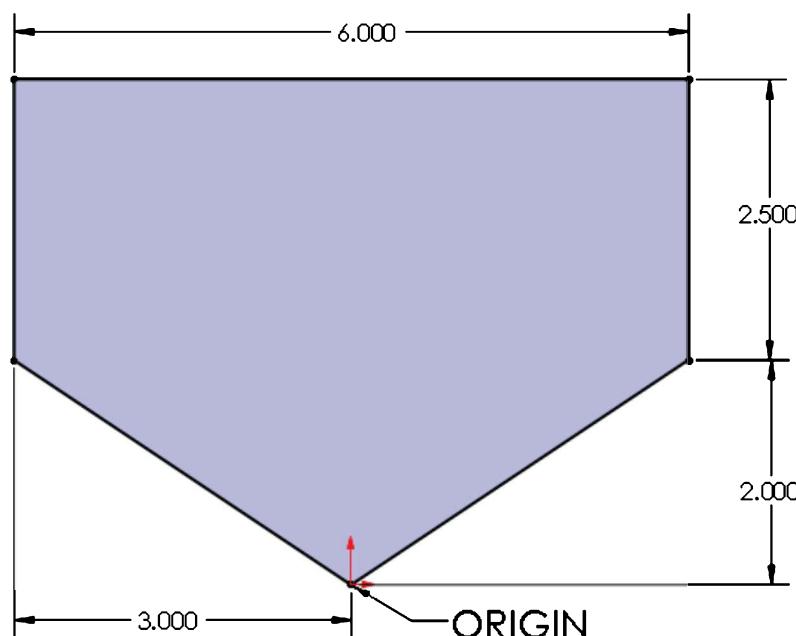
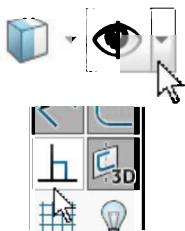


Figure 1-14. The Initial Outline for the Carbon Fiber Gasket.



To visualize the geometric relations automatically captured and manually added, click in the **Hide/Show** drop down command in the **View** toolbar, and turn on the **View Sketch Relations** command. The existing sketch relations will be indicated with the green icons next to each geometric element. This option can be turned on or off as needed, as seen in **Figure 1-15**.

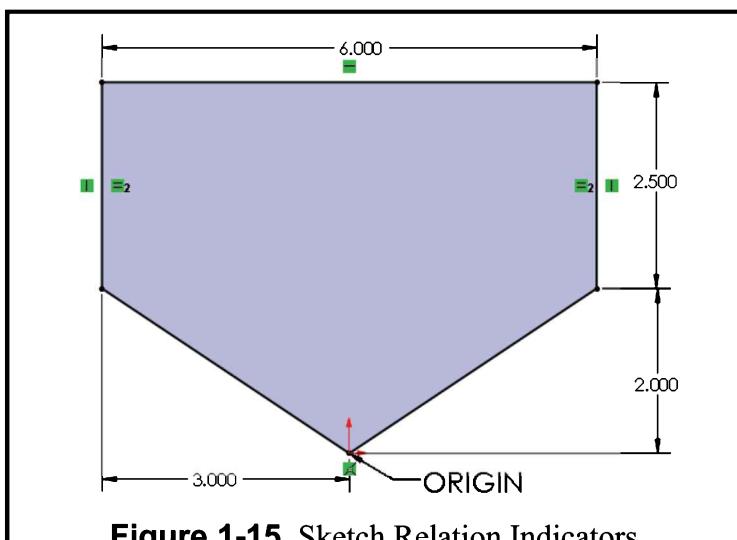


Figure 1-15. Sketch Relation Indicators.

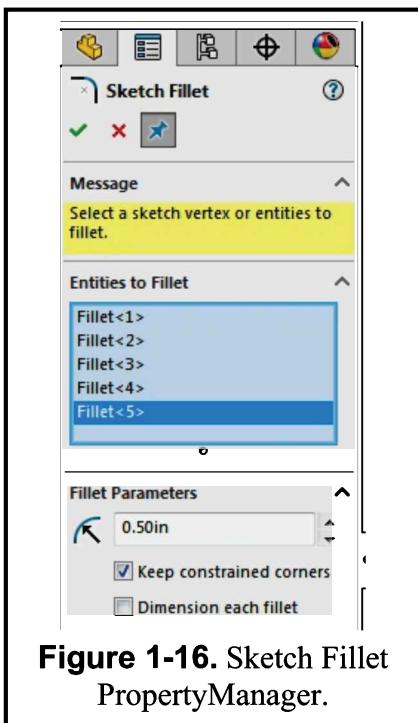


Figure 1-16. Sketch Fillet PropertyManager.

The next step is to round the corners of the gasket. This will be done using the **Sketch Fillet** command. Select the **Sketch Fillet** icon from the Sketch toolbar. The **Sketch Fillet** PropertyManager will be displayed in place of the FeatureManager (**Figure 1-16**). In the fillet radius enter **0.50** inches and select each of the vertices (intersections) in the sketch to create a fillet.

Since we added an Equal geometric relation to two lines, when adding the sketch fillet will change the length of the lines, and you will be asked if you want to proceed. Click Yes to continue and select the next intersection. Click OK to finish and close the Sketch Fillet command.

Note there is only one fillet dimension added to our sketch. The reason is that all fillets made at the same time will have an **Equal** sketch relation and changing this dimension will change the size of all fillets at the same time. The sketch profile with fillets is shown in **Figure 1-17**.

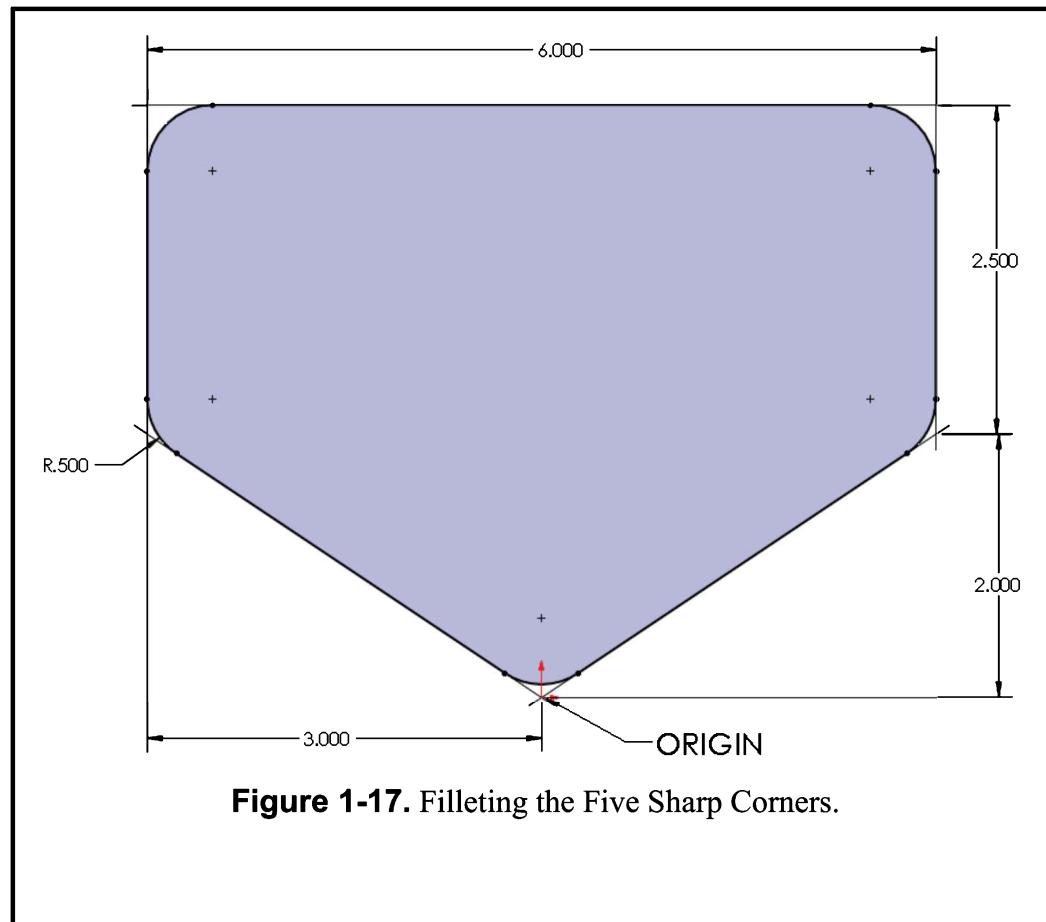
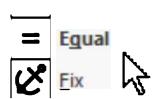


Figure 1-17. Filleting the Five Sharp Corners.

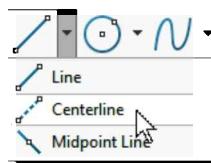


Next, select the **Circle** command, and create the five circles concentric to the fillets (rounded corners) of the outline. All circles will have the same diameter of **0.375"**.



After the five circles are drawn, cancel the **Circle** command, hold down the **Ctrl** key and select the five circles, release the **Ctrl** key, and add an **Equal** geometric relation to make the five circles the same diameter. Use the Smart Dimension tool and dimension one circle **0.375"**. Note all five circles change size at the same time.

Note: While you are using a sketch entity tool like **Line**, **Rectangle**, **Circle**, etc., the cursor will show a small icon that indicates the current type of sketch entity that you are using.



Since the gasket is symmetrical, it is convenient to have a centerline to use it as support geometry. Select the **Centerline** command from the **Line** drop down command, and draw a vertical centerline starting at the origin going up through the part. This centerline will be used in the next two steps.

□ Select the **Straight Slot** tool and draw a horizontal slot above the bottom circle. If the option **Add Dimensions** is turned on; the slot will have a width and height dimension added automatically. Double click and change the width to **3.00"** wide, and the height to **0.50"**. Select the **Smart Dimension** tool and add a dimension of **2.00"** between the slot's centerline and the origin.



To center the slot in the gasket, turn off the dimension tool, hold down the **Ctrl** key to select the **Point** in the slot's midpoint and the vertical centerline, and add a **Coincident** relation. The fully defined slot is shown in **Figure 1-18**.

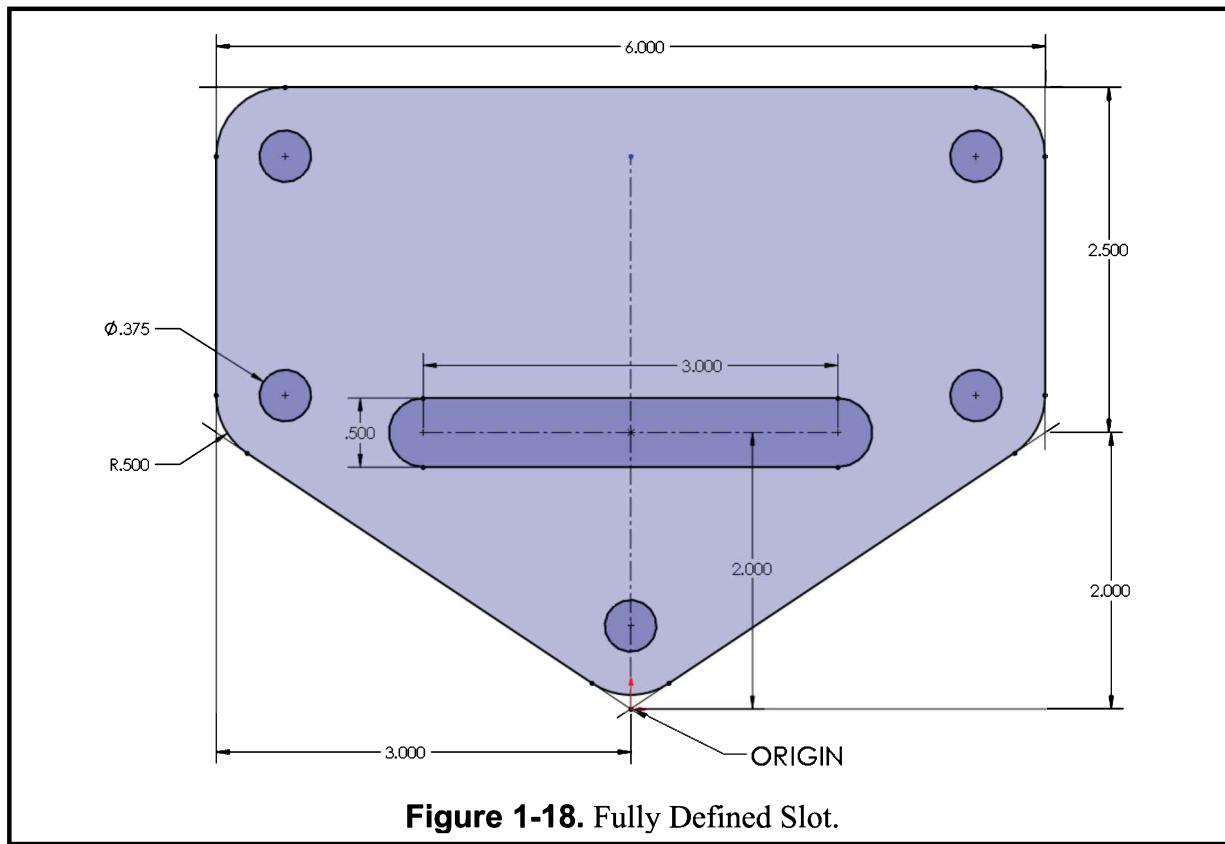


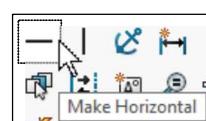
Figure 1-18. Fully Defined Slot.

Many models have symmetrical features, and SOLIDWORKS offers tools to make those features easy to duplicate. You will now draw two polygons on the gasket that are symmetrical about a centerline. First, select the vertical centerline previously drawn, and select the menu **Tools, Sketch Tools, Dynamic Mirror**. The centerline will now have an equal sign at either end to indicate the Dynamic Mirror is active. This means that anything drawn on one side of the centerline, will be automatically mirrored on the other side.



Select the “**Polygon**” command from the sketch toolbar. When polygon’s options are displayed, set the number of sides equal to **6** and select the Inscribed circle option, as indicated in **Figure 1-19**.

Draw a polygon on either side of the vertical centerline. First locate the center of the polygon, then move your cursor horizontally to define the size and rotation of the polygon. Select the **Smart Dimension** tool and dimension the horizontal and vertical location of the polygon, as well as the diameter of the inscribed circle.



To fully define the sketch, select a hexagon’s horizontal line, and from the pop-up menu click on **Make Horizontal** to add a geometric relation to prevent the hexagon from rotating. Your finished sketch will look like **Figure 1-20**.

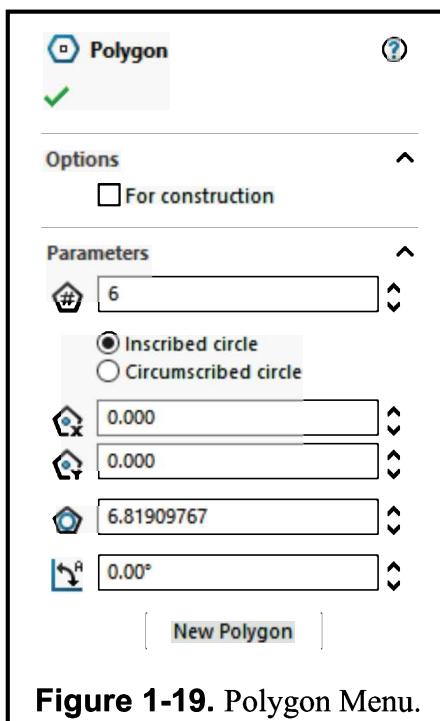


Figure 1-19. Polygon Menu.

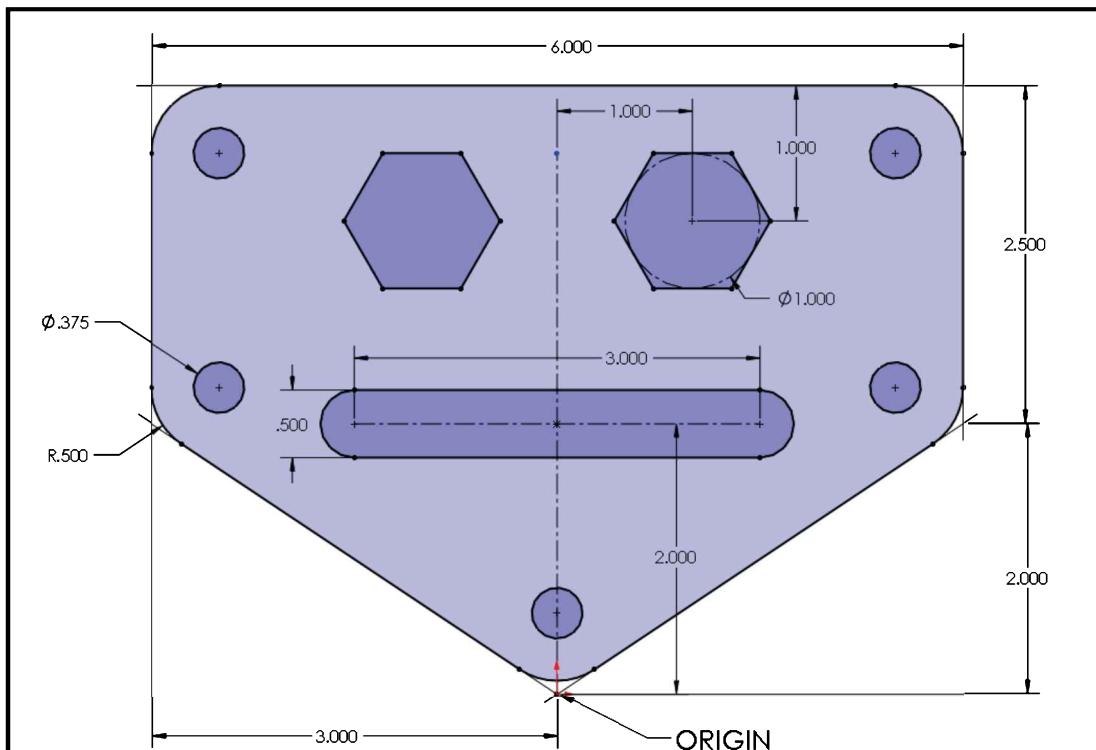
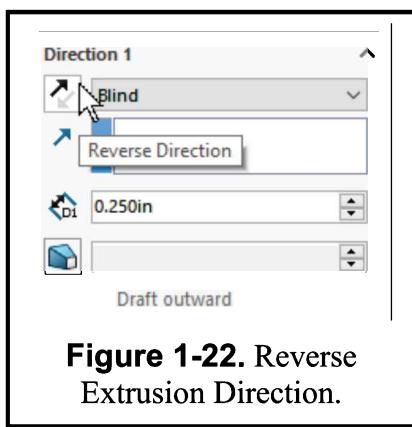


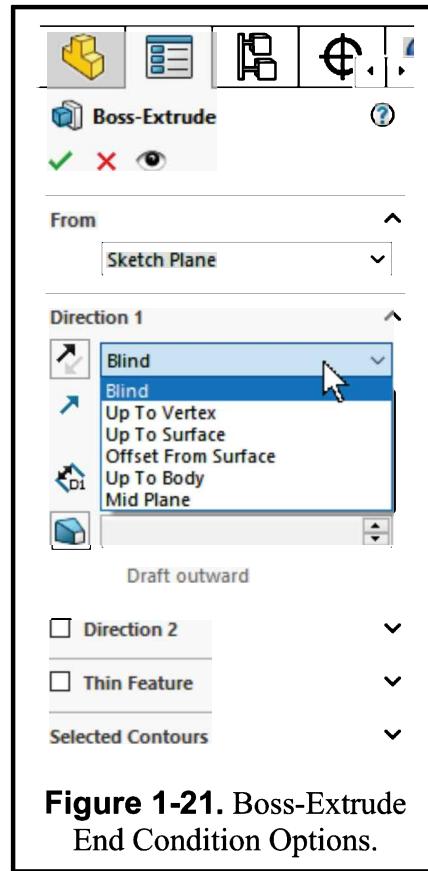
Figure 1-20. Fully Defined Finished Sketch.

EXTRUDE THE GASKET SKETCH

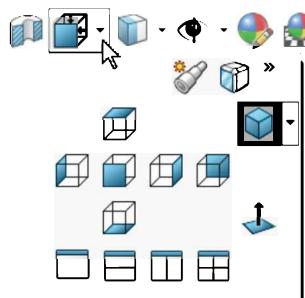
The **Extruded Boss/Base** command is in the menu **Insert, Boss, Extrude**, or the **Features** tab of the **CommandManager**. As its name implies, this command will extrude a sketch or selected sketch contours in one or two directions to create a solid feature. The default end condition for an extrusion is **Blind**, which will extrude the sketch a defined distance. Different extrusion end conditions are available, as seen in **Figure 1-21**. Many of these end conditions will be covered in detail in following lessons.



If needed, you can change the direction of the extrusion with the **Reverse Direction** button next to the “**End Condition**” selection box (See **Figure 1-22**).



Now extrude the gasket to its needed thickness. In the **Boss-Extrude** properties set the end condition to **Blind** and set the extrusion depth to **0.25"**, as shown in **Figure 1-21**. The first time you create an extrusion in a part, the model’s view is automatically rotated to an Isometric orientation, and a preview of the extrusion is shown in the screen. Click **OK** to finish the extrusion.



Using the View Orientation toolbar, explore the different orientations of the model, including all orthographic, **Isometric**, **Dimetric** and **Trimetric** views.

Feel free to manipulate the view; you can **Rotate**, **Zoom** and **Pan** the model. To **Rotate** the view, press down the middle mouse button and drag in the screen. To **Zoom** roll the mouse wheel to zoom out, and reverse to zoom in. The model will zoom in at the cursor location. Additional view controls are found in the menu **View, Modify**.



If you would like to change the color of your model, right click on the model name at the top of the FeatureManager tree and select **Appearances** in the context menu. You can then assign any color to your model.

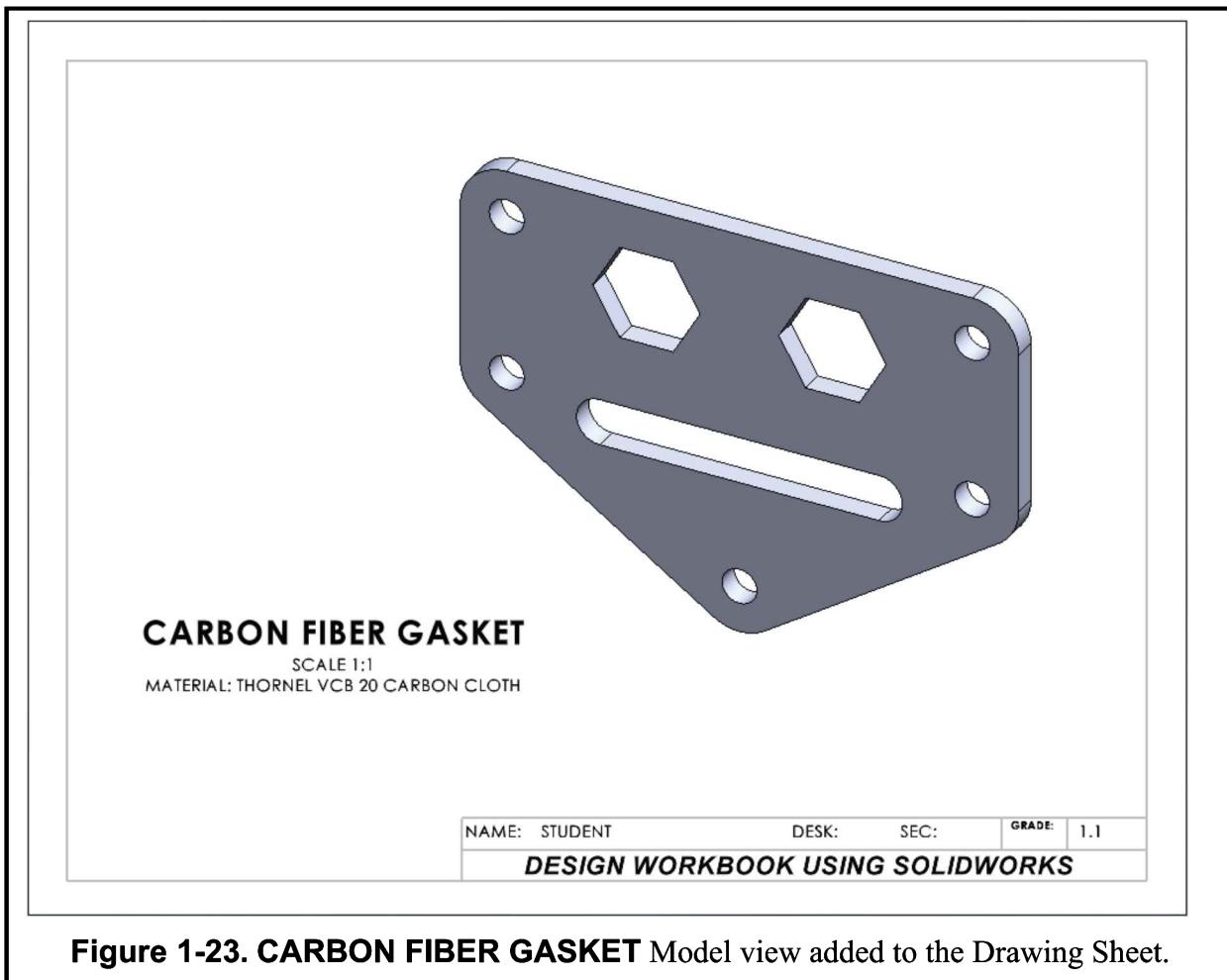
Save your part as **CARBON FIBER GASKET** in your personal folder. The file type will be automatically set to **Part (*.sldprt)**.

To complete this exercise, you need to print a copy for submission to your instructor. *Check with your instructor for any special printing instructions.*

Using the **TITLEBLOCK-INCHES** template, follow the instructions provided on page 1-7 to add a **Trimetric** view, and change the view's scale to 1:1. Use the menu **Insert, Annotations, Note** to add the missing notes to complete the drawing as shown in **Figure 1-23**. Be sure to use the recommended font and size for the required annotations, as indicated in **Image 1-12**. Don't forget to add the exercise number in the space to the right of the GRADE box.

Save your drawing as **CARBON FIBER GASKET**, using the **Drawing (*.SLDDRW)** file type. Notice that the title of the part and the drawing are the same, but the extension is different. The 3D solid model and the 2D drawing are linked, meaning that any changes made to the solid model will be automatically updated on the drawing.

Before you print your copy, make sure you have set the **File, Page Setup** to **Landscape** mode. Use the menu **File, Print Preview** to send it to the default printer.



Exercise 1.2: COVER PLATE

There are many different approaches to creating a 2D sketch using **SOLIDWORKS**. The approach will depend on the design intent and the designer's own preferences and choices. In the previous Exercise 1.1, the sketch was added to the Front plane. In this exercise you will add a sketch on the Top plane since this is the normal orientation of the cover plate.

Make a new part using the **ANSI-INCHES** template following the instructions on [Page 1-7](#). Select the **Top plane** in the FeatureManager, and from the **Sketch** tab in the **CommandManager** select **Sketch**. After adding the first sketch, the part's orientation will change to make the selected sketch plane parallel to the screen, in this case to a **Top** view.

 Select the **Circle** command and draw a circle centered at the origin. Use the **Smart Dimension** command and make the circle's diameter **10.00"**. After the circle is dimensioned, the sketch is fully defined, and you are ready to make an extrusion.

 From the **Features** tab select **Extruded Boss**, or from the menu **Insert, Boss/Base, Extrude**. Make the extrusion using the Blind end condition, **0.500"**, and click **OK** to finish. The cover plate's **Isometric** view is shown in [Figure 1-24](#).

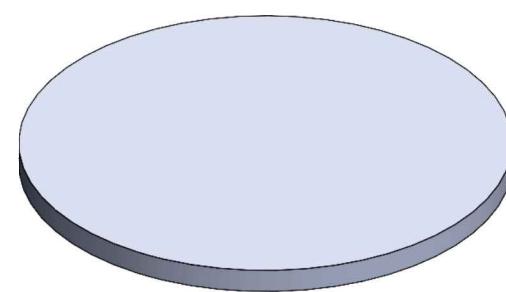


Figure 1-24. Base Feature of the Cover Plate.

The next step is to make the sketch for the indentation of the Cover Plate. Change the view orientation to a **Top** view, select the top face of the part, select the **Sketch Tab** and then click on the **Sketch** icon to start the next sketch. Select the **Circle** tool and draw two concentric circles centered at the origin. Using the **Smart Dimension** tool, set the diameter of the smaller circle to **3.50"** and the diameter of the larger circle to **8.50"**.

Use the **Line** tool and draw a horizontal line from the origin to the large circle. Next, draw a line from the origin at an angle above the horizontal line, also up to the large circle. Add a **22.5°** angle dimension between the two lines. Select the horizontal line and then the angled line to add the angular dimension, as seen in [Figure 1-25](#).

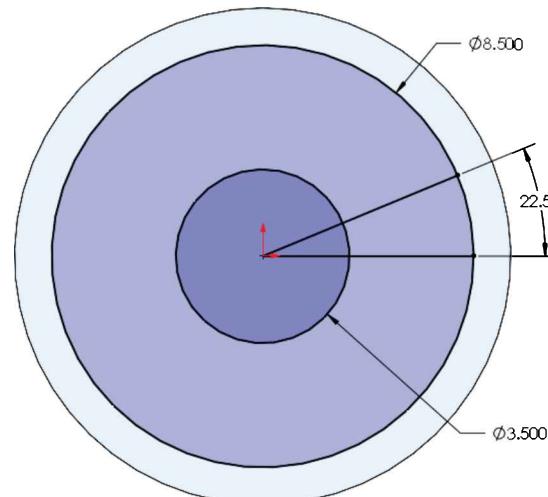


Figure 1-25. Start Geometry for the Indentations.



Now select the **Trim** tool from the sketch toolbar. In the **Trim command options** in the PropertyManager select the **Trim to Closest** option. Note the cursor has a small scissors icon next to it, letting you know the trim command is active. Use the left mouse button to Trim the lines and circles to match **Figure 1-26**.

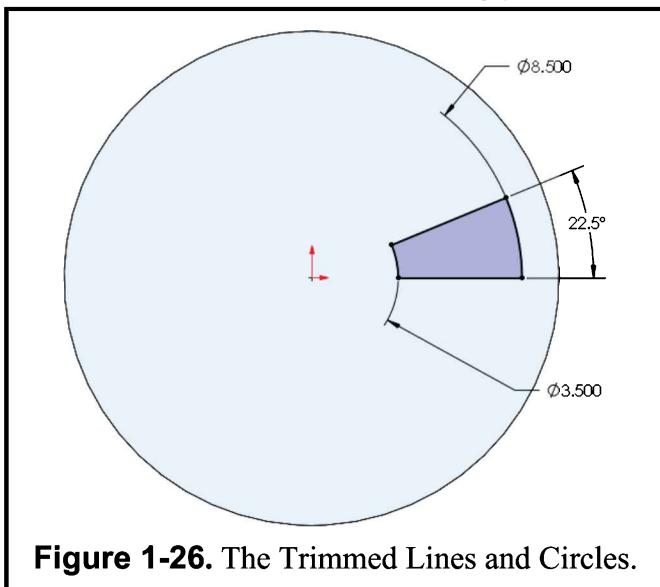


Figure 1-26. The Trimmed Lines and Circles.

After the angled line is trimmed, it turns blue, meaning it is no longer defined. To see what this means, click and drag it with the left mouse button; you will see it move up and down.

To fully define it again, hold down the **Ctrl** key, select the angled line and the origin, and add a **Coincident** geometric relation.

The next step is to add a circular pattern of the sketch profile. From the **Linear Sketch Pattern** drop down command, select **Circular Sketch Pattern**.

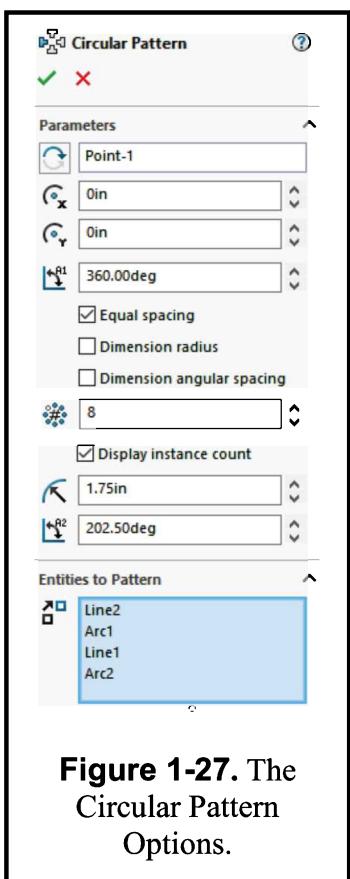
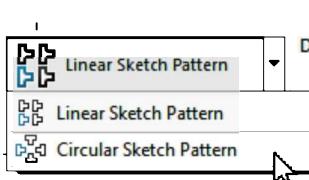


Figure 1-27. The Circular Pattern Options.



Activate the first selection box (Center of Circular Pattern) and select the Origin, make sure the **Equal spacing** option is checked, and change the number of instances to **8**. Keep in mind this value includes the original. The last step is to activate the **Entities to Pattern** selection box and select the four sketch elements trimmed in the previous step. Review the preview and click OK to finish. The completed sketch is shown in **Figure 1-28**.

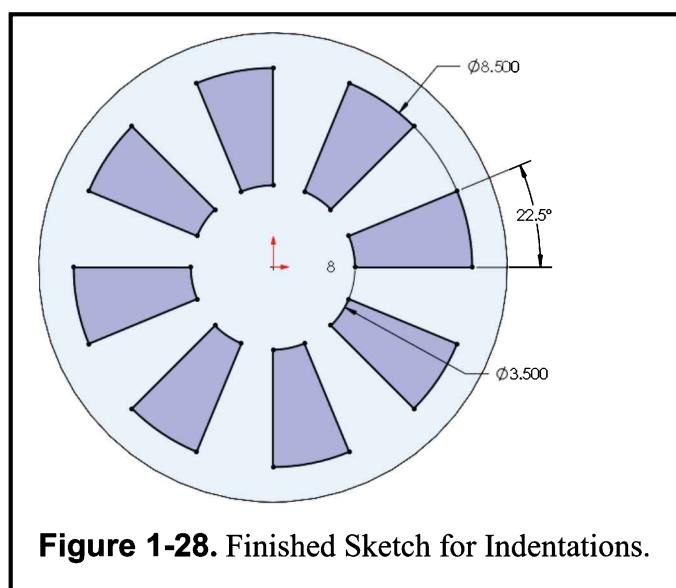


Figure 1-28. Finished Sketch for Indentations.

Now you will make a Cut Extrude to make the indentations in the cover plate. At this point, it is helpful to change to an **Isometric** view orientation. From the **Features** tab select the **Extruded Cut** command. In the Cut-Extrude PropertyManager enter set the direction 1 End condition to **Blind** and set the distance to **0.25"**. Click OK to finish. The finished part's **Isometric** view is shown in **Figure 1-29**.

You will now create the lift hole in the middle of the plate. Change to a **Top** view orientation, select the top face of the cover plate, and add a new **Sketch**. Since the lift hole will be symmetric about the origin, you can use the mirror function to better capture the design intent. Start by drawing a vertical **Centerline** through the origin. Next, select the menu **Tools**, **Sketch Tools**, **Dynamic Mirror**. An equal sign will be added to both ends of the centerline. Remember every operation you perform on one side will be duplicated on the other side. Now, draw a small **Circle** to the right side of the centerline and two parallel **Lines** that start at the centerline and go to the circle. Use the Smart Dimension to dimension the sketch as indicated in **Figure 1-30**.

Hold down the **Ctrl** key, select the center of one circle and the **Origin**, and add a **Horizontal** geometric relation.

Select the **Trim** tool and remove the inside lines of the circles between the two parallel lines. If you get a warning from SOLIDWORKS about eliminating associated dimensions while you are trimming, answer **Yes** to proceed with the trim, and re-create the missing dimensions.



Use the **Extruded Cut** command from the **Features** tab, or the menu **Insert**, **Cut**, **Extrude**. Change to an **Isometric view**, select the **Through All** end condition for Direction 1 and click the **OK** button. The finished part up to this point is shown next.

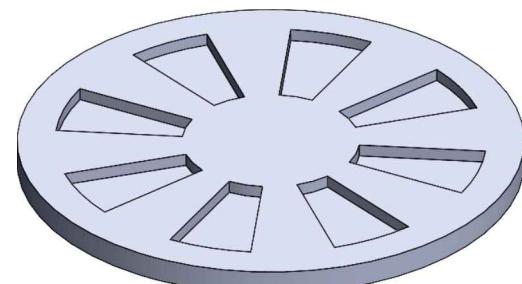


Figure 1-29. The Indentations Pattern Cut into the Cover Plate.

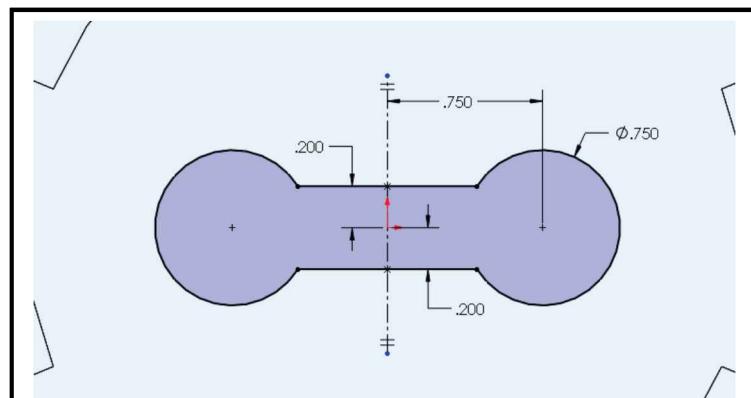
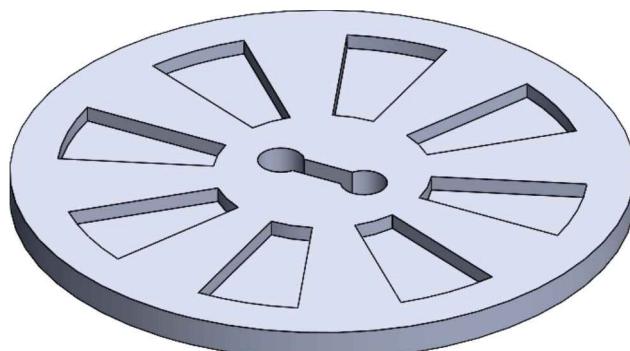


Figure 1-30. Finished Lift Hole Sketch.



To finish the cover plate, you need to add four circular cuts around the perimeter for the guide holes. Change to a **Top** view, select the top face of the cover, add a new **Sketch**, and draw four **Circles**. The circles should be at 90 degrees to each other, and their centers will be located on the perimeter aligned horizontally and vertically with the origin.

NOTE: For the left and right circles, add a **Horizontal** geometric relation between their center and the origin, and for the top and bottom circles, add a **Vertical** relation to the origin. This way the circles will be fully defined.

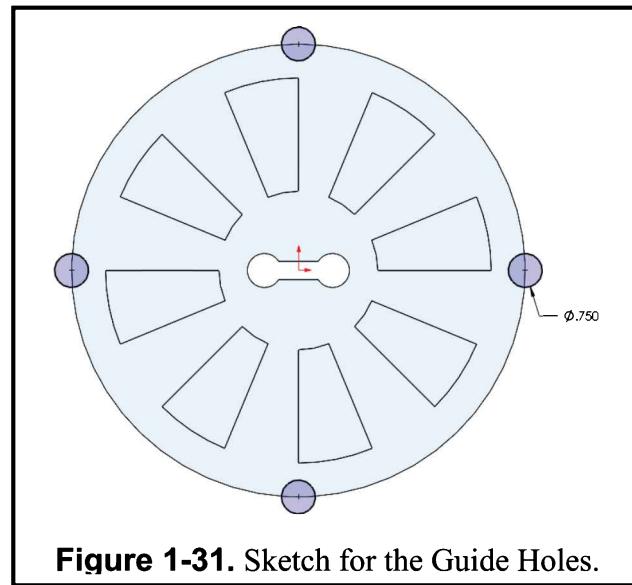


Figure 1-31. Sketch for the Guide Holes.

After drawing the circles select all four of them and add an **Equal** relation to them. To finish, dimension one of the circles **0.75"**, as shown in **Figure 1-31**.

OR

Draw a single circle and use the **Circular Sketch Pattern** tool, using the part's origin as the center of the pattern. Set the number of instances to **4** and click **OK** to finish.

Now select the **Extruded Cut** command from the Features tab. Set the end condition to **Through All** and click **OK** to finish. Your model will look like **Figure 1-32**. To change the part's color, right click in the model name in the FeatureManager, select the **Appearances** command from the context menu, and assign a color to your model.

Save your part as **COVER PLATE.sldprt**, make a new drawing using the **TITLEBLOCK-INCHES** template and add an isometric view. Follow the instructions given on **page 1-7**. Remember to add the missing notes to complete the drawing.

Save the new drawing as **COVER PLATE.slddrw**. Remember the part name and the drawing are the same, except with a different extension. The solid model and the drawing are linked, meaning that any changes made to the solid model will be automatically updated in the drawing.

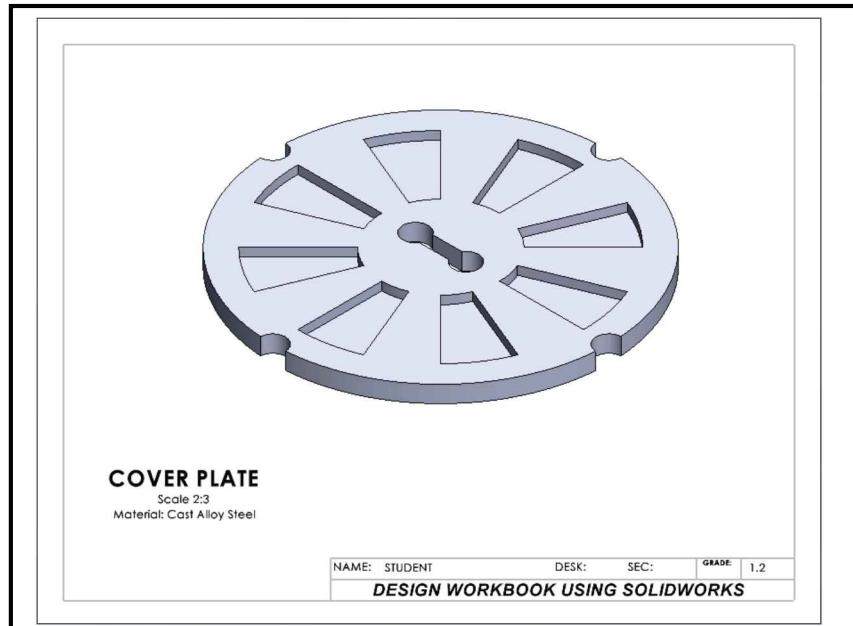


Figure 1-32. The Finished Cover Plate Part.

Exercise 1.3: WALL BRACKET

Up to this point you have used simple sketch geometry like lines, circles, and polygons, which are enough in many cases. In this **Wall Bracket** exercise, you will learn how to draw an irregular curve using a **Spline**.

Start a new part using the **ANSI-INCHES** template and save it as **WALL BRACKET**. Select the **Front** plane in the FeatureManager and click on the **Sketch** command to add a new sketch. The model will be automatically rotated to a **Front** view. Start by drawing the horizontal and vertical lines of the Wall Bracket using the **Line** tool. Refer to **Figure 1-33** for the correct model **Dimensions**. Units are in inches and origin is located in the upper left corner.

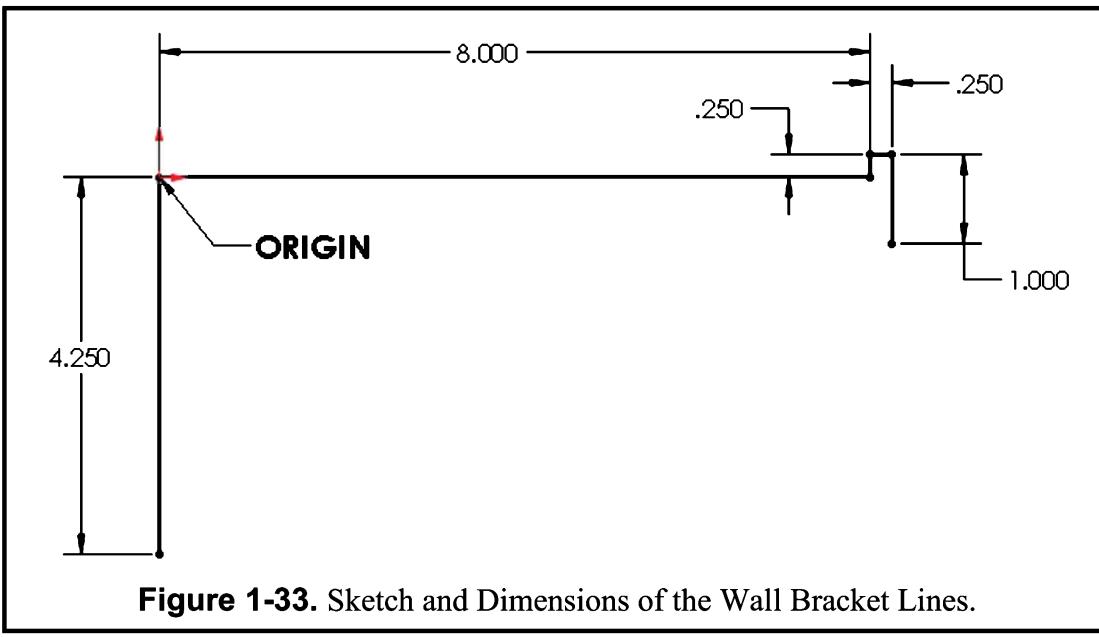


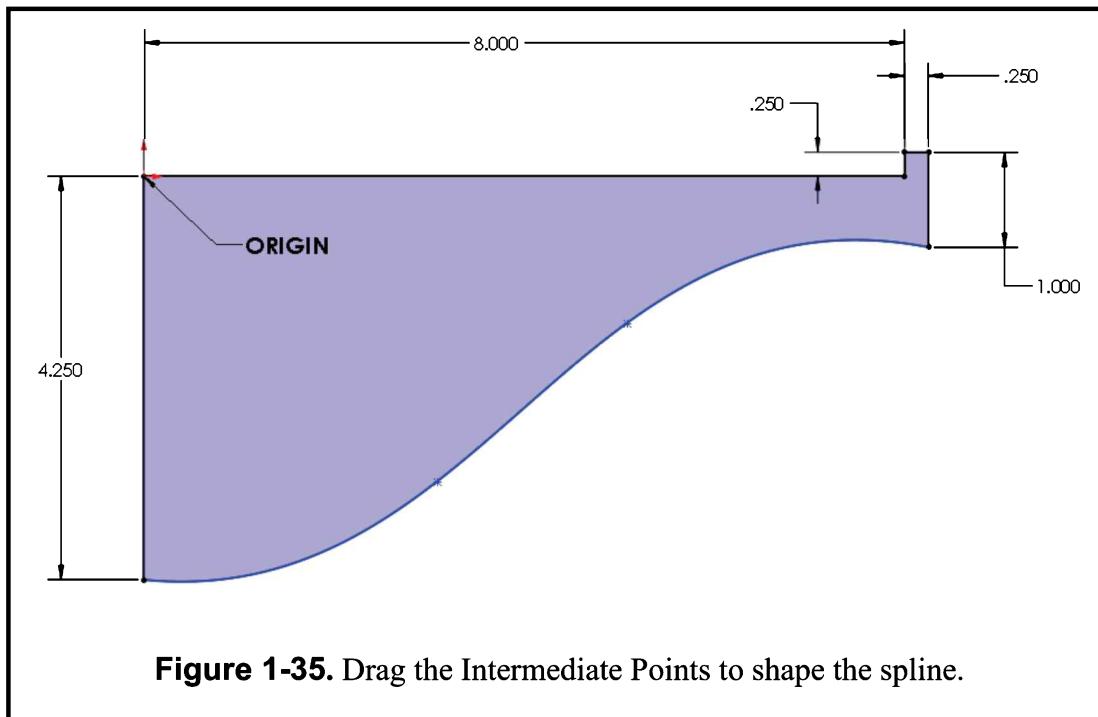
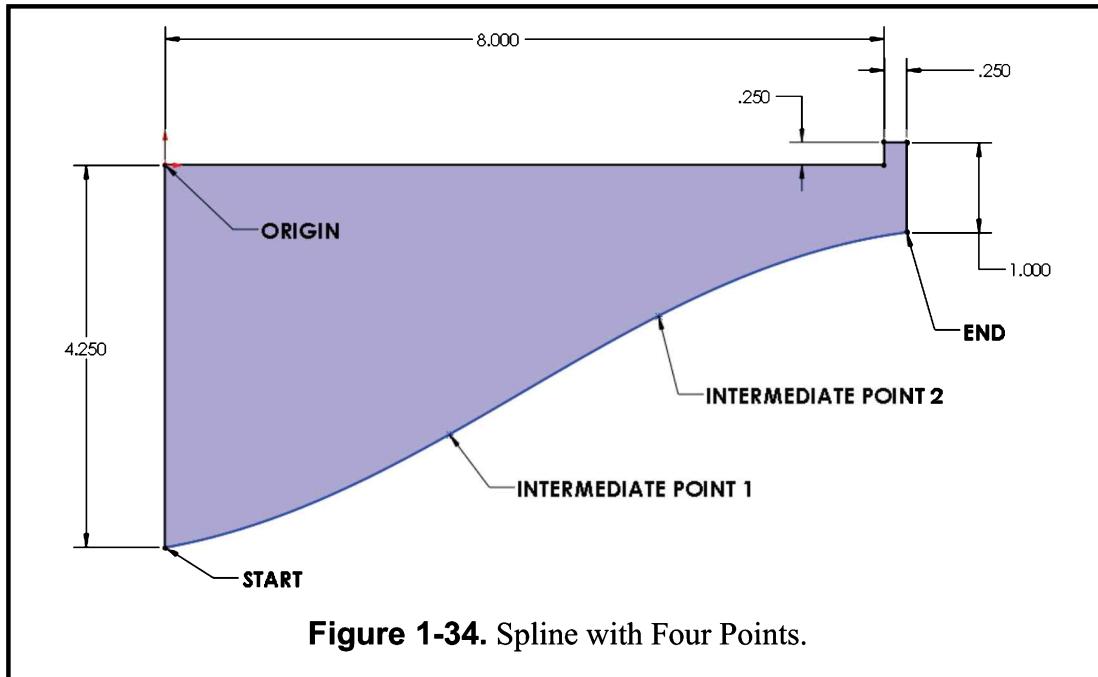
Figure 1-33. Sketch and Dimensions of the Wall Bracket Lines.

 Now you will draw a spline to close the sketch profile. Select the **Spline** command from the Sketch tab, click on the first point at the bottom of the **4.25"** line, click a second time to locate the first intermediate point about one third up and to the right, then click again to locate the second intermediate point approximately two thirds up and to the right. Finally, click on the bottom endpoint of the **1.00"** line. To finish the spline after adding the last spline point, press the **ESC** key.

Note: The spline can also be done using a click-and-drag approach. Click-and-drag with the left mouse button from one point to the next for each spline point; the spline will be finished on the last point. Refer to **Figure 1-34** for the location of the spline points' locations.

The two intermediate points are used to control the shape of the spline. With the **Select** cursor, use the left mouse button to click and drag the intermediate point 1 to create a slight bulge. Next, drag the intermediate point 2 up to create a slight inflection, as seen in **Figure 1-35**.

Note: In this case the spline will be approximated; there are no exact dimensions given for the intermediate points, simply adjust as needed to obtain a shape close to **Figure 1-35**.



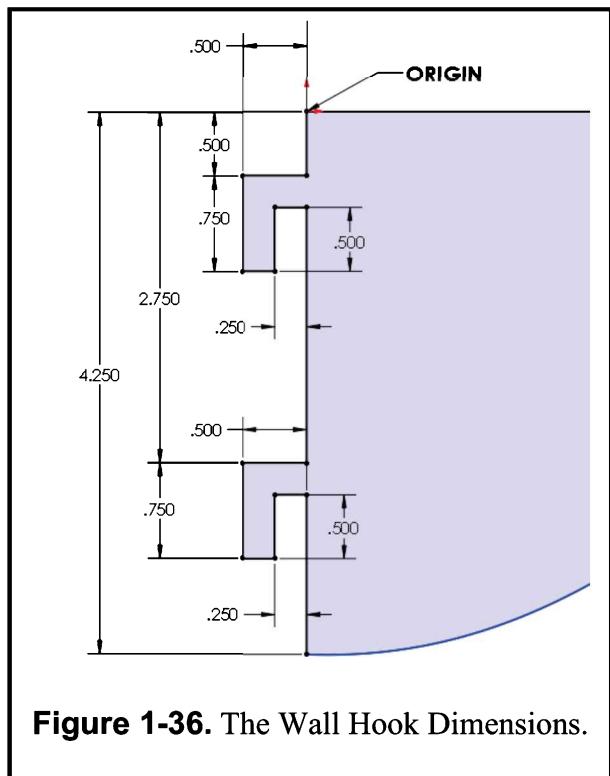
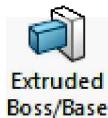


Figure 1-36. The Wall Hook Dimensions.



continue and finish the sketch fillet. The 2D sketch is now complete and ready to be extruded. From the Features tab select **Extruded Boss/Base**, set the end condition to **Blind** with a depth of **0.125"** and click OK to finish. Save the part as **WALL BRACKET**.

Change the part's color to your liking as you did in previous exercises.

Using the instructions from page 1-7, make a new drawing, add a **Trimetric** view with a scale of 1:1 and fill in the missing annotations.

Save your drawing as **WALL BRACKET.slddrw** and **Print** a hard copy for your instructor.

Now you will complete the left side of the Wall Bracket by adding the wall hooks. Add the two hooks using the **Line** tool and **Dimension** as indicated in **Figure 1-36**. You will have to use the **Trim** tool in two places to make the wall hooks contiguous with the rest of the profile.

Note: The bottom wall hook has the same dimensions as the top one.

To finish the Wall Bracket, you need to add a fillet to the sharp bottom corner on the right side. Select the **Sketch Fillet** command and enter a radius of **0.25"**. Adding a fillet to a spline will cause the **1.00"** dimension to be deleted. When adding a fillet to linear edges, the corner is usually constrained to maintain the dimension; that is not the case with a spline. When you select the endpoint to add the fillet, **SOLIDWORKS** will give you a warning letting you know the dimension will be deleted. Click Yes to

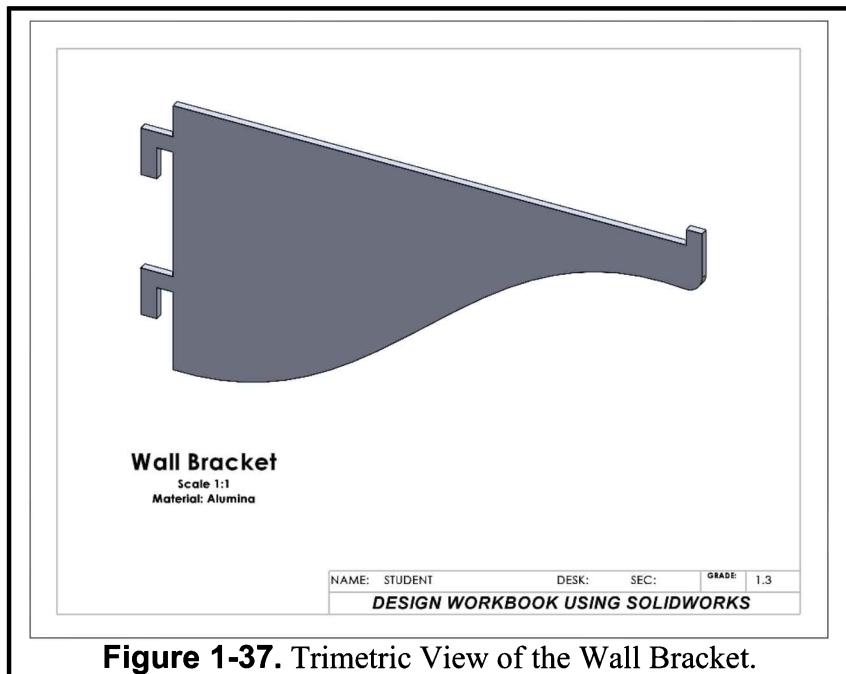


Figure 1-37. Trimetric View of the Wall Bracket.

Exercise 1.4: MACHINE HANDLE

In the previous exercises, the parts were designed using English units (inches). International System (SI), or Metric units, are also common in engineering practice. In Exercise 1.4, the Machine Handle will be designed using millimeters as the base units. Since the machine handle is a cylindrical body, it will be created using a **Revolved Base** feature, by rotating a 2D sketch about an axis of revolution.

Start a new part using the **ANSI-METRIC** part template and save it as **MACHINE HANDLE** to continue. In the FeatureManager select the **Front** plane and add a new sketch. Add a horizontal **Centerline** starting at the origin going to the right. Use the **Line** and **CenterPoint Arc** tools to complete the profile, and dimension as indicated in **Figure 1-38**.

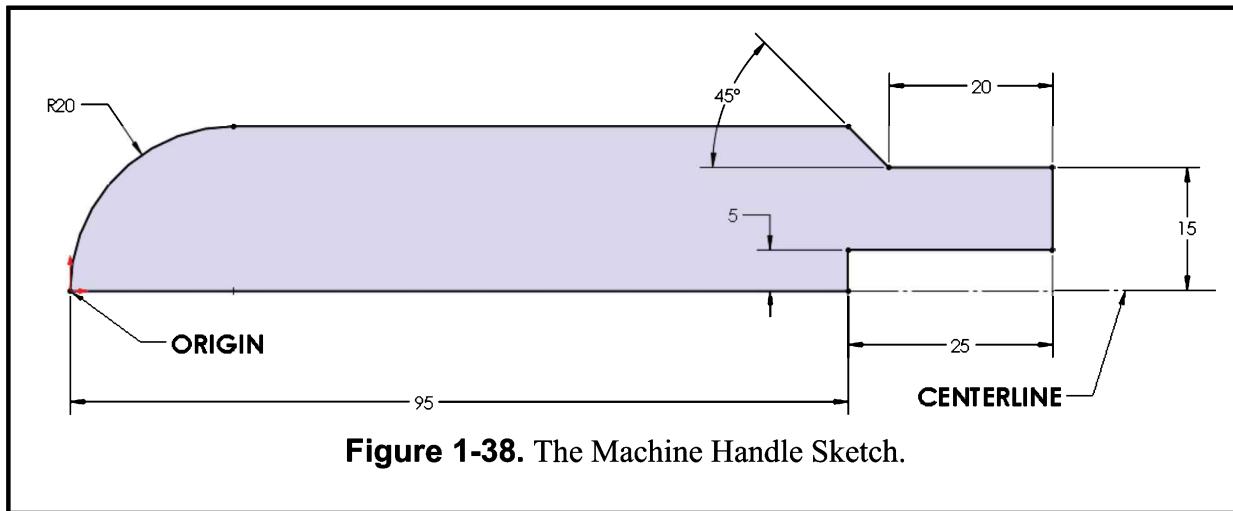
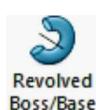


Figure 1-38. The Machine Handle Sketch.



When the sketch is finished you are ready to create the revolved part. From the Features tab select the **Revolved Boss/Base**.

In the PropertyManager, if the sketch has a single **Centerline**, it will be automatically selected as the Axis of Revolution. If the sketch has multiple centerlines, you will need to select which one will be the axis of revolution.

The other option to define is how many degrees to revolve the sketch about the axis of revolution. Use the default of **360°** to make a full revolution, as shown in **Figure 1-39**, and click OK to finish. Your part's Isometric view will look like **Figure 1-40**.

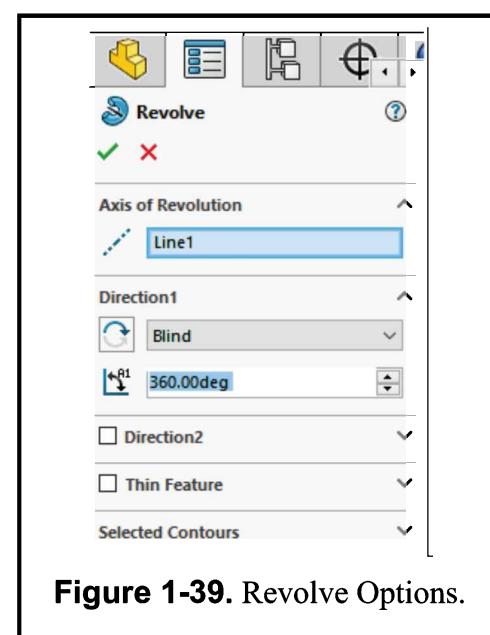
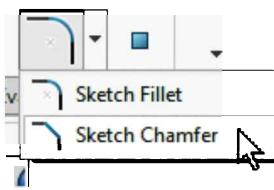


Figure 1-39. Revolve Options.

There are many different designs for a Machine Handle, and the one here is quite typical. After reviewing it, you notice that the sharp edge on the right side poses a safety risk. SOLIDWORKS allows you to easily make changes after creating the solid model. One way is to go back and edit the original sketch.

In the FeatureManager, SOLIDWORKS maintains a chronological list of the features used to model a part. Your first feature is “**Revolve1**,” which is the operation you just completed. On the left of it you can see a right arrow. Click on this arrow to reveal the sketch used to create it, in this case “**Sketch1**.”

Selecting **Revolve1** or **Sketch1** with the left mouse button will show a context menu with common operations. If you right click instead, you will also reveal a context menu with expanded options, as shown in **Figure 1-41**. Select the **Edit Sketch** command to modify the original sketch. If needed, change the view orientation to a **Front** view to better see the sketch.



To eliminate the sharp edge, we are going to use the **Sketch Chamfer** command, which is nested in the **Sketch Fillet** icon. Click in the down arrow next to the **Sketch Fillet** and select the **Sketch Chamfer** command.

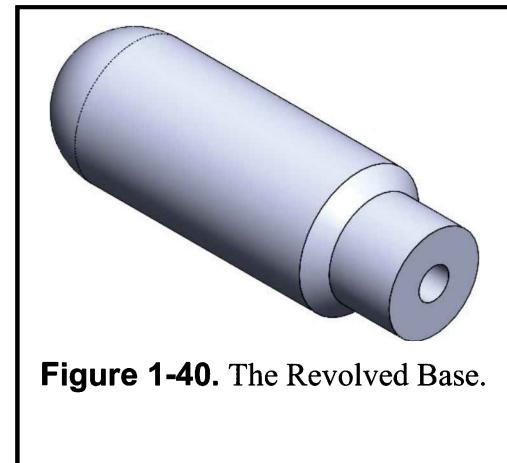


Figure 1-40. The Revolved Base.

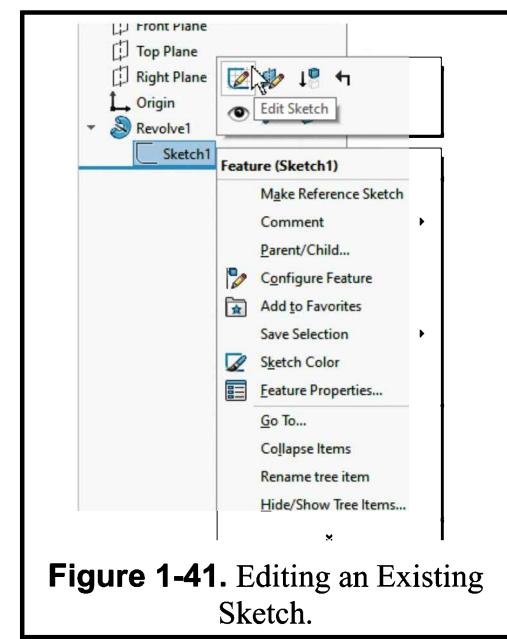


Figure 1-41. Editing an Existing Sketch.

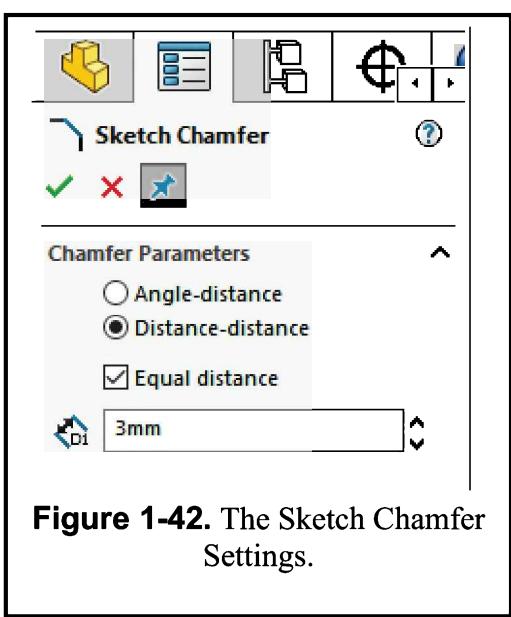


Figure 1-42. The Sketch Chamfer Settings.

In the **Sketch Chamfer** PropertyManager select the Distance-Distance chamfer and turn on the Equal distance option and set the distance to 3mm as indicated in **Figure 1-42**. Now click on the endpoint in the corner, or the two lines that form the sharp edge, and click OK to finish.

After the chamfer is added to the sketch, you need to rebuild the model to update the 3D model. Select the menu **Edit, Rebuild** or the Rebuild command from the main toolbar.

With this example you can see how easy it is to modify the sketch of a 3D solid model by simply editing its sketch. The updated model with the chamfer is shown in **Figure 1-43**.

If you would like to change the color of your model, right click on the model name in the FeatureManager tree and select the Appearances command from the context toolbar, to assign a different color to the model.

Save your part as **MACHINE HANDLE.sldprt**, make a new drawing using the **ANSI-METRIC** drawing template and add an **Isometric** view.

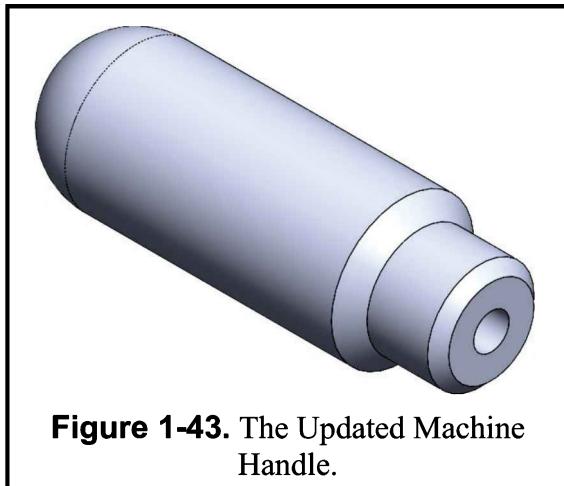


Figure 1-43. The Updated Machine Handle.

Make the **Isometric** view's scale 3:2 and add the missing annotations as shown in **Figure 1-44**, including a note with the Exercise number (**1.4**) in the upper right-hand box of the Title Block. Save your drawing to your personal folder as **MACHINE HANDLE**.

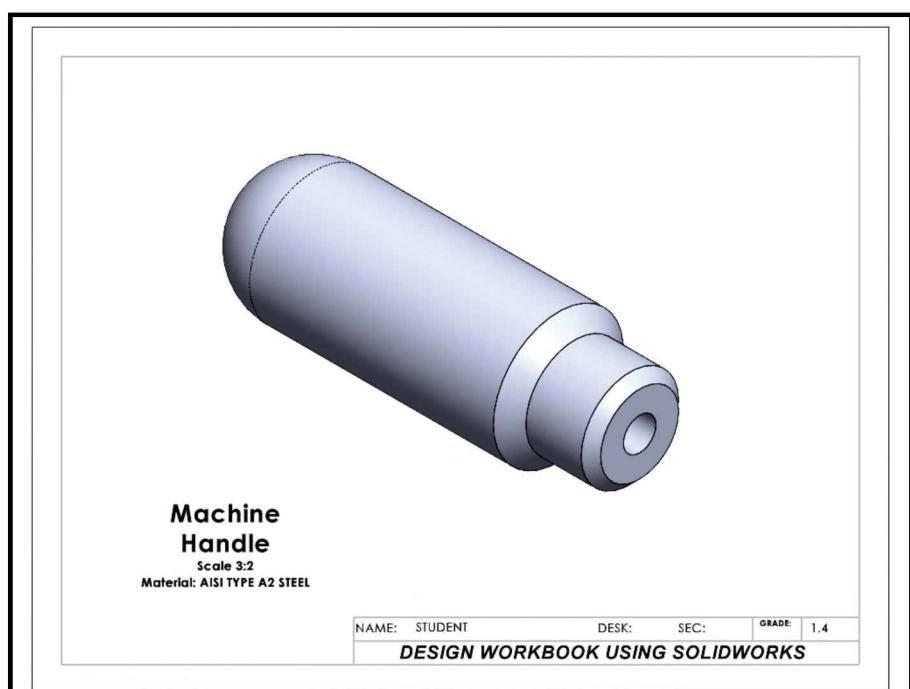
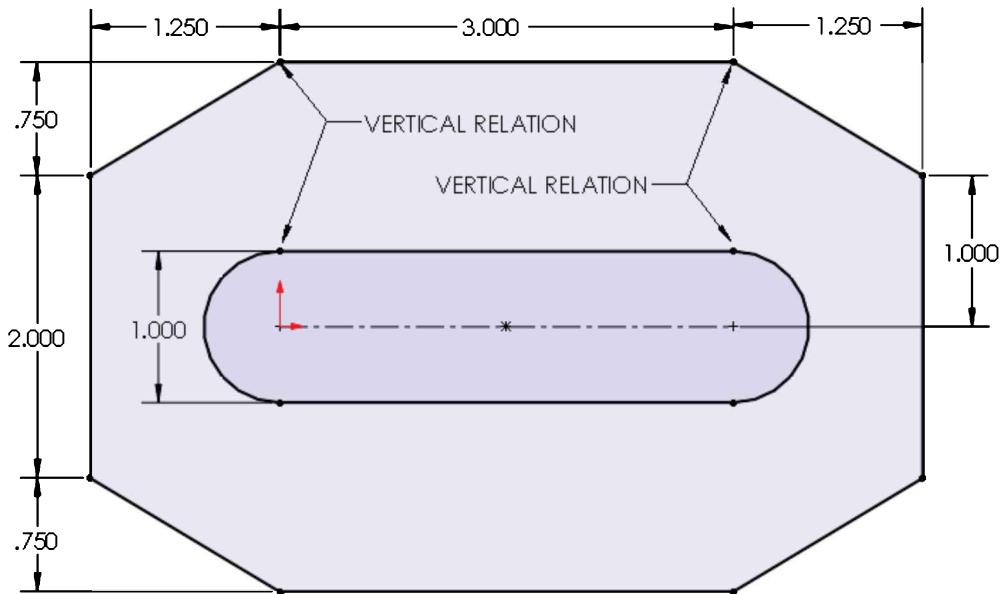


Figure 1-44. The Machine Handle Isometric view.

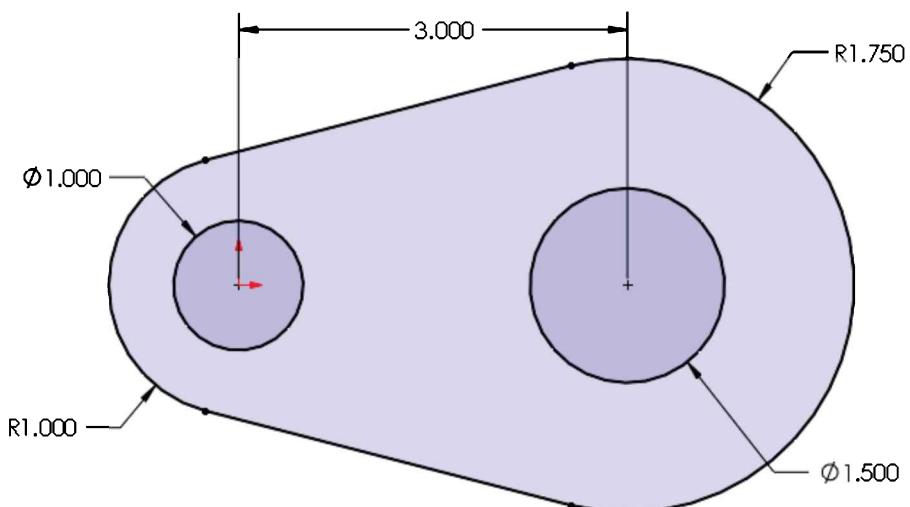
Supplementary Exercise 1-5: SLOTTED BASE

Make the sketch shown below in the **Top Plane** and extrude it **0.50** inches. Make a new drawing with an **Isometric** view and save it as “**SLOTTED BASE**.” To fully define your sketch, you need to add the necessary geometric relations. Make the horizontal lines **Equal**, the vertical lines **Equal**, and the angled lines **Equal**, and make the indicated endpoints **Vertical**.



Supplementary Exercise 1-6: TRANSITION LINK

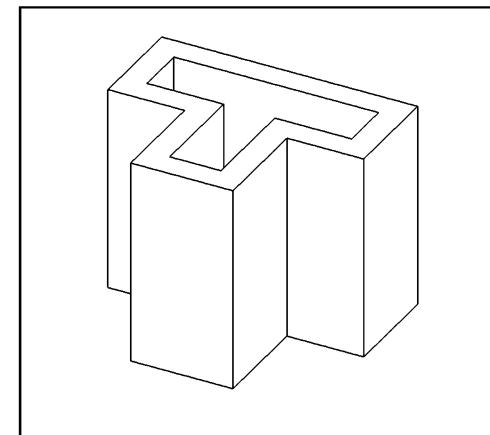
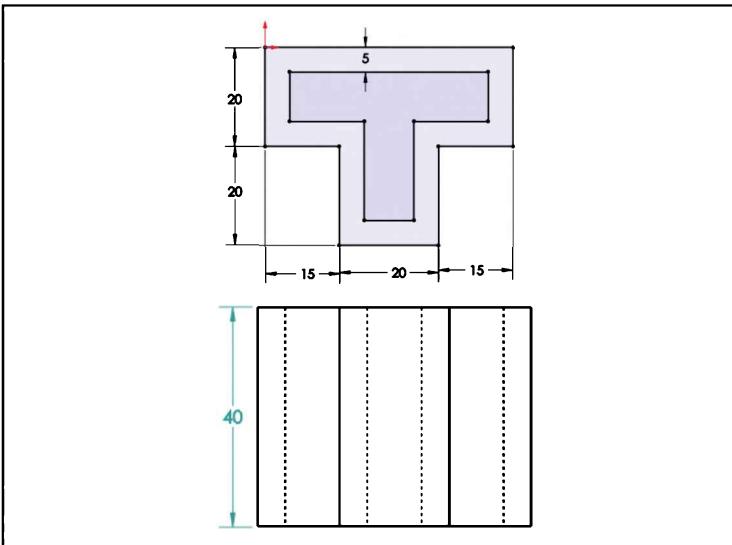
Make a sketch of the figure below in the **Front Plane** and extrude it **0.375"**. Make a new drawing with an **Isometric** view and name it “**TRANSITION LINK**.”



Supplementary Exercise 1-7: TEE BRACKET

Make a sketch of the figure below in the **Top Plane** using the **ANSI-METRIC** template and extrude it **40 mm** as indicated. Make a new drawing, add a **Trimetric** view in it and save it as **TEE BRACKET**.

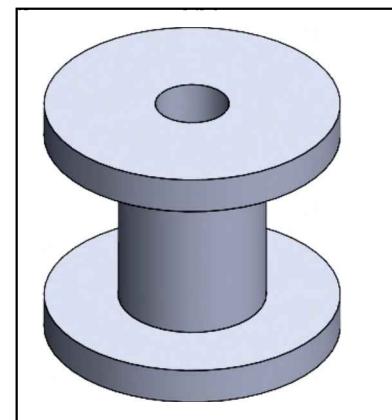
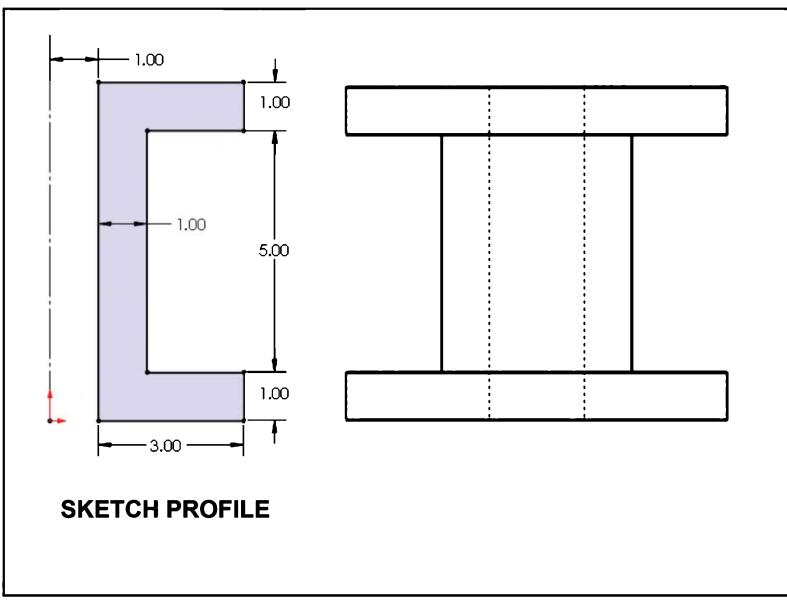
TIP: Draw the outside perimeter and use the **Offset Entities** command from the Sketch tab to draw the inside profile. Select the perimeter and set the offset distance to 5mm.



TRIMETRIC VIEW OF THE
TEE BRACKET

Supplementary Exercise 1-8: CABLE SPOOL

Make a sketch in the **Front Plane** with the profile shown below and make a **Revolved Base**. Remember to add a centerline to revolve about. Make a new drawing with an Isometric view and save it as **CABLE SPOOL**.



ISOMETRIC VIEW OF THE
CABLE SPOOL

NOTES:

Design Workbook Lab 2: Advanced 2D Sketching

ADVANCED 2D SKETCHING

In the first Computer Graphics Lab, you learned some of the basic 2D sketch tools available in SOLIDWORKS. The first exercises reinforced how to draw a **Line**, **Circle**, **Rectangle**, **Arc**, **Polygon**, **Centerline**, and **Spline**. You also learned how to modify a 2D sketch using **Dimensions**, the **Trim**, **Mirror**, **Fillet**, and **Chamfer** commands. In this lesson you will learn more advanced 2D sketch commands available in SOLIDWORKS.

SKETCH ENTITIES MENU

The complete list of sketch drawing tools available is listed in the menu **Tools**, **Sketch Entities**, as shown in **Figure 2-1**, and most of them have additional options available to help you define your design's geometry, depending on your design intent. The commands more commonly used are included in the default sketch toolbar. The complete list of sketch entities available include:

Line

Rectangle (multiple definition options)

Parallelogram

Slot (multiple definition options)

Polygon

Circle

Perimeter Circle

Centerpoint Arc

Tangent Arc

3 Point Arc

Ellipse (multiple definition options)

Partial Ellipse

Parabola

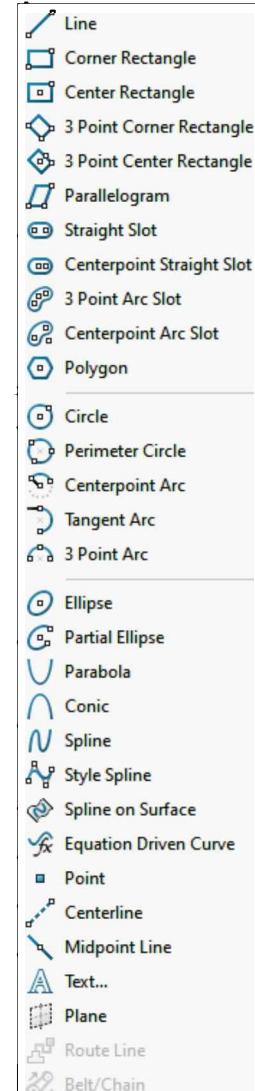
Spline

Spline on Surface

Point

Centerline

Text



While some of these 2D sketch entities are more commonly used than others, you will have a chance to use most of them while completing your exercises.

Figure 2-1. The Sketch Entities Menu.

SKETCH TOOLS MENU

All of the 2D sketch editing tools are included in the menu **Tools, Sketch Tools**, and just like sketch entities, the most commonly used are included in the default Sketch toolbar. The list of tools includes:

Fillet is used to round a corner with a radius.

Chamfer is used to cut a corner at an angle.

Offset Entities is used to create a copy of a sketch entity at an offset distance from the original.

Convert Entities projects a 3D feature's edge as a sketch entity in the current sketch.

Trim cuts away a piece of a sketch entity.

Extend extends an entity to meet another entity.

Mirror copies a sketch element about a selected mirror line.

Dynamic Mirror dynamically mirrors new sketch geometry about a centerline. First select the entity about which to mirror, activate the Dynamic Mirror command, and draw the sketch entities to be mirrored.

Jog Line moves a piece of the line up or down in a rectangular shape.

Construction Geometry converts sketch entities to construction geometry and vice versa.

Linear Sketch Pattern creates a linear or rectangular pattern of identical entities (see **Figure 2-3**).

Circular Sketch Pattern creates a radial (or polar) pattern of identical entities around a center point (see **Figure 2-4**).

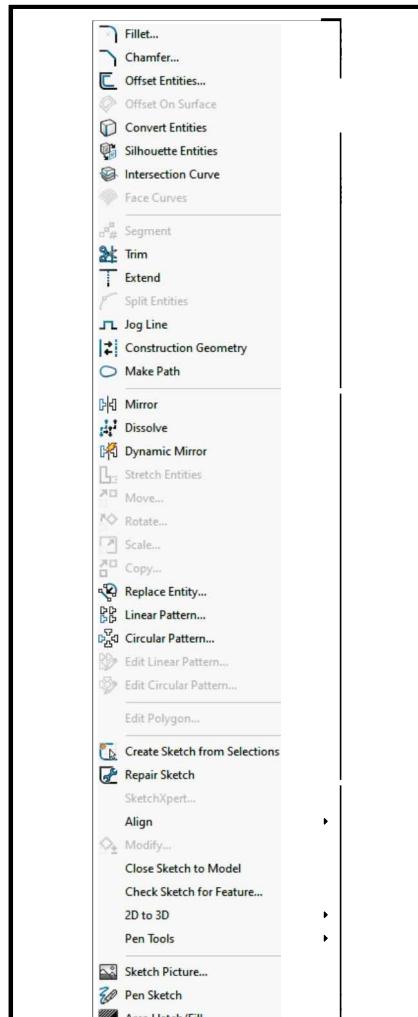


Figure 2-2. The Sketch Tools Menu.

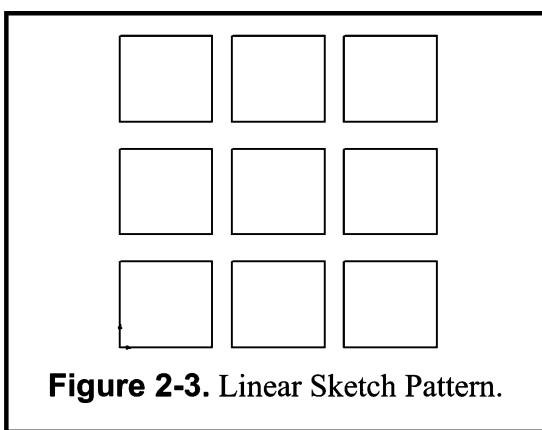


Figure 2-3. Linear Sketch Pattern.

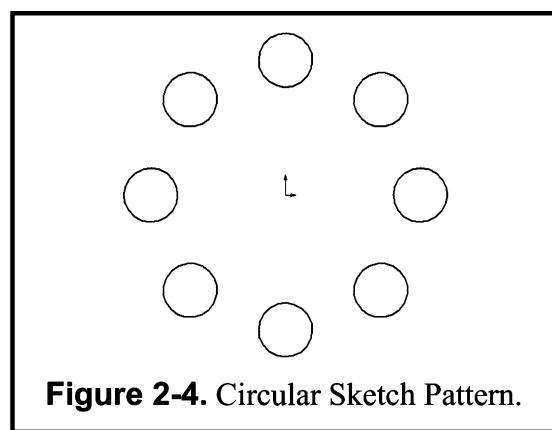
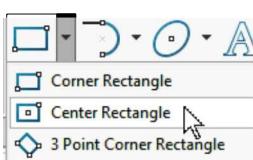


Figure 2-4. Circular Sketch Pattern.

Exercise 2.1: METAL GRATE

In Exercise 2.1, you will design a Metal Grate. A grate usually has multiple identical slots, but instead of adding them individually, you can draw one slot, and use the **Rectangular Pattern** command to copy the rest. After the sketch is complete you can extrude it to build the base feature.

To make the grate, create a new part using the **ANSI-METRIC** template. Select the **Front** plane in the FeatureManager and then click on the **Sketch** command from the Sketch Tab.



Since the grate is symmetrical about the X and Y directions, it is to your advantage to locate the part centered in the **Origin**. To do this, select the **Rectangle's** drop-down list and click on **Center Rectangle**.

Draw the rectangle starting at the **origin** and dimension it **280 mm** wide by **195 mm** high. Next, select the **Rectangle** tool, draw the initial rectangular slot and dimension as shown in **Figure 2-5**. Once the rectangles are dimensioned, your sketch will be fully defined (geometry will be black). Use the **Sketch Fillet** command to add a **3 mm** fillet to the corners of the small rectangle.

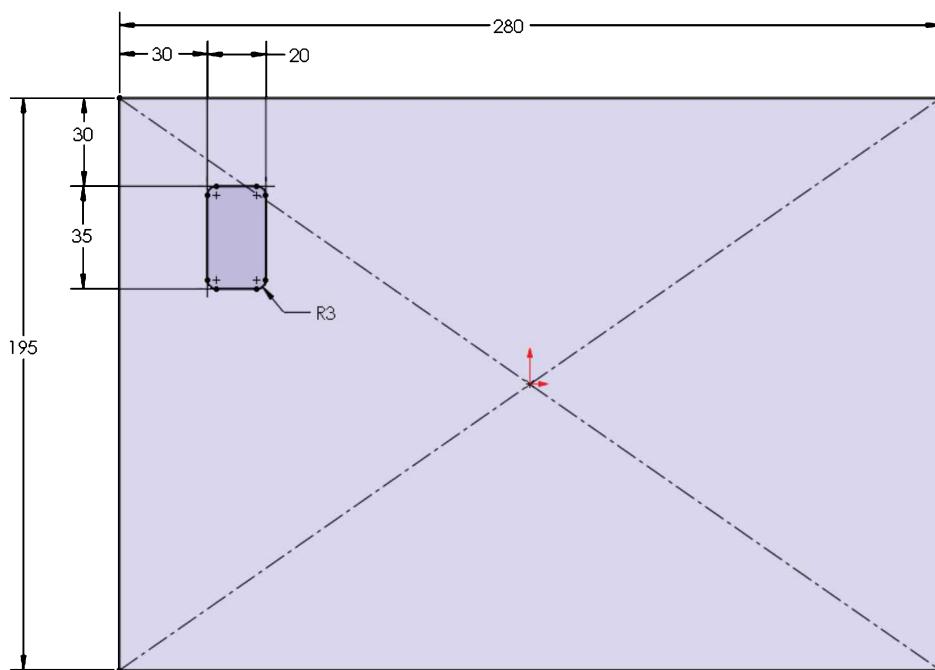
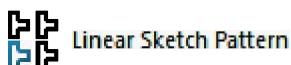


Figure 2-5. Initial Sketch for the Metal Grate Centered at the Origin.



Now select the **Linear Sketch Pattern** command from the Sketch toolbar, or from the menu **Tools, Sketch Tools, Linear Pattern**. When the Linear Pattern PropertyManager is displayed, activate the Entities to Pattern selection box and select the lines and fillets of the small rectangle.

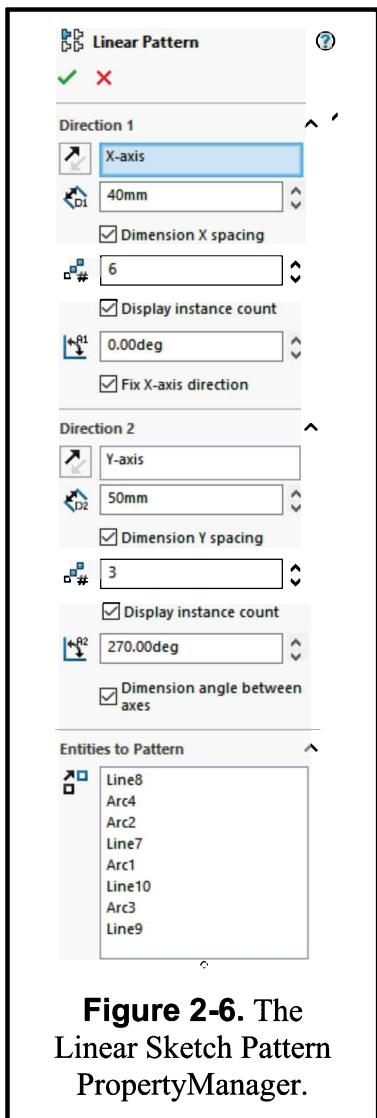


Figure 2-6. The Linear Sketch Pattern PropertyManager.

The settings for the grate's rectangular pattern are shown in **Figure 2-6**. “Direction 1” is horizontal and will have **6** instances spaced **40** mm at **0** degrees. To activate the “Direction 2” options, you have to change the number of instances to **3** and set the vertical spacing to **50** mm. The angle will be automatically set to **270** degrees.

As you change the pattern parameters, notice the **Preview** is shown. For Direction 2 you will have to select the Reverse Direction button for the pattern to go down. Your pattern preview should look like the image in **Figure 2-7**. When finished, click the **OK** button to complete the pattern.

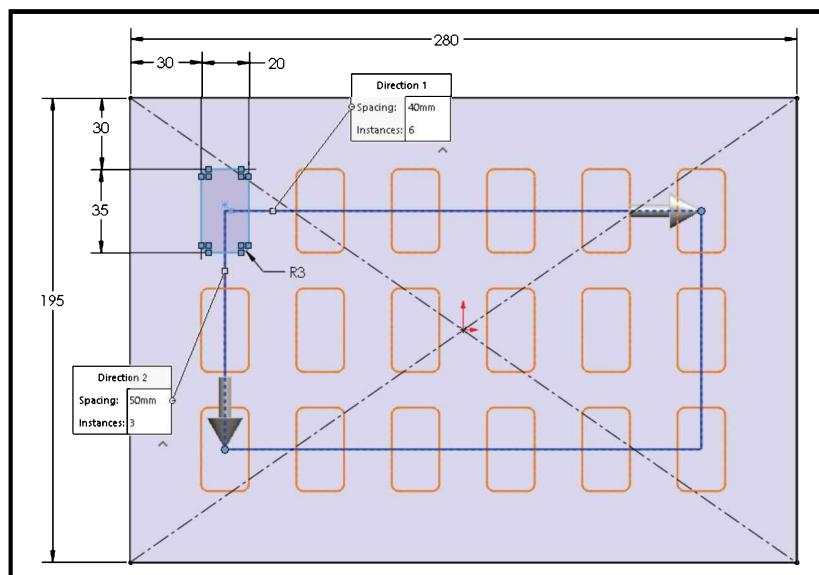


Figure 2-7. Linear Sketch Pattern Preview.

To fully define the sketch, you will have to activate the option Dimension angle between axes, or add a Vertical relation to the centerline connecting the first rectangle to the first vertical pattern copy.

From the Features Tab select **Extruded Boss**, set the end condition as **Blind**, make the extrusion **5 mm** and click **OK** to finish. Your grate will look as **Figure 2-8**.

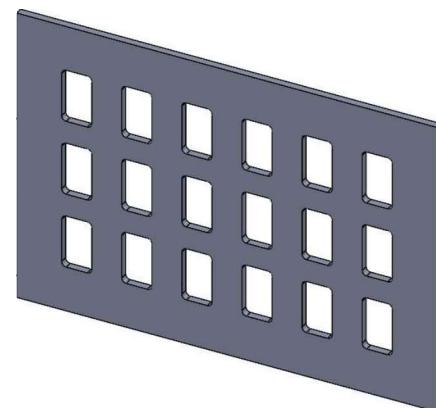


Figure 2-8. Grate's Base feature.

The next step is to add a lip to the metal grate in order to provide a support when it is attached. Change to a **Front** view, select the front face of the Metal Grate and add a new sketch.



To create the lip's sketch, you will leverage the previously made geometry. With the front face pre-selected, click in the **Convert Entities** command from the sketch tab. The outer edges of the face will be projected as sketch entities into the active sketch.

Notice the lines are all black; each projected entity is automatically added an **On Edge** geometric relation to the projected edges. This means that if the previous geometry is changed, the projected sketch entities will update accordingly.



To create the smaller internal rectangle, select the **Offset Entities** command. Click to select one of the previously projected entities, set the offset distance to **15 mm** and make sure the **Select Chain** option box is selected; this way all connected entities will be automatically selected. If the offset preview is outside, select the **Reverse** option to create the offset inside and click **OK** to finish.

The offset command adds a single 15 mm dimension, letting you know the offset value. Also, each entity is added an **Offset** geometric relation to fully define the sketch. Add a **5mm** Fillet to the inside corners of the offset rectangle as shown in **Figure 2-10**.

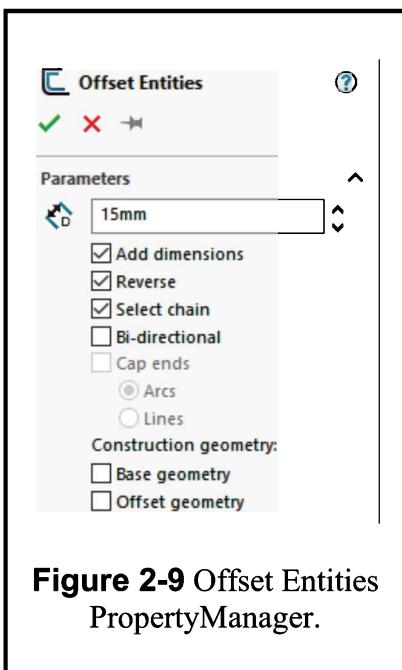


Figure 2-9 Offset Entities PropertyManager.

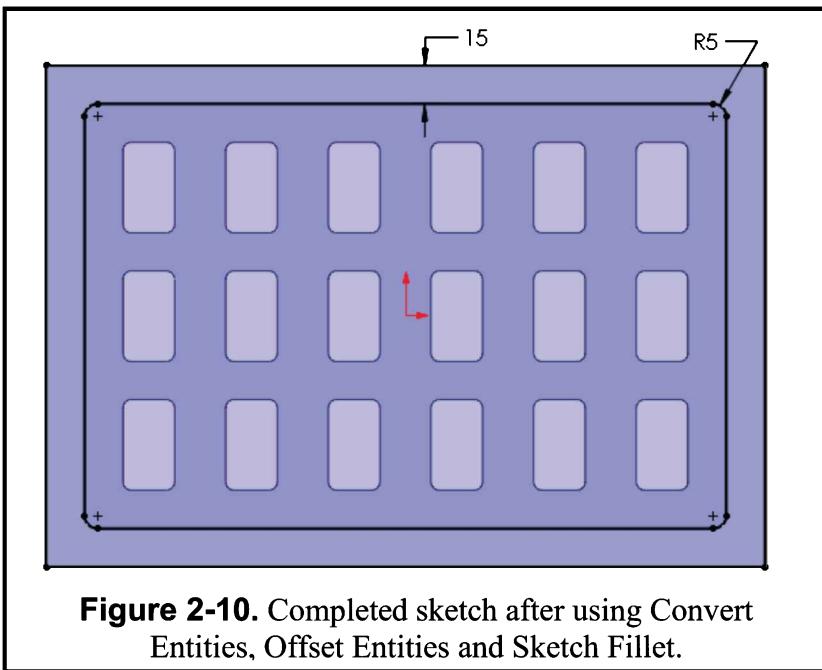


Figure 2-10. Completed sketch after using Convert Entities, Offset Entities and Sketch Fillet.

Before extruding the sketch, it is a good idea to change your view orientation to an **Isometric** view; this way it will be easier to see which direction the extrusion is going. Select the **Extruded Boss** command from the Features tab, set the end condition to **Blind** and change the extrusion depth to **5 mm**. Click **OK** to complete the boss.

Now you need to add the four attachment holes to the corners of the grate. Select the face of the last extrusion and add a new sketch on it. Draw an **8 mm** diameter circle in the upper left corner of the grate, and dimension it **9 mm** from the side and **9 mm** from the top edge. To better maintain the design intent, you'll use the **Sketch Mirror** command to create the other three circles. Starting in the origin, draw a vertical and a horizontal centerline, as seen in **Figure 2-11**.



From the Sketch toolbar select the **Mirror Entities** command. In the “Entities to Mirror” selection box select the circle, in the “Mirror about” selection box select the vertical centerline, and click OK to finish. Next use the **Mirror Entities** again; now select the two circles and mirror them about the horizontal centerline.



Message
Select entities to mirror and a sketch line, linear model edge, plane or planar face to mirror about

Options

Entities to mirror:



Arc1

Copy

Mirror about:



Line4

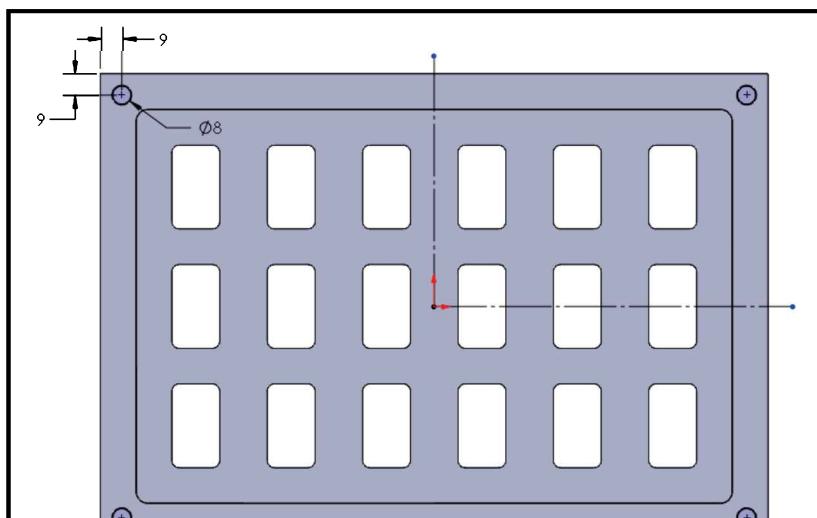


Figure 2-11. Small Hole Dimension and Location.

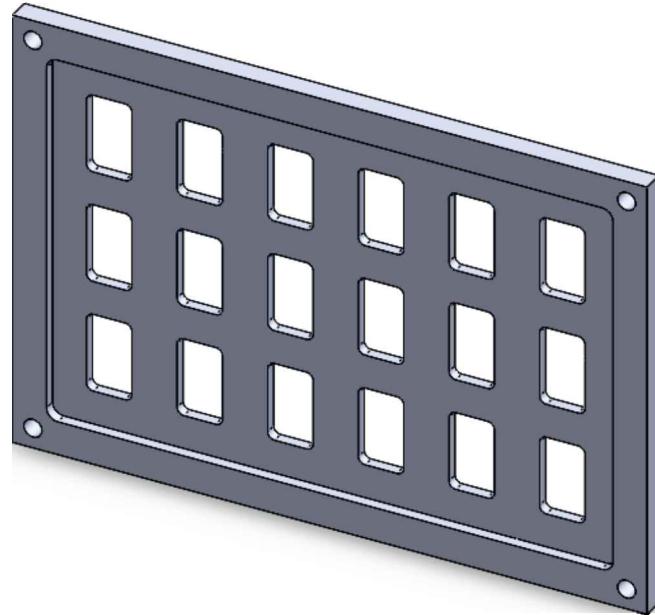


Figure 2-12. The Finished Metal Grate.

Change the model view's orientation to an Isometric. From the **Features** tab select **Extruded Cut**, set the End Condition to **Through all** and click OK to finish. The four corner attachment holes are now created on the grate.

The part is now complete, and you can view the lip feature more clearly by rotating the model, as shown in **Figure 2-12**.

If you'd like to, change the color of your model, and save your part as **METAL GRATE.sldprt**. Make a new drawing, add a **Trimetric** view, change the view's scale to 2:3 and complete the missing notes as before, including the exercise number, as shown in **Figure 2-13**.

Save your drawing as **METAL GRATE.slddrw** and Print a copy to submit to your lab instructor.

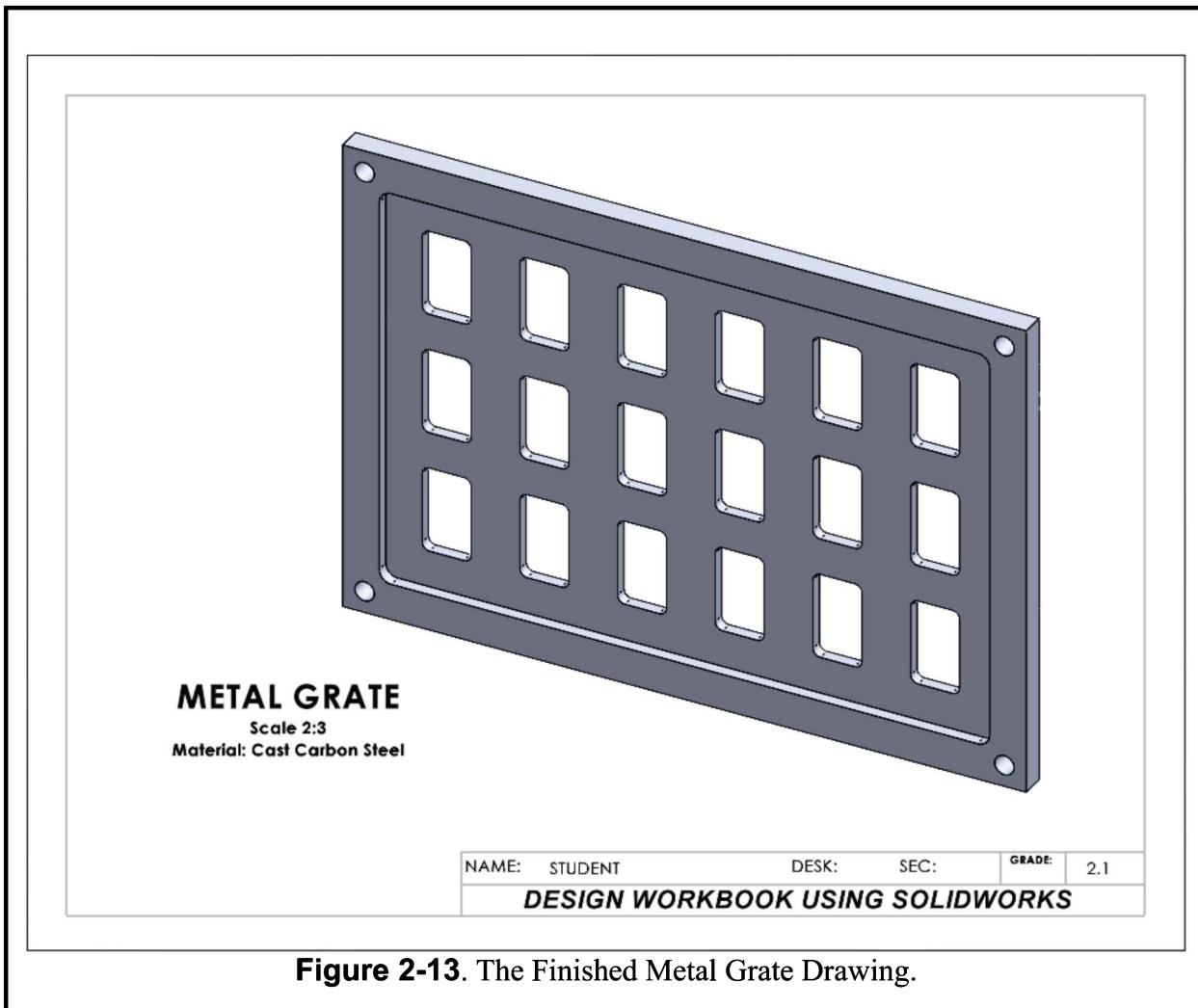


Figure 2-13. The Finished Metal Grate Drawing.

Exercise 2.2: TORQUE SENSOR

In Exercise 2.2 you will design a Torque Sensor casing. Since it is a cylindrical (axis-symmetrical) object, it will be created using a **Revolved Base** feature, and certain features will need to be made using a **Circular Pattern** to complete the part.

Start a new part using the **ANSI-INCHES** template as before and save it as **TORQUE SENSOR.sldprt**.

The first feature you will make is the revolved base. Select the **Front** plane and add a new sketch. Add a vertical centerline; draw and dimension the closed profile shown in **Figure 2-14**.

To fully define the sketch, you have to add a few geometric relations.



Coincident Select the bottom horizontal line and the origin and add a **Coincident** relation.



Collinear Select the two left short vertical lines and add a **Collinear** relation.



Equal Select the two right short vertical lines and add a **Collinear** and **Equal** relation.



From the Features tab, select the **Revolved Boss** command. Make sure the centerline is selected as the axis of revolution, enter 360° to make a full revolution and click OK to finish.

The resulting cylindrical part will have a **4.00"** outside diameter as shown in **Figure 2-15**.

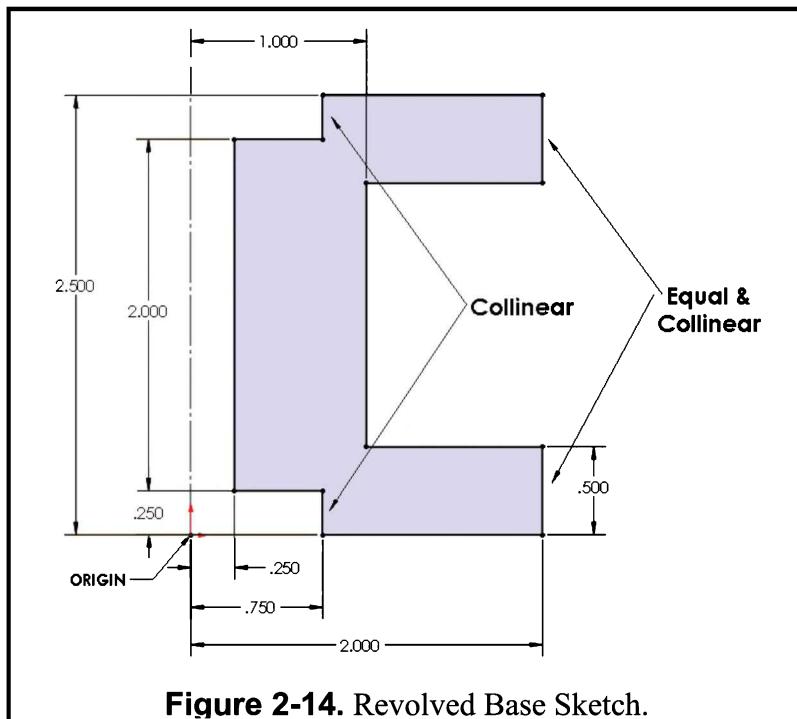


Figure 2-14. Revolved Base Sketch.

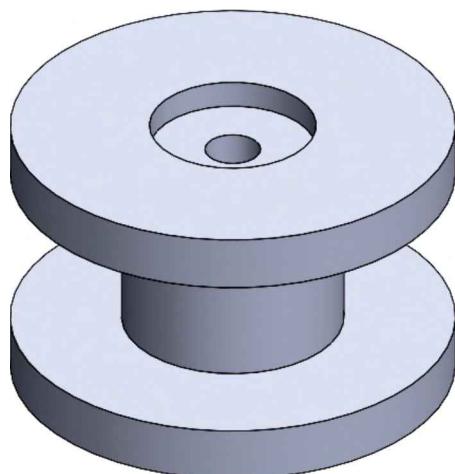


Figure 2-15. Revolved Base Feature.

The next step is to add a circular pattern of holes at the top of the part. Change to a **Top** view orientation, select the top face of the part and add a new sketch.

To locate the first hole, draw a **Circle** centered in the origin, and dimension it **3.25"**.

Options
 For construction
 Select the circle and activate the “**For construction**” option in the PropertyManager to change the circle into construction geometry. Now draw the first circle of the pattern horizontal to the origin at the right quadrant of the construction circle. By doing this you will automatically capture a **Coincident** relation. Finally dimension it **0.25"** diameter, as shown in **Figure 2-16**.

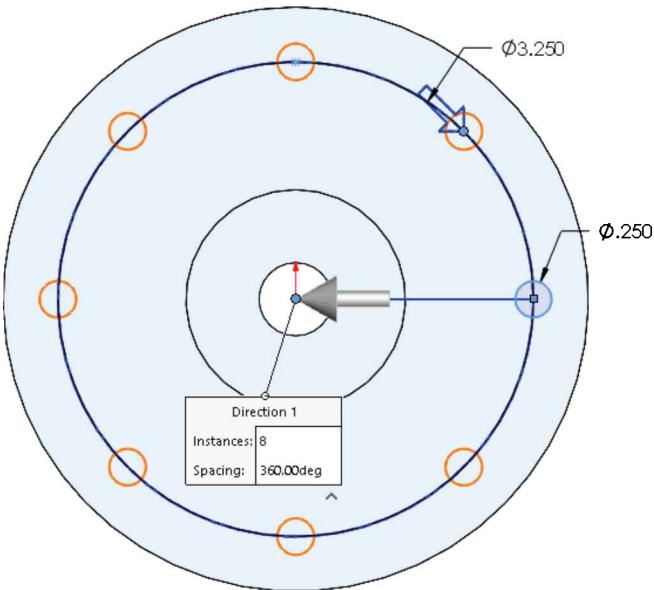
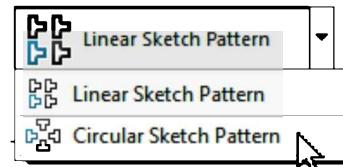


Figure 2-16. First Circle and Circular Pattern.



From the drop-down menu in the **Linear Sketch Pattern** select the **Circular Sketch Pattern** command.

In the Circular Pattern options activate the Pattern Center selection box and select the origin. Activate the options “**Equal spacing**” and “**Dimension radius**” and set the angle to **360°**.

Change the number of instances to **eight (8)**, activate the “**Entities to Pattern**” selection box and select the circle to be patterned. When finished entering the selections click **OK** to continue. The circular pattern options are shown in **Figure 2-17**.

To fully define the sketch, add a **Horizontal** relation to the horizontal line added by the pattern starting in the origin up to the first hole.

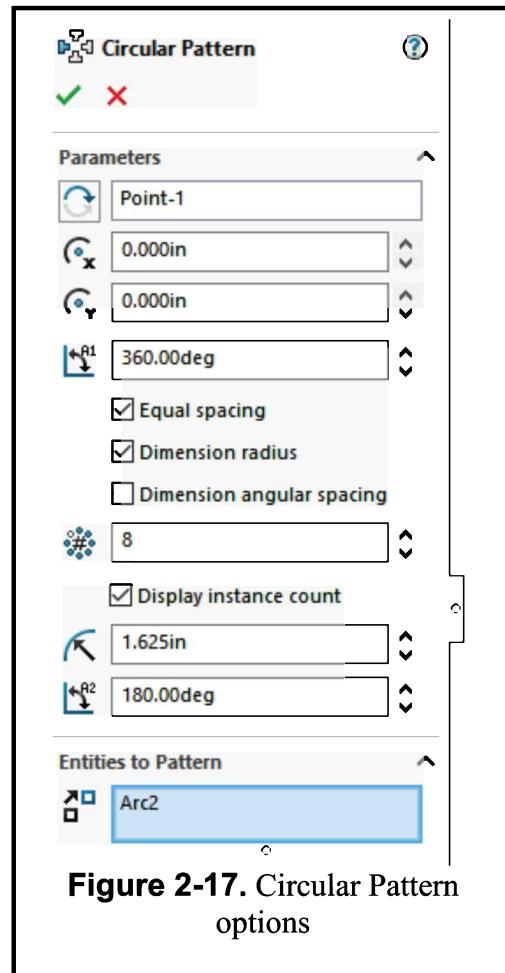
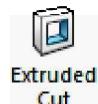


Figure 2-17. Circular Pattern options



When your sketch is complete, select the **Extruded Cut** command from the Features tab. Change to an **Isometric** view to visualize the result, set the end condition to **Through all** and click **OK** to finish. The model is complete as shown in **Figure 2-18**.

When finished, make sure your part is saved as **TORQUE-SENSOR.sldprt**. Follow the instructions on Page 1-7 to make a new drawing using the **TITLEBLOCK-INCHES.drwdot** template and save it as **TORQUE SENSOR.slddrw**.

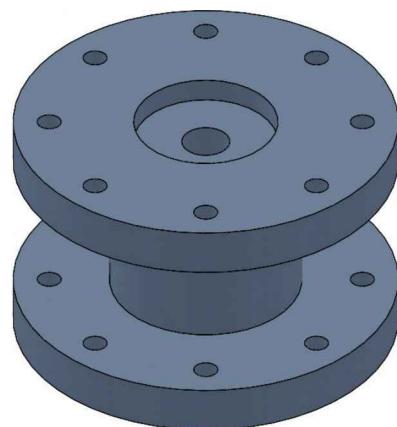


Figure 2-18. The Finished Torque Sensor.

Add an **Isometric** view using the **Shaded with Edges** display mode and change the view's scale to 1:1. Remember to add the missing annotations to the drawing including the exercise number and print a copy for your lab instructor (see **Figure 2-19**).

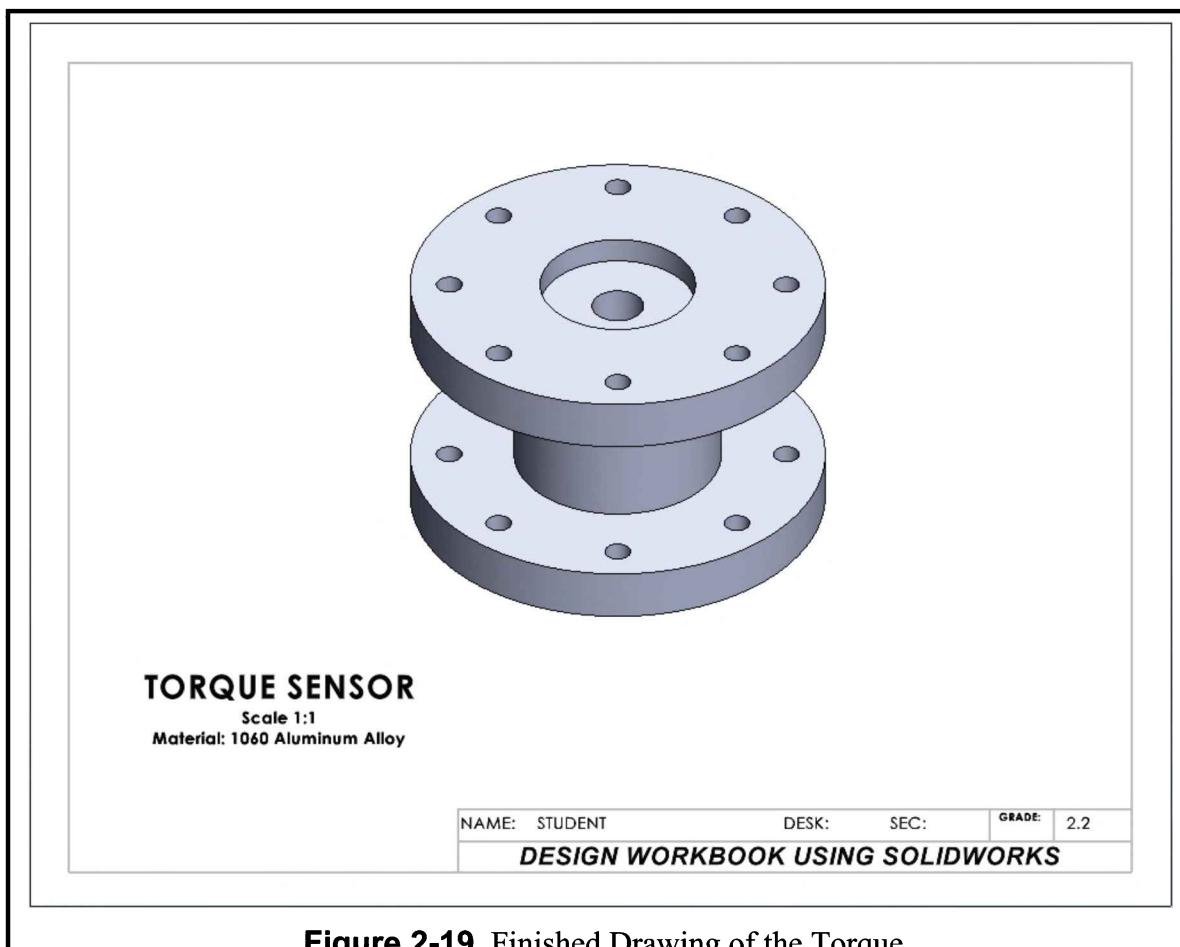
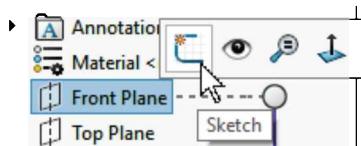


Figure 2-19. Finished Drawing of the Torque.

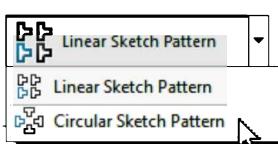
Exercise 2.3: SCALLOPED KNOB

In Exercise 2.3, you will design a Scalloped Knob that has some complicated geometry around its edges. This particular knob design will be a hexagon type. Since the hexagonal features are equally spaced around the center of the knob, you will use a circular pattern to maintain the design intent.

Start by making a new part using the **ANSI-INCHES** template and save your part as **SCALLOPED KNOB.sldprt**.



Select the **Front** plane in the FeatureManager and add a new sketch. You can select the **Sketch** icon from the pop-up toolbar immediately after selecting the plane. Using the **Line** and **Sketch Fillet** tools, draw and dimension the profile to fully define the sketch, as shown in **Figure 2-20**.



From the **Linear Sketch Pattern** drop-down menu select the **Circular Sketch Pattern** command.

Activate the “**Pattern Center**” selection box and click in the Origin. Set the number of copies to **6** with a Total Angle of **360°**; activate the “**Equal Spacing**” option.

In the “**Entities to Pattern**” selection box add the three lines and the two fillets and click **OK** to continue. Your pattern will look like **Figure 2-21**.

After the circular pattern is complete you will notice that some of the lines may overlap. The next step is to add the sketch fillets. By adding a fillet between the intersecting lines, a fillet will be created, and the lines will be trimmed at the same time.

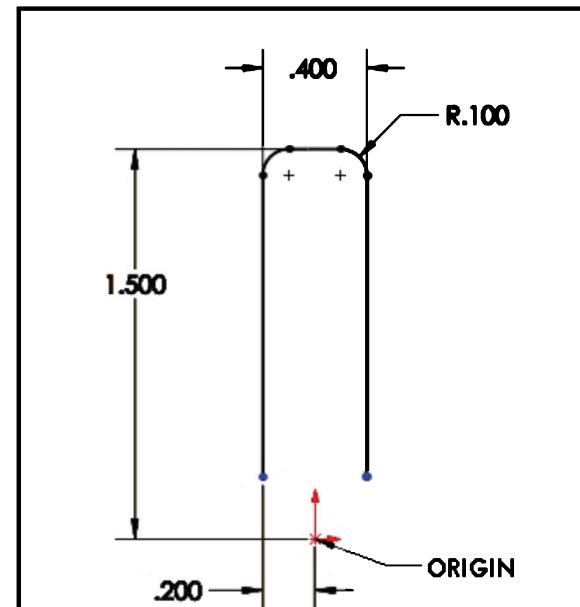


Figure 2-20. The Initial Knob Sketch.

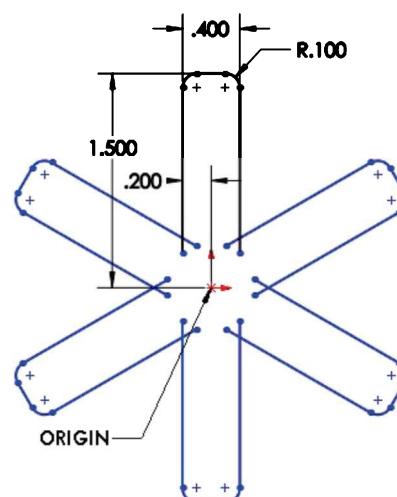
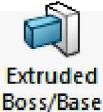


Figure 2-21. The Sketch after completing the Circular Pattern.

 Select the **Sketch Fillet** command and set the radius to **0.45"** in the PropertyManager. Now select two intersecting lines to preview the fillet. Select the rest of the intersecting pairs of lines and click **OK** to finish. A single **0.45"** radius dimension will be added. Remember that all fillets created at the same time have an **Equal** relation to the dimensioned fillet. Your sketch should look like **Figure 2-22**.

 From the Features tab select **Extruded Boss**. Extrude the sketch **0.375"** using the **Blind** end condition and click **OK** to finish. The part's **Trimetric** view orientation is shown in **Figure 2-23**.

To finish the part, you will now add the attachment base. Change to a **Front** view orientation, select the front face, and add a new sketch. Draw a **Circle** centered at the origin and dimension it **1.125"** in diameter.

 Now draw a **Hexagon** at the origin. Check **Inscribed Circle** and set the diameter to **0.625"**.

 **Horizontal** To fully define the sketch, select the top line and add a **Horizontal** relation; this will prevent the hexagon from rotating.

From the Features tab select the **Extrude** command. Change to an **Isometric** view for better visibility and extrude the sketch **0.75"** away from the part using the **Blind** end condition.

 To finish the knob, you need to add a few fillets. From the Features tab select the **Fillet** command and set the radius to **0.05"**. To remove the sharp edges, you can:

- Select the front and back edges of the first extrusion and the edge connecting the first extrusion with the second one
- Or
- Select the front and back faces of the first extrusion and click **OK** to finish.

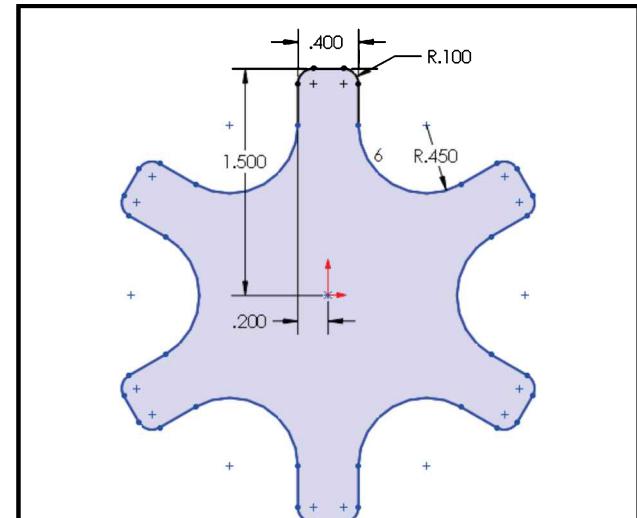


Figure 2-22. The Sketch after Filleting the Intersecting Lines.

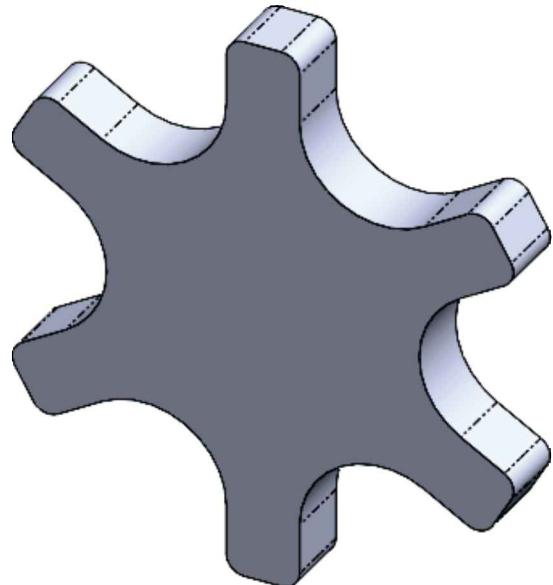


Figure 2-23. The Extruded Sketch.

Use the **Appearances** command to change the color of the part to your liking and save your model as **SCALLOPED KNOB.sldprt**.

Make a new drawing using the **TITLEBLOCK-INCHES** template, add a **Trimetric** view using the **Shaded with Edges** display style and a scale of 2:1. Add the missing notes and save your drawing as **SCALLOPED KNOB.slddrw**. Print a copy to submit to your lab instructor. The finished drawing is shown in **Figure 2-26**.

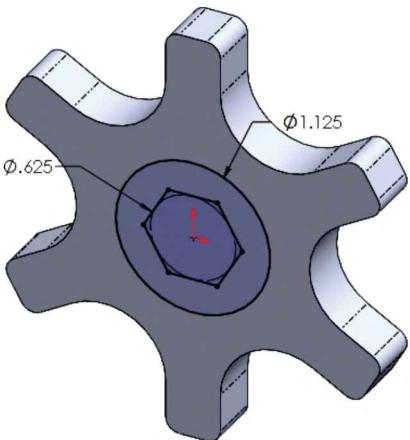


Figure 2-24. Isometric View of the Dimensioned Sketch.

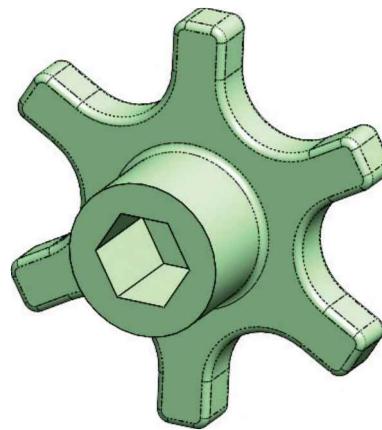
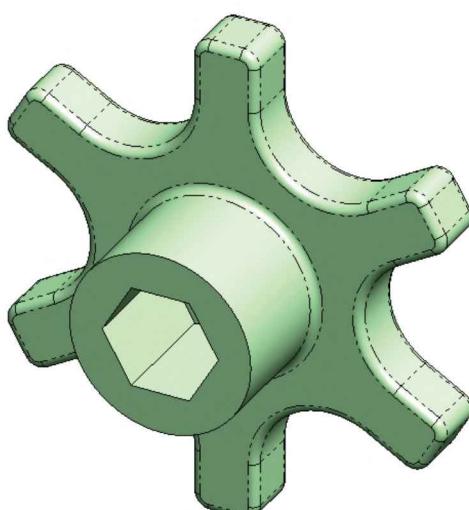


Figure 2-25. The Finished Scalloped Knob.



SCALLOPED KNOB
Scale 2:1
Material: Mild Steel

NAME: STUDENT	DESK:	SEC:	GRADE:	2.3
DESIGN WORKBOOK USING SOLIDWORKS				

Figure 2-26. The Finished Scalloped Knob Drawing.

Exercise 2.4: LINEAR STEP PLATE

In Exercise 2.4, you will design a Linear Step Plate used for linear motion control in machinery. There are a lot of holes on this part, and you will find the linear pattern and mirror commands to be very useful in this case. Start a new part using the **ANSI-INCHES** template and save it as **LINEAR STEP PLATE.sldprt**.

Select the **Right** Plane and add a new **Sketch**. Your part will be automatically rotated to a Right view. Draw a vertical centerline through the Origin, and with the centerline selected, go to the menu **Tools**, **Sketch Tools**, and select **Dynamic Mirror**. Draw the right half of the profile shown in **Figure 2-27**. Each line drawn on the right side of the centerline will be automatically duplicated on the left side. Use the **Smart Dimension** command to fully define the sketch as shown in the **Figure 2-27**.

From the Features tab select the **Extruded Boss** command. To make the base feature symmetrical about the sketch plane you have two options:

- Extrude the sketch **4.20"** in **Direction 1** using a **Blind** end condition, and **4.20"** in **Direction 2** using a **Blind** end condition.
- or
- Extrude the sketch **8.4"** using the **Mid Plane** end condition.

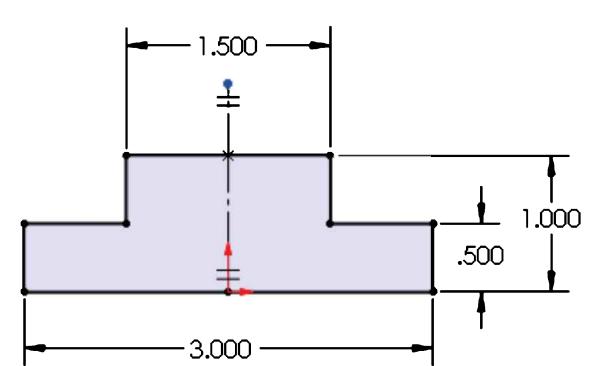


Figure 2-27. The sketch for the Base Feature.

If the part is symmetrical, using the **Mid Plane** end condition is a better option to maintain your design intent. Use either approach and click **OK** to finish the extrusion. The base feature looks like **Figure 2-28**.

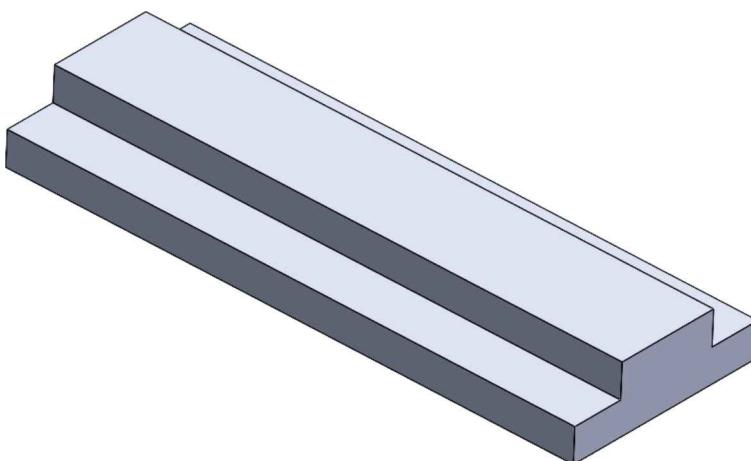
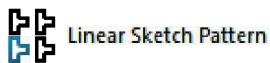


Figure 2-28. The Extruded Base Feature.

The next step is to add the holes to the part. Change to a **Top** view orientation and add a new sketch in the face of the lower step. Draw a small circle and dimension as shown in **Figure 2-29**.



Now create a Linear Pattern. Select the **Linear Sketch Pattern** from the Sketch tab. In the **Direction 1** options set the number of copies to **6** with a spacing of **1.200"** and activate the “**Dimension x spacing**” and “**Fix X-axis direction**” options.

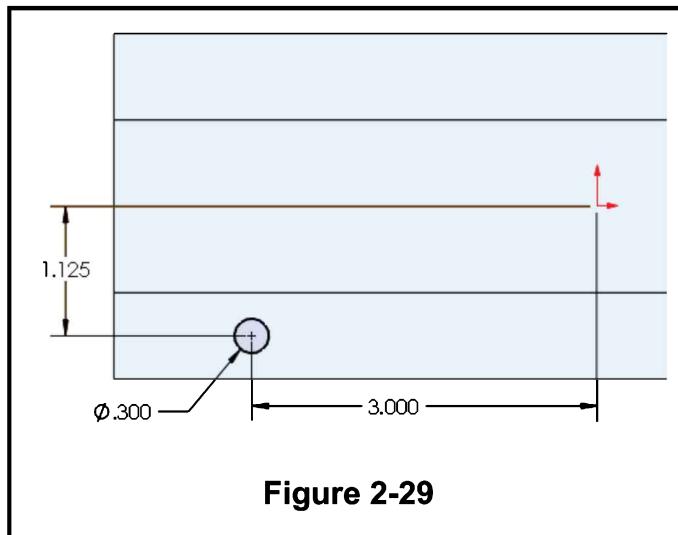


Figure 2-29



Now that one side of the circles is complete, you need to add the six circles on the other surface. To copy the linear pattern, you will use the **Mirror** command. First draw a **Horizontal** centerline starting at the Origin and then select the **Mirror Entities** command. In the Mirror options add the six circles from the linear pattern in the “**Entities to mirror**” selection box. Activate the “**Mirror about:**” selection box and select the centerline. When you see the mirrored circles preview, click **OK** to finish, as shown in **Figure 2-30**.

From the Features tab select **Extruded Cut** command. Use the **Through All** option and click **OK** to finish. Now that you have made the small holes for the part, you need to add the larger diameter holes to create the counterbores.

NOTE: SOLIDWORKS has a **Hole Wizard** command that can easily create many different types of holes for your designs, including counterbore holes, which can then be patterned and mirrored, simplifying the process significantly. The **Hole Wizard** will be covered in a later lesson; in this lesson the focus is on sketch patterns.

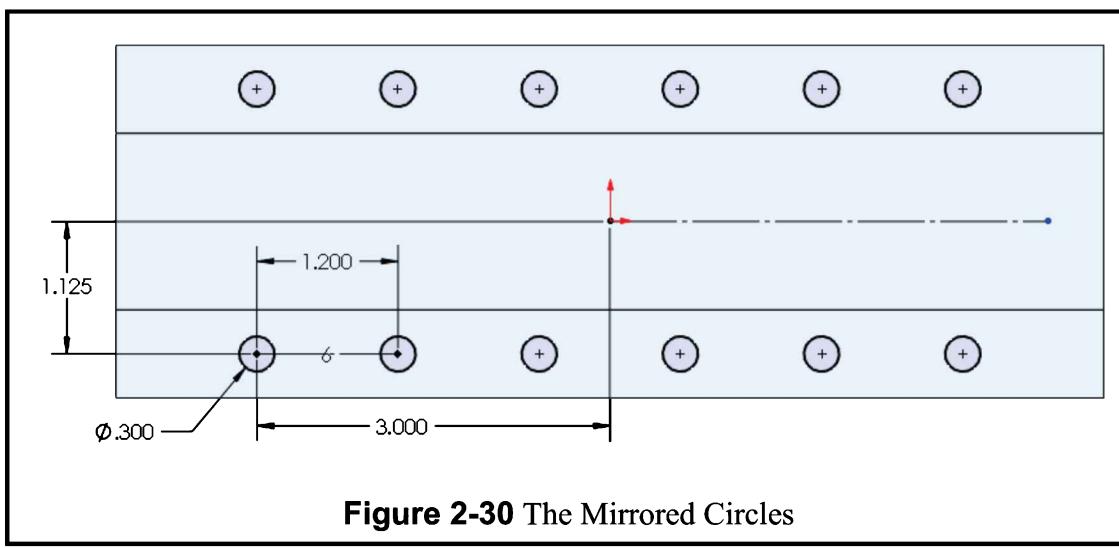
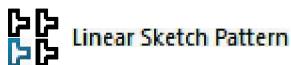


Figure 2-30 The Mirrored Circles

To add the counterbore holes, select the same face as before and add a new sketch. Draw a **Circle** concentric to the first hole, and dimension it **0.600"** diameter.



Now add a **Linear Pattern** as before, but this time you will make the pattern in two directions. In **Direction 1** make six (**6**) copies spaced **1.200"**. In **Direction 2** make two (**2**) copies spaced **2.250"**, turn on the “Dimension Y spacing” and “Dimension angle between axes” to fully define the pattern. Click **OK** to finish the pattern. See **Figure 2-31**.

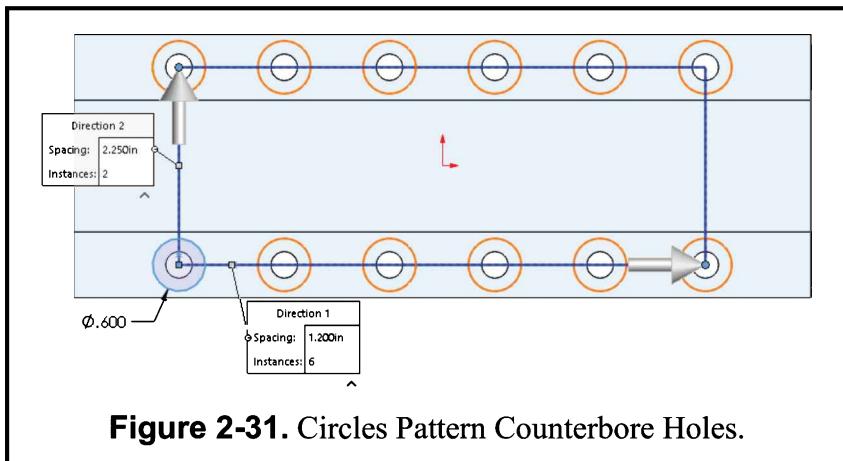


Figure 2-31. Circles Pattern Counterbore Holes.

From the Features tab select the **Extruded Cut** command. Use the **Blind** option to a depth of **0.125"** into the material and click **OK** to finish. Now that you have bored the counterbores into the plate's two steps, your part will look as shown in **Figure 2-32**.

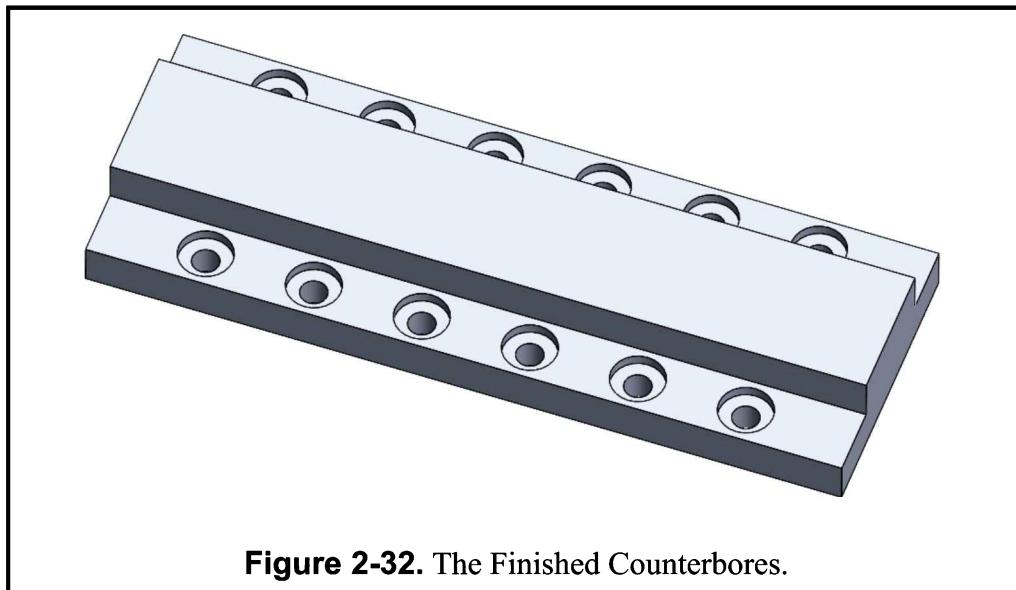


Figure 2-32. The Finished Counterbores.

Note: If you make a mistake or need to modify a feature's definition, you can click on the feature to be modified in the FeatureManager, and select **Edit Feature** from the pop-up context menu, as seen in **Figure 2-33**.

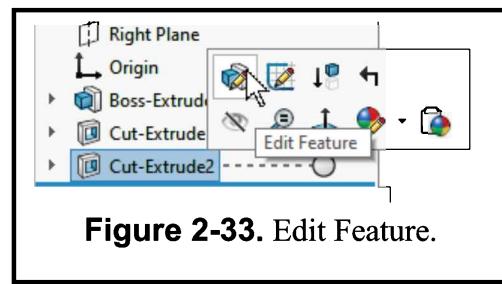


Figure 2-33. Edit Feature.

The next step for your design is to create four holes on the top of the plate. Change to a **Top** view orientation and add a new Sketch on the top face. Draw the first **Circle** and dimension it as indicated in **Figure 2-34**.

 **Horizontal** To maintain the design intent, select the center of the circle and the origin, and add a **Horizontal** relation to fully define the circle.

Use a **Sketch Linear Pattern** to get a second circle **1.20"** to the right of the first circle. Draw a vertical **Centerline** starting at the Origin, and finally use the **Mirror Entities** command to copy the two circles. The completed sketch is shown in **Figure 2-34**.

Change to an **Isometric** view for better visibility. From the Features tab select **Extruded Cut**, use the **Through All** end condition and click **OK** to finish. You have completed the small holes all the way through the thick part of the plate, as seen in **Figure 2-35**.

Now you need to add two counterbore slots. Change to a **Top** view and add a new sketch on the top face of the part. Select the **Slot** command from the sketch tab and use the **straight slot** option. Start the slot concentric to the left circle of the top face and end it concentric to the hole immediately to the right. This will make the slot concentric with the two circles to the left of the center. Using the same approach, draw a second slot on the holes to the right of the origin and change the slots' height to **0.80"**, as shown in **Figure 2-36**.

NOTE: Using the slot's "**Add dimensions**" option will add width and height dimensions to the slot. Since we defined the slot's width using the hole's centers, the width dimension is redundant and it will be colored grey, meaning it is a reference dimension. In other words, it cannot change the slot's size.

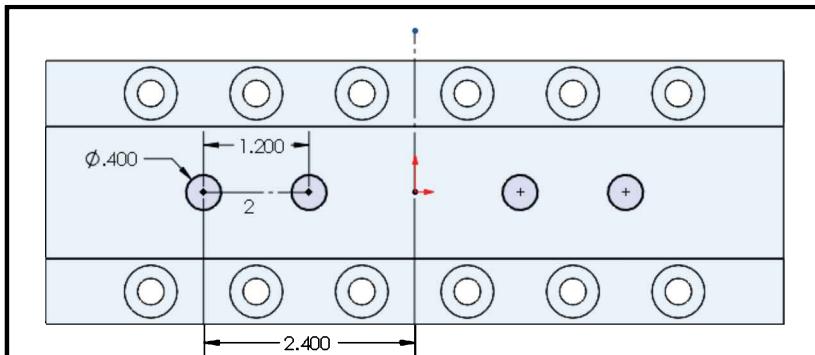


Figure 2-34. Creating the Holes in the Top Surface.

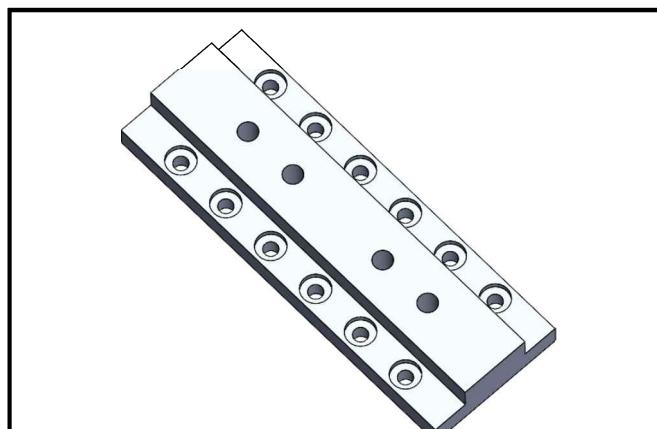


Figure 2-35. The Four Holes in the Top Face.

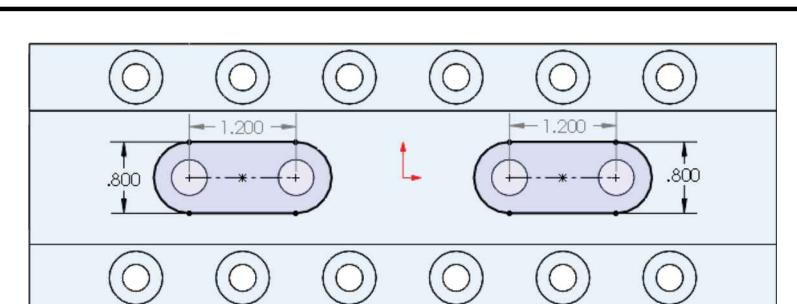
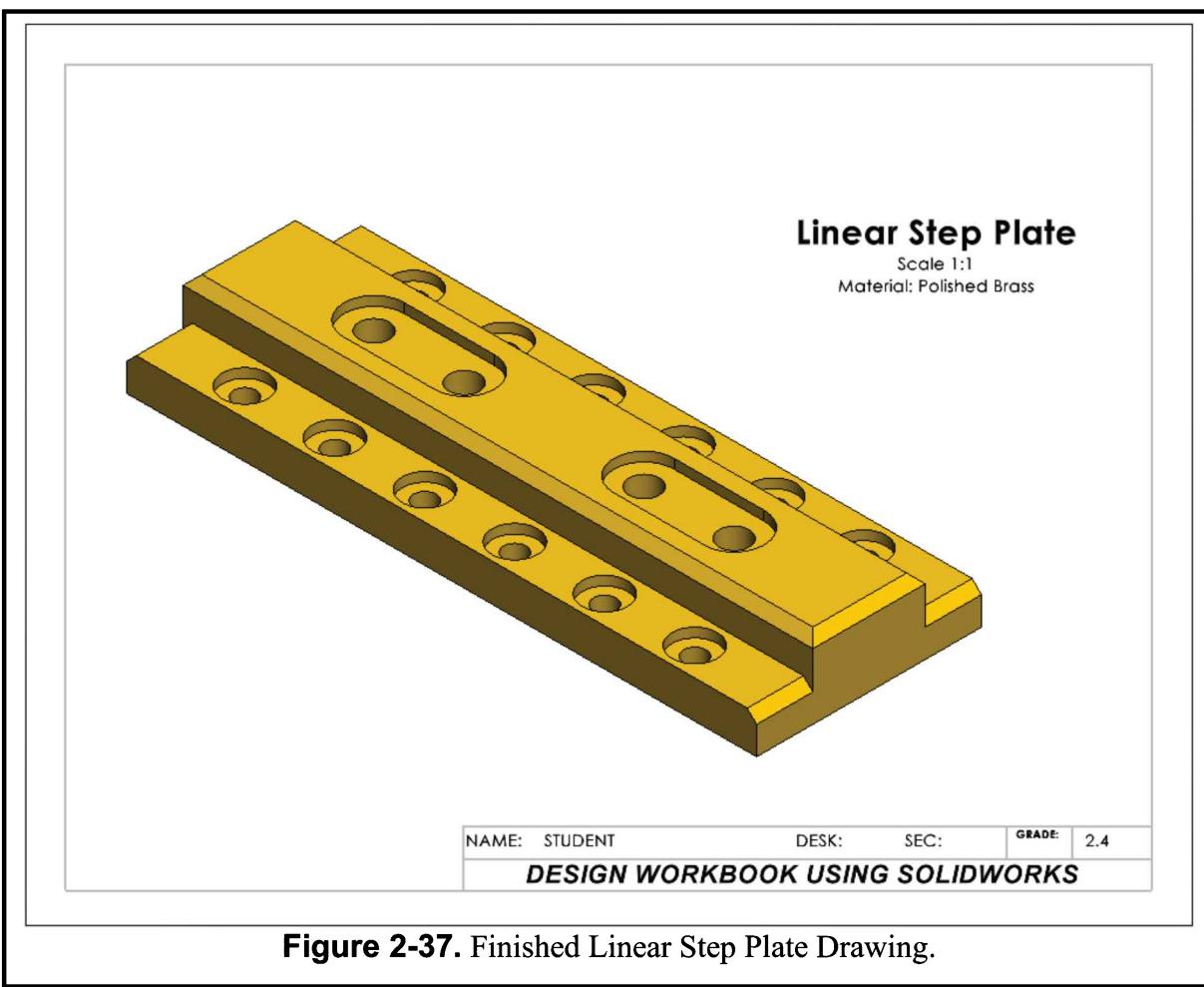


Figure 2-36. The Completed Linear Step Plate.

To add the counterbores select the **Cut Extrude** command using a **Blind** end condition of **0.250"**. These counterbore slots are shown in **Figure 2-37**. The final step is to chamfer the indicated model edges. In the Features tab click on the **Fillet** command's pull-down menu and select **Chamfer**. Set the chamfer value to **.125"**, select the six short edges at the sides of the step plate and the two long edges of the top surface. Click **OK** to complete the exercise.

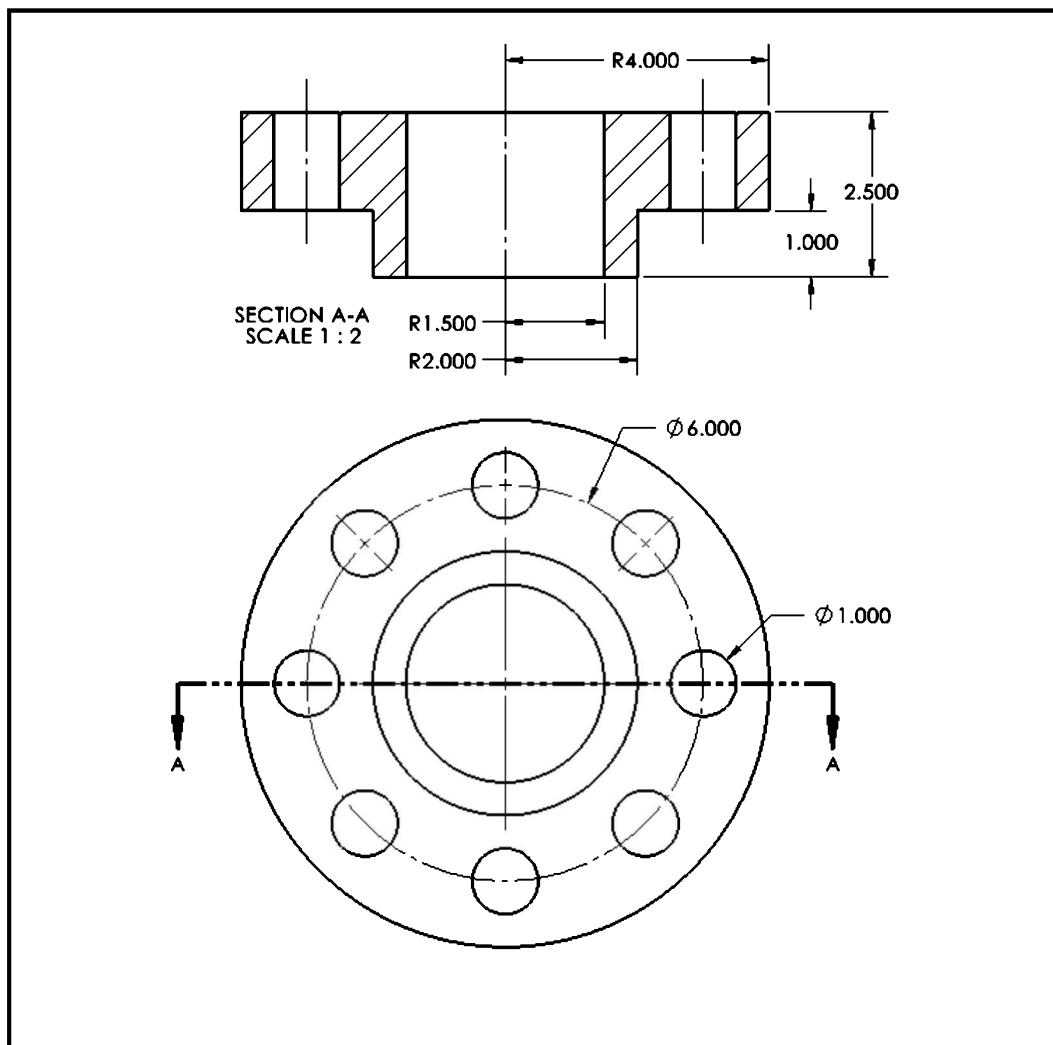
To finish the part, right click on **Material** in the FeatureManager and select **Edit Material**. Expand the **Copper Alloy** materials category and assign **Brass** to the Linear Step Plate. The finished part is as shown in **Figure 2-37**. Save your part as **LINEAR STEP PLATE.sldprt**.

Make a new drawing using the **TITLEBLOCK-INCRES.drwdot** template, add a shaded with edges **Isometric** view with a 1:1 scale, and add the necessary annotations. Save your drawing as **LINEAR STEP PLATE.slddrw** and print a copy for your lab instructor.



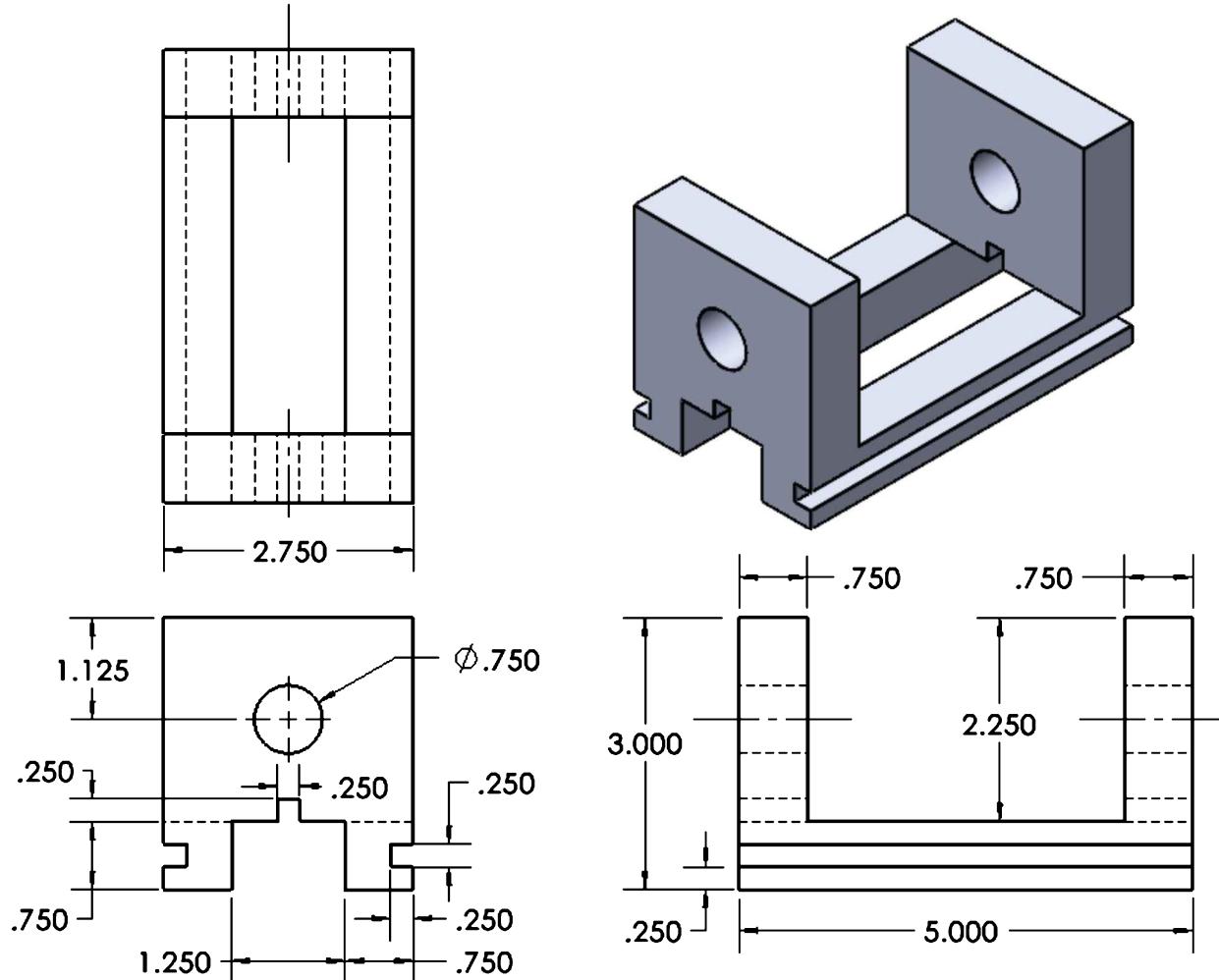
Supplementary Exercise 2-5: FLANGE

Build the following model using the **Revolve** and **Circular Pattern** commands learned in this lesson. Add the sketch for the **Revolve** feature in the **Top Plane** and dimension it using the next drawing as a guide. The units of measure are in Inches. Make a drawing with an Isometric view and save it as “**FLANGE**.”



Supplementary Exercise 2-6: STEEL VISE BASE

Make a model of the figure below using the commands learned so far. All holes and slots are through. Make a drawing with an **Isometric** view and save it as **STEEL VISE BASE**.
TIP: Use the **Dynamic Mirror** command for the base feature.



Design Workbook Lab 3: 3D Modeling Part I

In the first two Labs you created some solid parts using simple 2D sketches. In these parts, most of the geometric information was defined in a 2D sketch, and then it was either extruded or revolved to make the 3D model. While many simpler components can be designed using the tools learned so far, SOLIDWORKS includes advanced tools to create more complex designs. In this Lab you will learn how to use some of these tools to create and modify your 3D models.

SKETCH RELATIONS

So far you have learned how to create and dimension basic sketch geometry and how to add basic sketch relations. In this lab you will learn more about adding relations to improve your designs by allowing you to define how sketch geometry is defined, in reference to the current sketch, other geometric elements, or existing 3D model features.

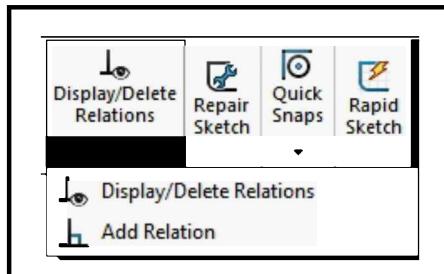


Figure 3-1. Sketch Relations.

In short, the design intent is the plan you make so that your parts update *predictably* when you make changes. For example, if your design is symmetrical, when you make a dimensional change it should remain symmetrical. Under the **Display/Delete Relations** button, you can also access the **Add Relation** command as shown in **Figure 3-1**. The following list includes most of the sketch relations available in a 2D sketch.

Horizontal makes one or more lines horizontal, or two or more endpoints horizontal.

Vertical makes one or more lines vertical, or two or more endpoints vertical.

Collinear makes two or more lines lie on the same infinite line.

Co-radial makes two or more arcs share the same center point and radius.

Perpendicular makes two lines perpendicular to each other.

Parallel makes two or more lines parallel to each other.

Tangent makes an arc, ellipse, or spline, and a line, edge, or arc tangent to each other.

Concentric makes two or more arcs, or a point and an arc to share the same center point.

Midpoint makes an endpoint to coincide with the midpoint of another line.

Intersection makes two lines and one point to remain at the intersection of the lines.

Coincident makes a point and a line, arc, or ellipse to lie on the line, arc, or ellipse.

Equal makes two or more lines or two or more arcs have the same length or radii.

Symmetric makes a centerline and two points, lines, arcs, or ellipses to be equal and symmetric about the selected centerline.

Fix makes an entity's size and location fixed. Should be used as a temporary solution.

THE FEATURES TOOLBAR

The Features toolbar is shown in **Figure 3-2**. This toolbar includes most of the commands available in SOLIDWORKS to create and modify 3D features. Below are descriptions of these features.

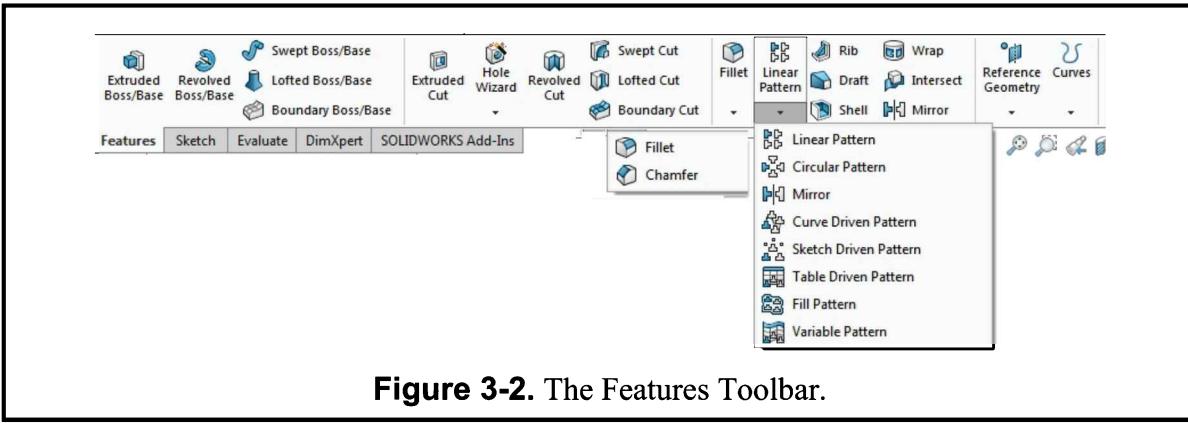


Figure 3-2. The Features Toolbar.

Extrude Boss/Base creates a base or boss by extruding a sketch in a linear direction.

Revolve Boss/Base creates a base or boss by revolving a sketch around a centerline.

Extruded Cut removes material from a solid body by linearly extruding a sketch through it.

Revolved Cut removes material from a solid body by revolving a sketch around a centerline.

Sweep creates a base, boss, cut, or face by moving a profile along a designated path.

Loft creates a feature by making a transition between multiple profiles.

Fillet creates a rounded internal or external face on the part by selecting an edge or face.

Chamfer creates a beveled feature on the selected face, edge or vertex.

Rib adds material of a specified thickness determined by a contour in an existing part.

Shell hollows out the part, optionally removing the selected faces.

Draft tapers faces of a part using a specified angle.

Wrap wraps a closed sketch contour onto a face.

Hole Wizard allows you to quickly add different types of standard holes to your part.

Linear Pattern creates multiple instances of selected features along one or two linear directions.

Circular Pattern creates multiple instances of one or more features uniformly around an axis.

Mirror Feature creates a mirror copy of one or more features about a plane.

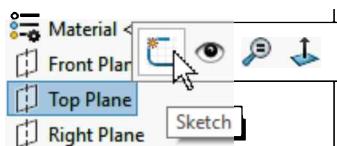
Reference Geometry creates reference geometry like planes, axis, coordinate systems, and points.

Curves - creates different types of curves including spiral and helix.

Exercise 3.1: CLEVIS MOUNTING BRACKET

In Exercise 3.1, you will design a Clevis Mounting Bracket. In this case the design intent requires that the part be symmetrical, and the hole in the clevis remains concentric to the outer circular edge even if the design dimensions change and should also go through the width of the part, regardless of its size. In order to maintain the design intent conditions, you will need to use different sketch relations, features and end conditions in your part.

Start a new part using the **ANSI-INCHES.prtdot** template and save it as **CLEVIS MOUNTING BRACKET**. In order to maintain the part's design intent first you will create the base, the side second, then create a 3D mirror of the side, and finally the cutouts in the base.



To start, select the **Top** plane in the FeatureManager and add a new sketch. The model's orientation is changed to a Top view. Draw a rectangle with a centerline across the diagonal and dimension it as shown in **Figure 3-3**. Do not start the rectangle at the origin. Open the **Add Relations** command from the Display/Delete Relations drop-down icon. In the “Selected Entities” box add the Centerline and the Origin.

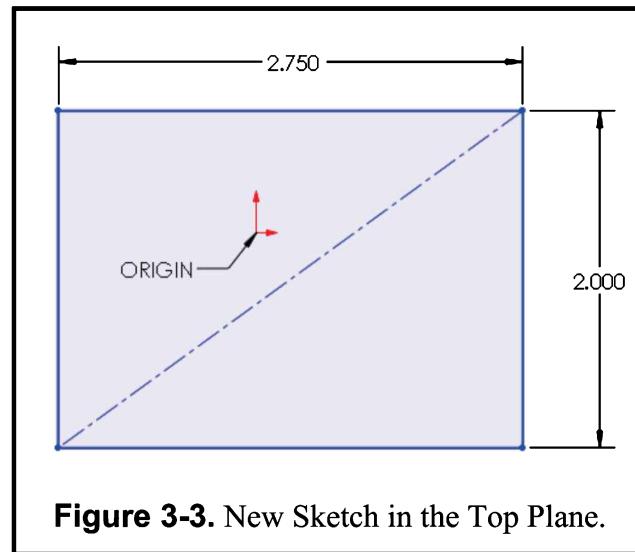


Figure 3-3. New Sketch in the Top Plane.

From the available relations click on **Midpoint**. By adding this relation, the centerline, and therefore the rectangle, will be centered about the origin, making the base feature symmetrical about the **Front** and **Right** planes. Make a **0.500"** **Extruded Boss** as seen in **Figure 3-4** and click **OK**.

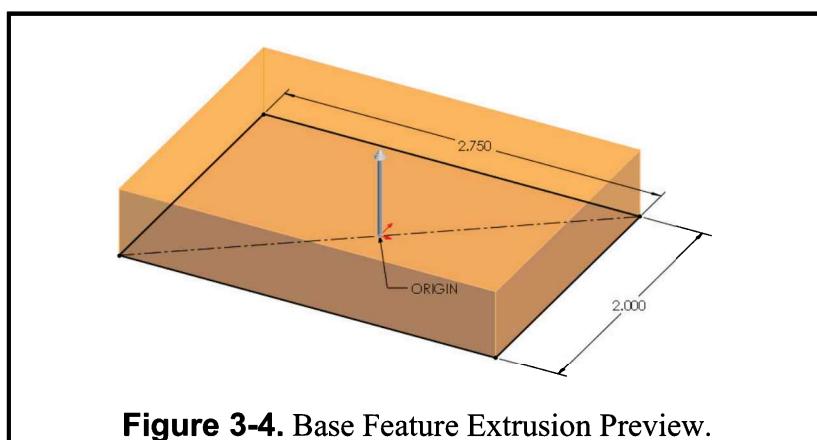
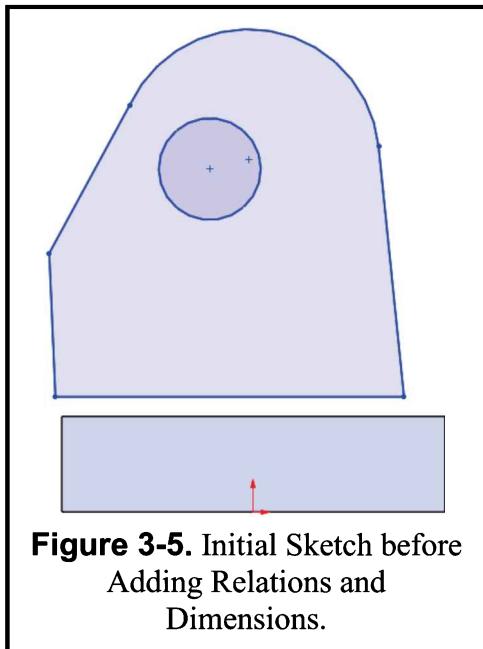


Figure 3-4. Base Feature Extrusion Preview.



For the next step change to a **Right** view orientation. Select the right face of the base feature and add a new sketch. In order to practice manually adding sketch relations, draw a sketch profile approximately as **Figure 3-5** above the base feature.

Open the Sketch Relations command. The different sections of the **Add Relations** command are:

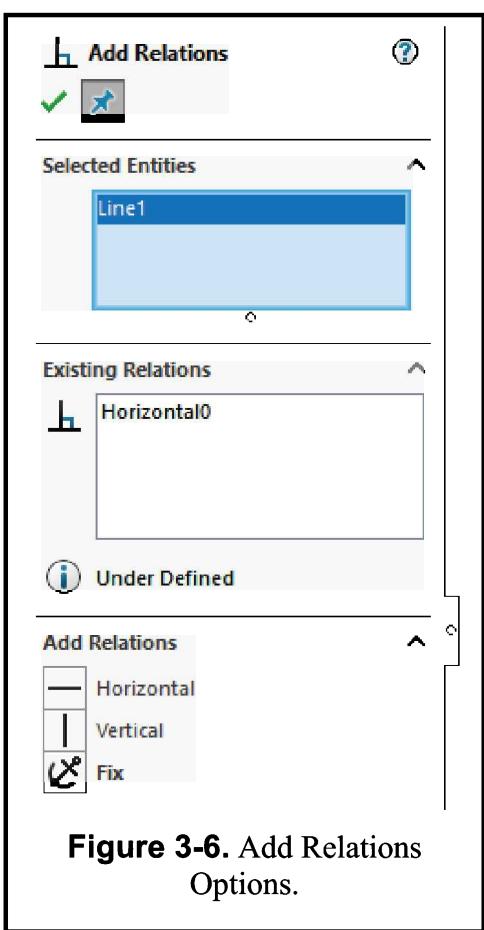
“**Selected Entities**” displays the currently selected entity(ies);

“**Existing Relations**” displays existing relations of the selected item(s);

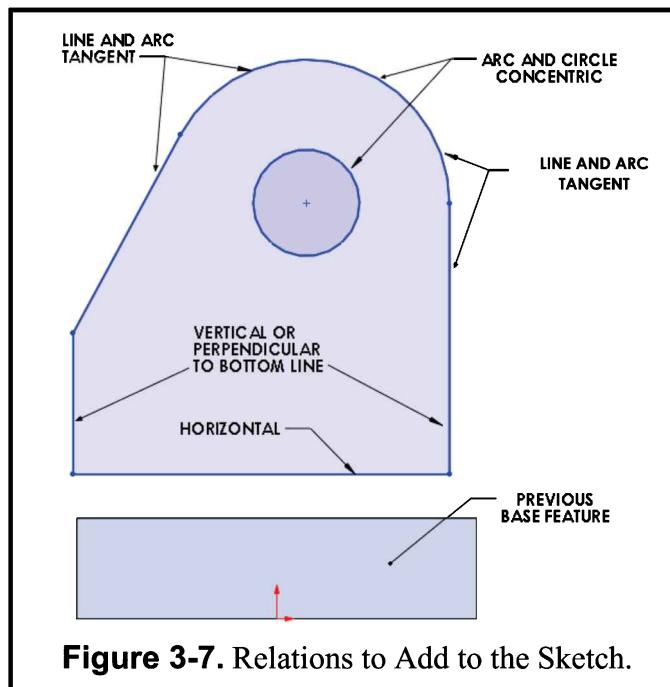
“**Add Relations**” lists all the possible relations the item(s) can have.

Select the bottom line and add a **Horizontal** relation. If you automatically captured this relation when the line was drawn, it will be listed in the “**Existing Relations**”

box when you select it. Continue adding the relations indicated in **Figure 3-7**. In each case select the corresponding entities and add the relations indicated.



After these relations are added, make the lower left endpoint of the sketch coincident with the lower left corner of the base feature, and repeat for the lower right endpoint. To fully define the sketch, make the upper endpoint of the left vertical line coincident to the upper left corner of the base feature, as shown in **Figure 3-8**.



To finish the sketch, add the dimensions indicated in **Figure 3-8**.

At this point, your sketch will be fully defined, and all geometry should be black. To verify the proper geometric relations have been added, double click on the **2.25"** dimension, and temporarily change it between **1.00"** and **3.00"**. Notice the tangent relations are maintained as the dimension is changed, and the sketch updates predictably based on your design intent.

Select the **Extruded Boss** command and change to an **Isometric** view orientation to preview the extrusion. Use the Reverse Direction button to make the new boss go *into* the existing base feature. Make the extrusion **0.375"** deep as shown in **Figure 3-9** and click **OK**.

Now you need to create a new extrusion equal to the last one on the other side of the bracket. Instead of adding a new sketch and extruding it, you will use the **Mirror** feature command which will create a 3D feature.



To use the **Mirror** command (and almost any command), you can pre-select the mirror plane and the feature(s) to mirror, or you can select the command, and make the selections after. In the **Mirror** command you need to select the mirror plane, which in your case will be the **Right** plane of the part. If you remember the first feature was centered about the origin, making it symmetric about the **Front** and **Right** planes, this is almost always a good idea if you anticipate your part to be symmetrical, or have a certain level of symmetry.

In the “**Mirror Face/Plane**” selection box you need to select the **Right Plane**; to do this click in the down-arrow to expand the FeatureManager at the top right side of the PropertyManager. After selecting the **Right Plane**, the “**Features to Mirror**” selection box is activated. Click to select the **Boss-Extrude2** feature and click **OK** to finish. You will see the preview of the mirror in the screen. Your finished part will look like **Figure 3-10**.

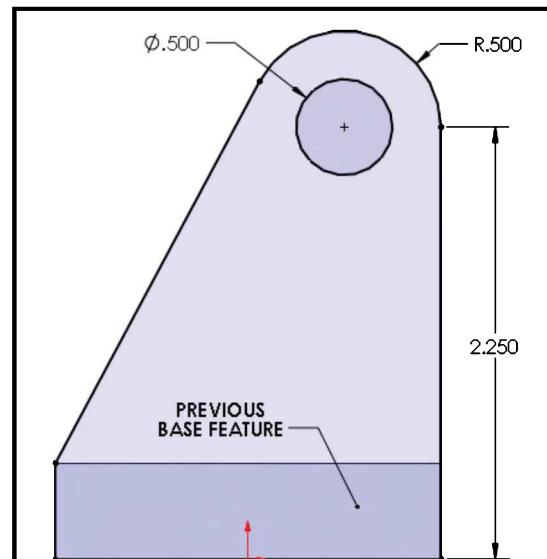


Figure 3-8. Fully Defined Dimensioned Sketch.

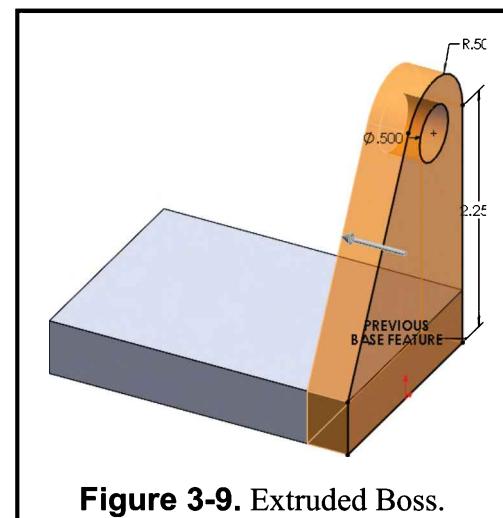
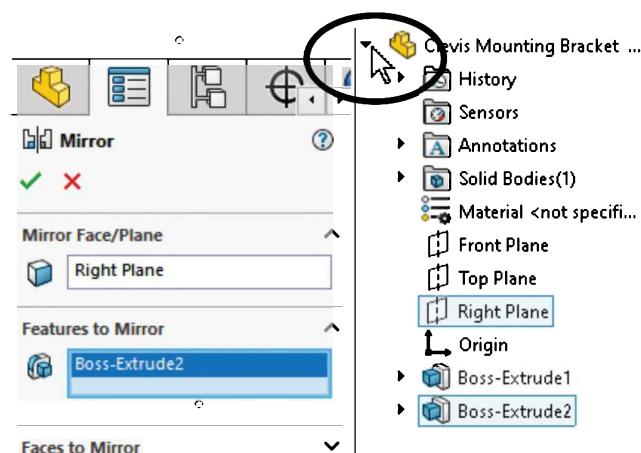


Figure 3-9. Extruded Boss.



To finish the bracket, you need to add a cutout in the base. Change to a **Top** view orientation and add a new sketch in the top face of the base feature. Draw the sketch profile shown in **Figure 3-11** using the **Line**, **Circle** and **Arc** tools, and dimension as indicated.

To center the slot in the part, add a **Vertical** geometric relation between the arc's center and the part's origin. Keep in mind the slot's profile must be closed; remember to add a line to close it at the edge of the face.

Select the **Extruded Cut** command. Use the **Trough All** end condition and change to an **Isometric** view orientation to make sure the cut is going down. Click **OK** to finish.

Use the **Fillet** command to round the inside edges where the base feature meets the vertical faces using a **0.125"** radius.

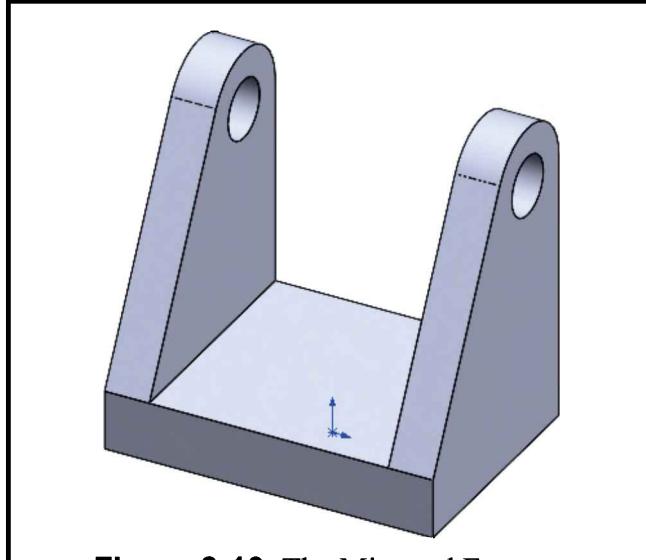
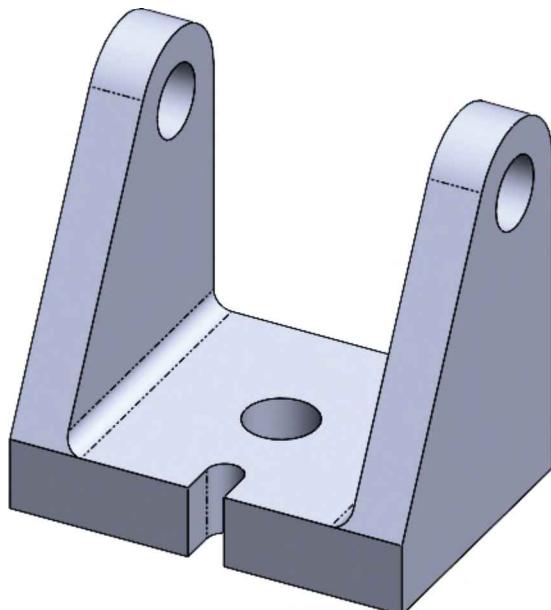


Figure 3-10. The Mirrored Feature.

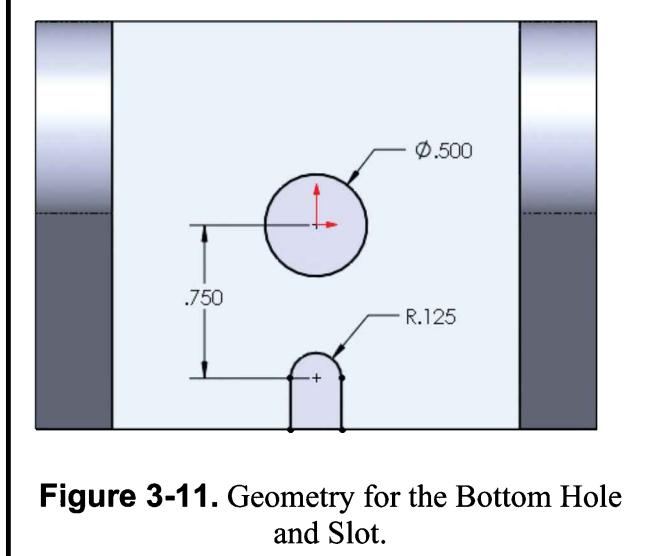
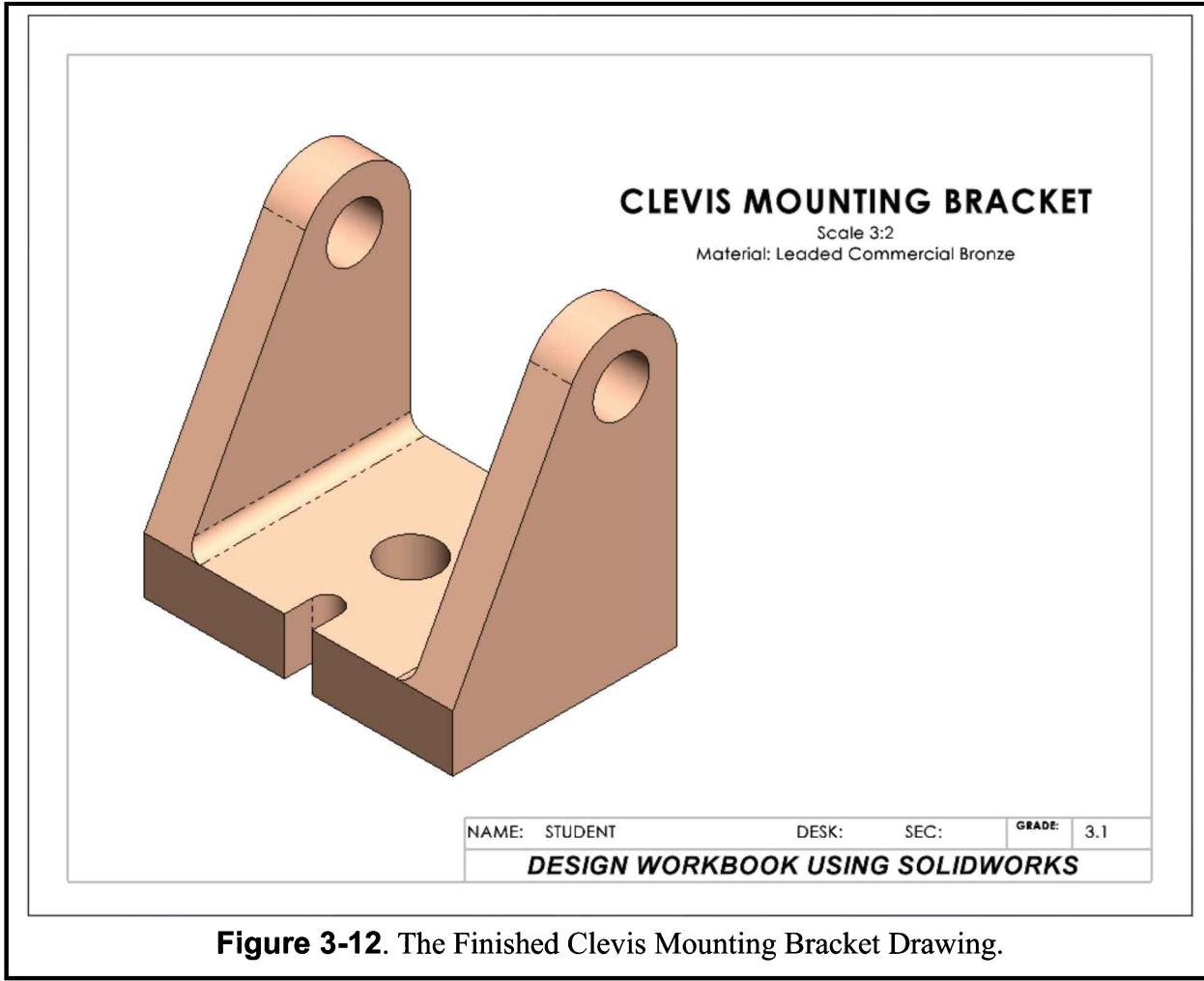


Figure 3-11. Geometry for the Bottom Hole and Slot.

Your Clevis Mounting Bracket is now complete. To define the part's material right click on **Material** in the FeatureManager, select **Edit Material** from the context menu, expand the **Copper Alloy** category and select **Leaded Commercial Bronze**.

If you haven't already, save your part as **CLEVIS MOUNTING BRACKET.sldprt**. Make a new drawing and add an **Isometric** view using the Shaded with Edges display mode. Add the missing notes and save your drawing as **CLEVIS MOUNTING BRACKET.slddrw**. The finished drawing is shown in **Figure 3-12**.

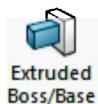


Exercise 3.2: MANIFOLD

In this Exercise you will design a Manifold. A manifold's purpose is to distribute air (or any fluid) to multiple locations through its ports. Since the manifold's ports are all similar, you will be able to design one of them, and use the **Pattern** and **Mirror** features to replicate them.

Start a new part using the **ANSI-METRIC** template and save it to your personal folder as **Manifold.sldprt**. Select the **Right** plane in the FeatureManager and add a new sketch. Your part will automatically change to a **Right** view orientation.

Draw two concentric **Circles** centered at the Origin, dimension the larger circle **60** mm in diameter and the smaller circle **45** mm in diameter. These two dimensions will fully define your sketch.



From the Features tab select the **Extruded Boss/Base** command, select the **Mid Plane** end condition, make it **300 mm** and click **OK**. You now have a long flute centered at the origin, as shown in **Figure 3-13**.

The next step is to add a collar on one side of the manifold. Select the right-side face of the part and add a new sketch. Select the inner circular edge

and click in the **Convert Entities**



command to project it to the new sketch. Now draw a new circle centered at the origin and dimension its diameter **75 mm**.

Select the **Extrude Boss/Base** command and use the **Blind** end condition. Reverse the direction to extrude it on top of the base feature, set the distance to **50 mm** and click **OK** to finish. You now have a small collar in one end of the manifold.

To duplicate the collar on the other side of the manifold you will use the **Mirror** command. In the previous exercise you learned how to use the **Mirror** command and display the FeatureManager at the same time to make the necessary selections. In this case you will use a pre-selection approach. Hold down the **Ctrl** key and pre-



select the **Right** plane and the previously made **Boss-Extrude2** feature. Notice both features are highlighted in the screen. From the Features tab select the **Mirror** command and click **OK** to finish. You now have a collar on each end of the manifold, as shown in **Figure 3-15**.



Figure 3-13. Manifold's Base Feature.

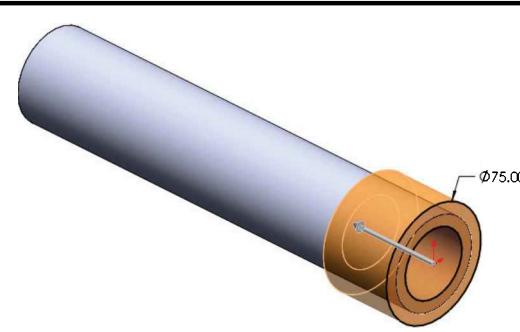
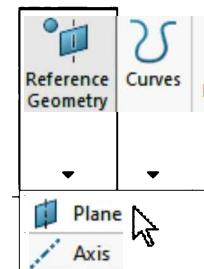


Figure 3-14. Manifold Collar Added.

The next step is to add the first port in the manifold. To do this you will need to create a reference plane to draw a new sketch above the part, since there are no planes or flat faces to add a sketch to. If needed, change to an **Isometric** view orientation for clarity. From the Features tab select the **Reference Geometry** command and click on **Plane**, or from the menu **Insert, Reference Geometry, Plane**. In the **First Reference** selection box add the **Top** plane from the FeatureManager, (remember you can expand it while the command is active) set the distance to **42.5 mm** above the **Top** plane and click **OK** to finish. If needed, activate the **Flip offset** option to reverse the side to create the new plane.



Now change to a **Top** view orientation, select **Plane1** and add a new sketch. Draw a circle and dimension it as indicated in **Figure 3-15**. Using the **Add Relations** command, select the circle's center (*not the perimeter*) and the part's origin, and add a **Horizontal** relation to fully define the sketch.

Select the **Extruded Boss/Base** command in the Features tab and change to an **Isometric** view for visibility. Set the **Direction 1** end condition to **Up to Surface**, select the long outer face of the manifold body (it will be highlighted in pink), and click **OK** to finish. Now you have a new boss extruded from **Plane1** down to the selected face.

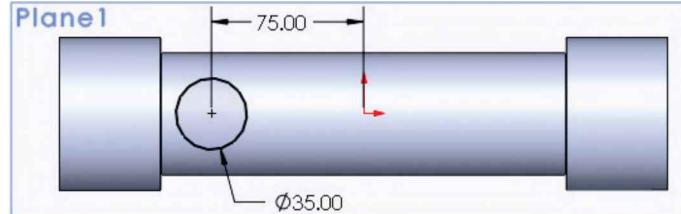


Figure 3-15. Sketch for the Manifold's First Port.

Now add a new sketch at the top of the new boss or **Plane1**, draw a circle concentric with the previous boss and dimension its diameter of **17.5 mm**. If needed, return to an Isometric view and select the **Extruded Cut** command from the Features Tab. Set the Direction 1 end condition to **Up to Surface**, select the cylindrical inside face of the manifold and click **OK** to finish. You have now made a hole going from **Plane1** to inside the manifold, as shown in **Figure 3-16**.

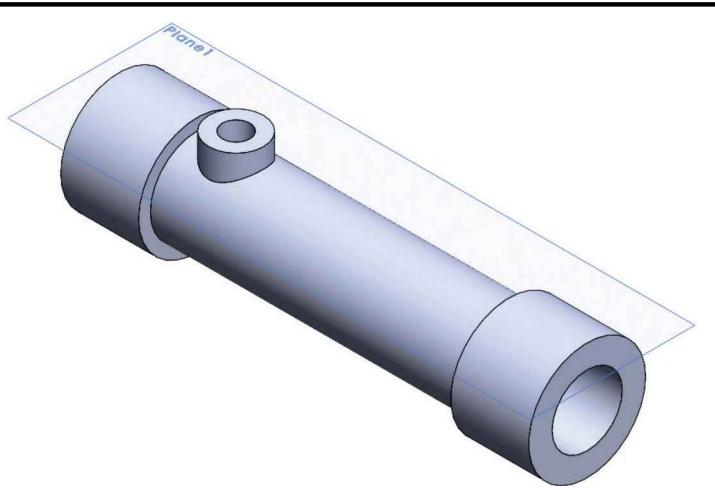
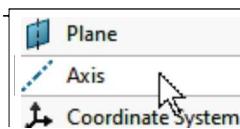


Figure 3-16. The Manifold's First Port Finished.

To better see the hole, use the **Rotate View** command to rotate your model in the screen. Try to view the manifold through the center hole of the part. Later in this exercise you will learn how to use a temporary section view to inspect the inside features of a part.

After the first port is complete you will create the rest using a 3D linear pattern. To create a 3D linear pattern, you need a linear edge or an axis to use as a direction reference. Since the manifold does not have a linear edge to be used as reference, you need to add a reference **Axis**.



From the Features tab select the **Reference Geometry** and click on **Axis** from the drop-down menu, or use the menu **Insert**, **Reference Geometry**, **Axis**.

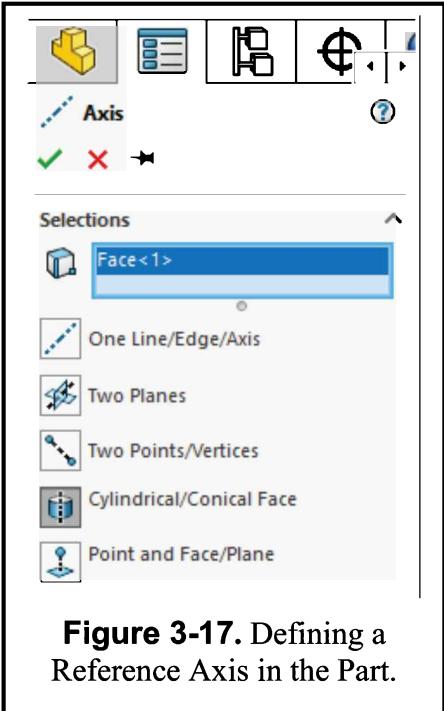
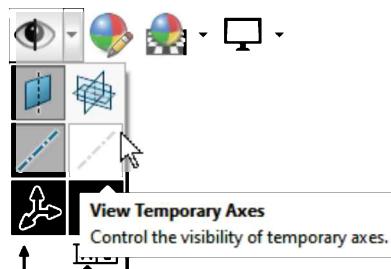


Figure 3-17. Defining a Reference Axis in the Part.

In order to define an auxiliary **Axis**, you can use a cylindrical face, a line, an edge, an axis, two points, or a plane and a point. In this exercise you will select the outer cylindrical face of the manifold and click **OK** to finish. Notice the preview of the **Axis** after selecting the face.

NOTE: Every cylindrical surface has an axis running through its center, and they can be used just as a defined **Axis** as you just did. To view the **Temporary Axis** select the **Hide/Show Items** icon from the **View** toolbar, and turn on the **Temporary Axis** button.



The next step is to create a **Linear Pattern** to add the rest of the manifold ports. From the Features tab select the **Linear Pattern** command.

In the **Direction 1** selection box add the **Axis1** feature made in the previous step, change the spacing to **50 mm** and the number of copies to **four** (remember this value includes the original).

Activate the **Features and Faces** selection box and from the graphics area or the fly-out FeatureManager select the **Boss-Extrude3** and **Cut-Extrude1**. Notice the preview in the screen and click **OK** to finish. The manifold with the four ports is shown in **Figure 3-19**.

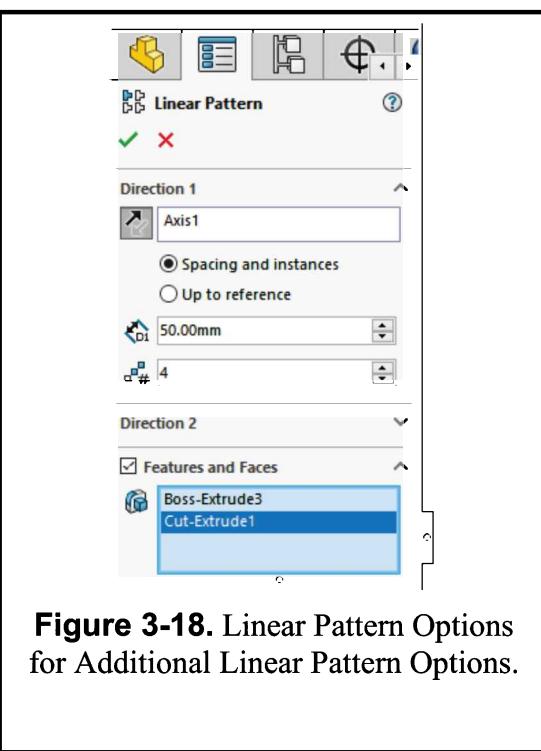


Figure 3-18. Linear Pattern Options for Additional Linear Pattern Options.

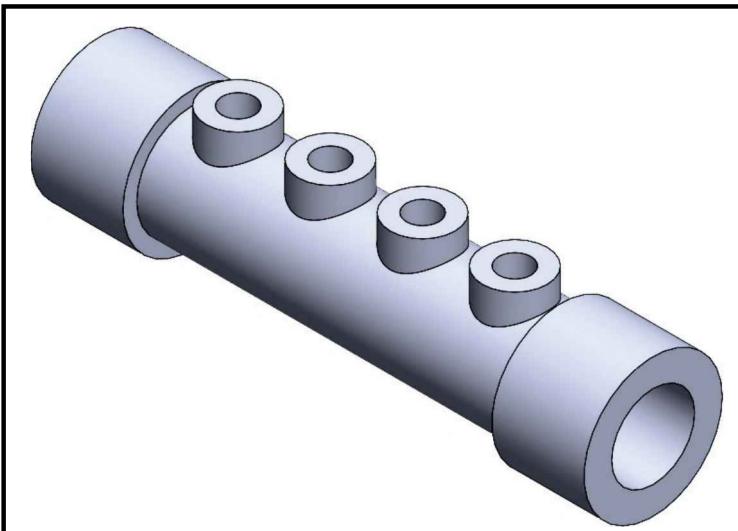


Figure 3-19. Linear Pattern of Manifold Ports.
(Plane1 and Axis1 are Hidden.)

Now you need to add two additional ports at the bottom of the manifold. Since they are identical to the top ports, you can use a **Mirror**. From the Features tab select the **Mirror** command, expand the fly-out FeatureManager and select the **Top** plane in the **Mirror Face/Plane**. In the **Features to Mirror** selection box add the **Boss-Extrude3** and **Cut-Extrude1**. Click **OK** to finish.

To add the last port, make a new **Mirror** feature using the **Right** plane and the previous **Mirror2** feature. After you complete this operation you will have a **Mirror3** feature, as shown in **Figure 3-20**.

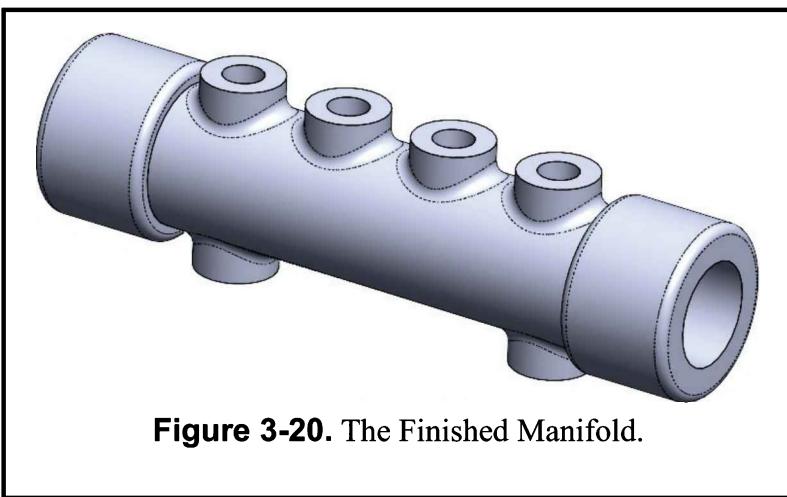
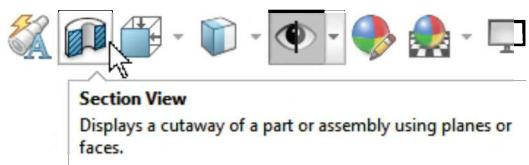


Figure 3-20. The Finished Manifold.

Now you must add a fillet to the edges where the ports meet the manifold's main body. Select the **Fillet** command, set the radius to **6 mm** and select the edge where the six ports meet the main body. Click **OK** to continue.

The final step is to add a **4 mm** fillet to the sharp edges of the two collars. The finished model is shown in **Figure 3-20**.



At this time, you can create a temporary section view of the model to inspect the internal features of the manifold. From the **View** toolbar select the **Section View** command, or from the menu **View, Display, Section View**.

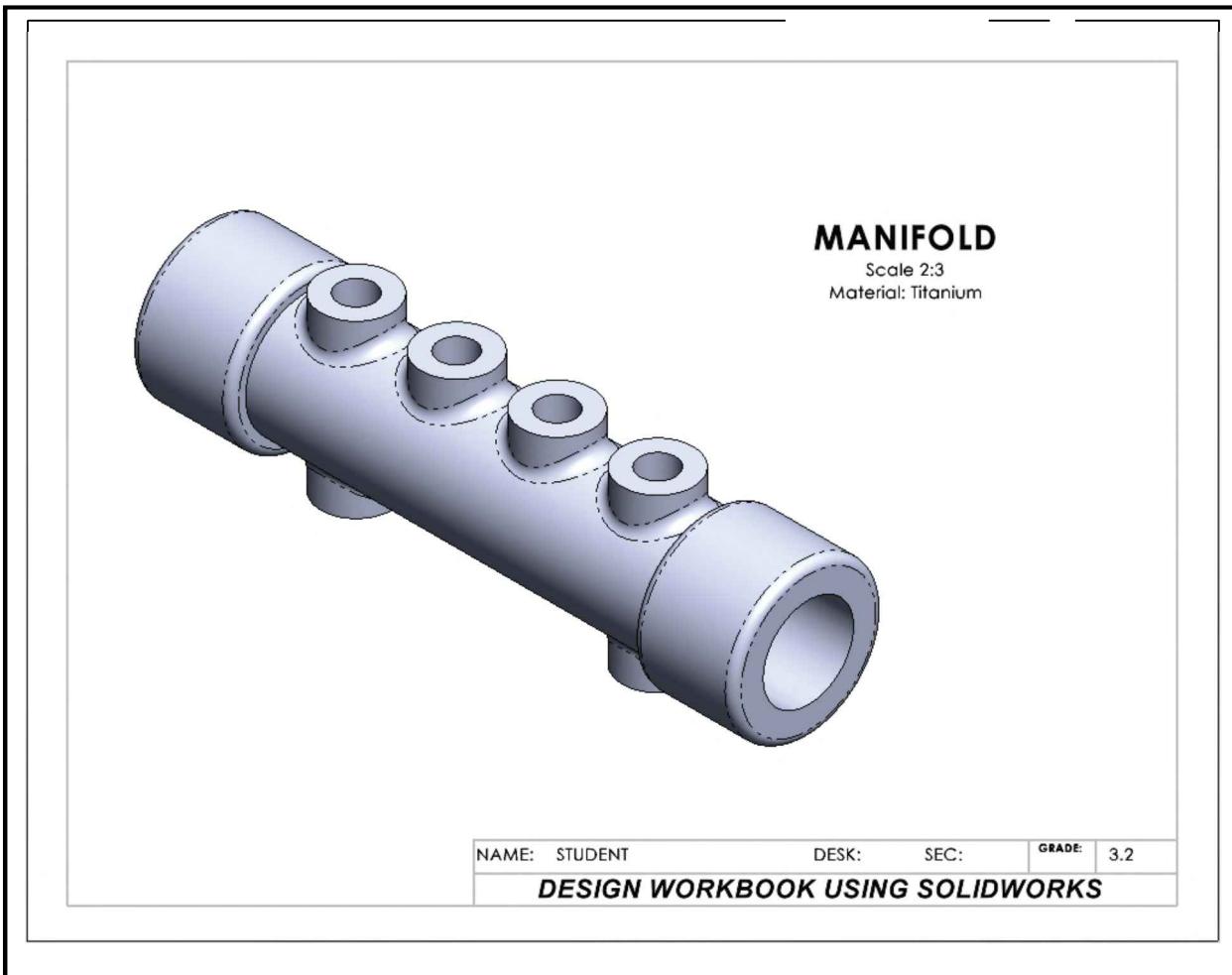
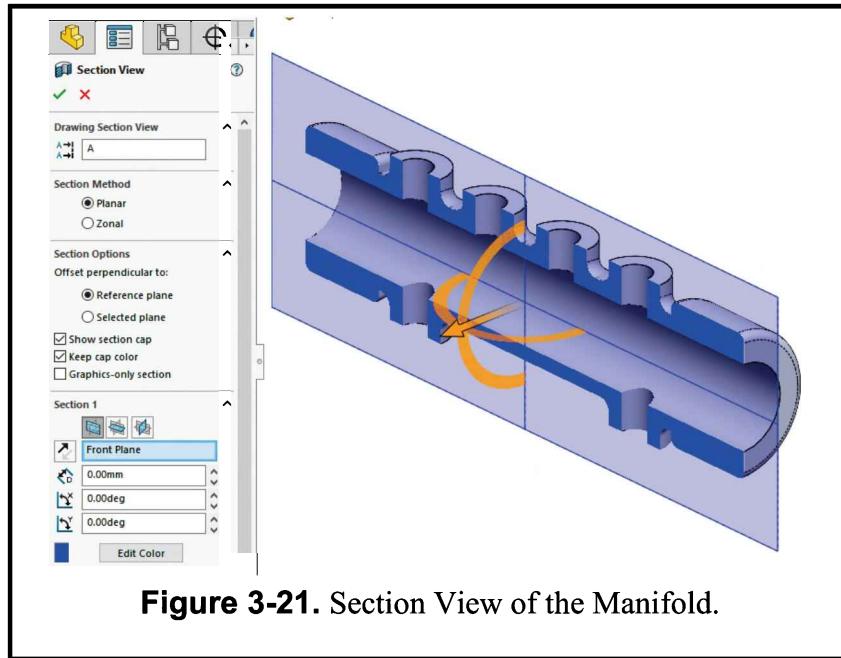
In the Section View options the Front plane is the default selection with a distance of 0 mm. By changing the distance, the section view will be moved parallel to the selected reference. Optionally you can click and drag the arrow and rotation rings to redefine and rotate the section plane, as shown in **Figure 3-21**. After reviewing the section view cancel the section view command to continue.

NOTE: If you click **OK** to close the Section View command, the model will remain sectioned. To exit the section view mode click on the Section View command again.

Design Workbook Using SOLIDWORKS 2020

The last thing to do is to define the manifold's material. In the FeatureManager right click in **Material**, select **Edit Material**, expand the **Other Metals** section and set the material to **Titanium**.

Save as **MANIFOLD.sldprt** in your personal folder and make a new drawing with an **Isometric** view using the **Shaded with Edges** display mode. Add the missing notes and save the finished drawing.



Exercise 3.3: HAND WHEEL

In this Exercise you will design a Hand Wheel with an elliptical cross-section, and five spokes. In this exercise you will learn how to make a Circular Pattern, as well as a couple other commands.

Start a new part using the **ANSI-INCHES** template and save it as **HAND WHEEL.sldprt**. Add a new sketch in the **Front** plane and draw a vertical centerline starting at the origin. From the Sketch tab select the **Ellipse** command and draw it as shown in **Figure 3-22**.

To draw the ellipse, click to locate the center, click to locate the minor axis horizontally to the left (or right) or the center, and finally click to locate the major axis vertically above (or below) the center.

After adding the dimensions indicated you need to add a **Horizontal** relation between the horizontal axis reference points (or **Vertical** between the vertical axis reference points), and finally a **Horizontal** relation between the ellipse's center and the origin.

From the Features tab select the **Revolve Boss/Base** command. In the PropertyManager set the centerline is selected, and the default angle of **360°** is set. Click **OK** to finish the first feature as shown in **Figure 3-23**.

The next step is to add one spoke going from the origin out to the wheel. To make this spoke, which also has an elliptical cross section, change to a **Front** view orientation, select the **Front** plane in the FeatureManager and add a new sketch.

Next select the **Ellipse** command and start its center in the origin, dimension it and add a **Horizontal** relation between the major axis endpoints, as shown in **Figure 3-24**.

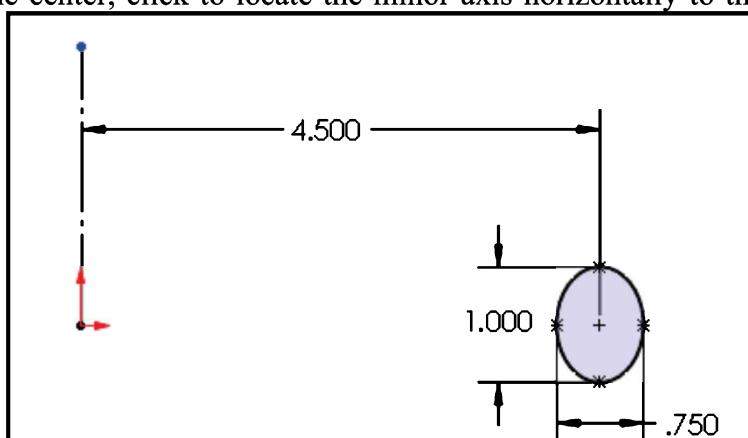


Figure 3-22. Initial Sketch Ellipse Dimensions.

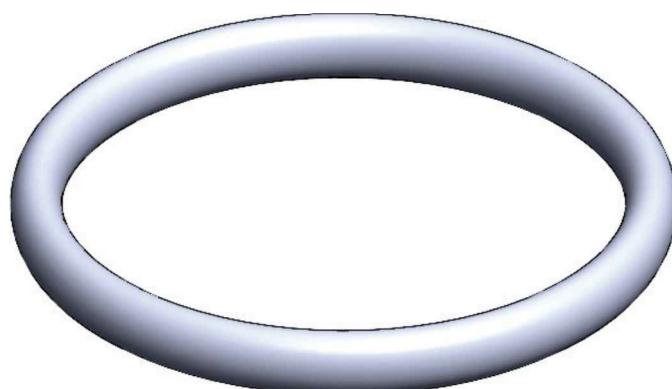


Figure 3-23. The First Feature of the Hand Wheel.

Select the **Extrude Boss/Base** command, set the **Direction 1** end condition to **Up to Next** and reverse the direction. This end condition will make the extrusion until the next model's face. Change to an **Isometric** view orientation for better visualization and click **OK** to finish. The first spoke should look like **Figure 3-25**.

With the first spoke finished you can make a **Circular Pattern** to copy the rest of the spokes. In order to make the pattern, first you need to have a reference to use as the direction axis.

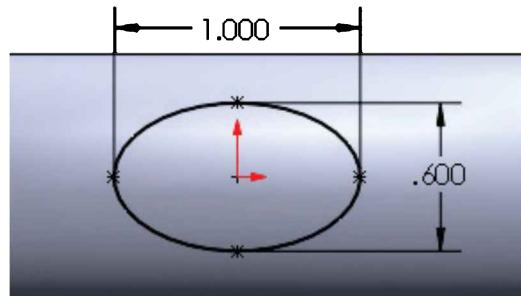
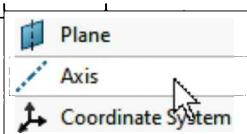


Figure 3-24. Cross Section Dimensions of the Elliptical Spoke.



From the Features tab select the **Reference Geometry** command and click on **Axis**, or from the menu **Insert, Reference Geometry, Axis**. In the **Axis** PropertyManager select the **Front** and **Right** planes from the fly-out FeatureManager, note the axis preview at the intersection of the two planes and click **OK** to finish.

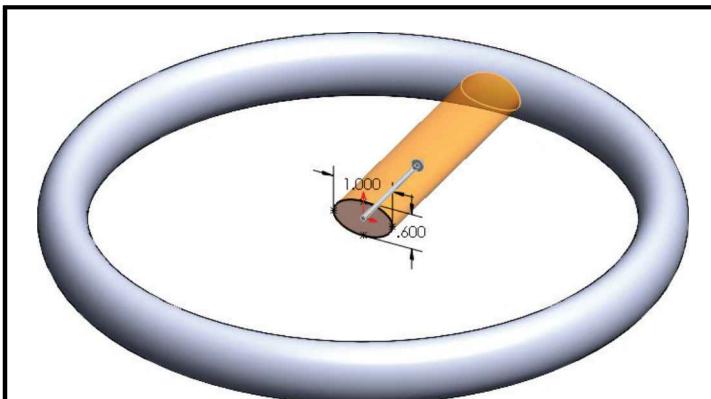


Figure 3-25. The Extruded Spoke.

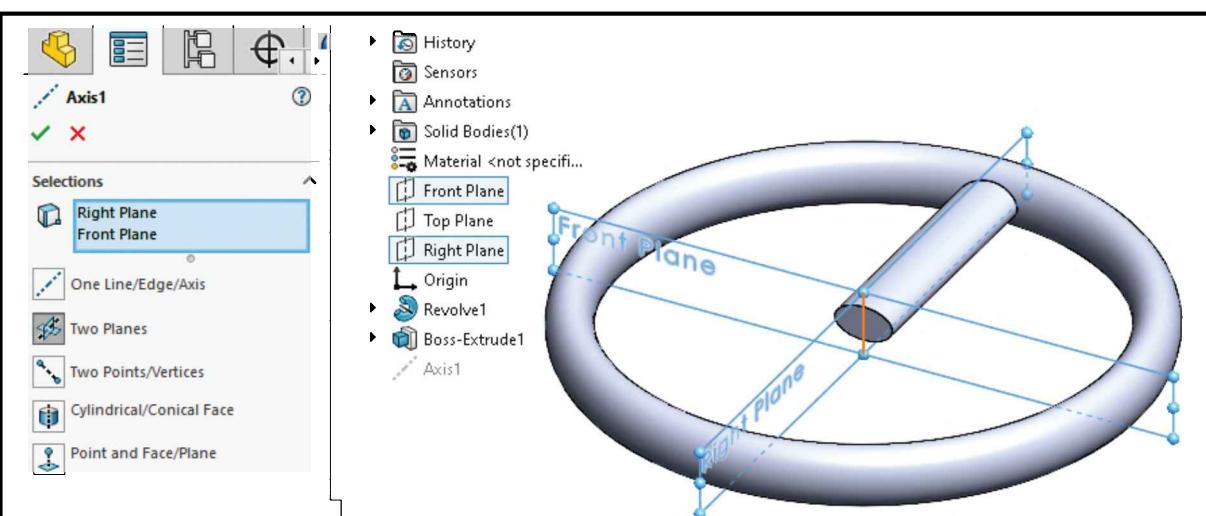
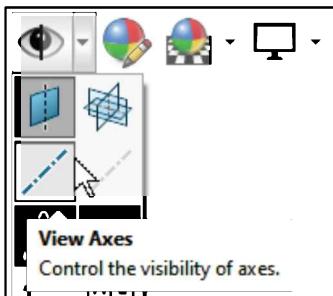
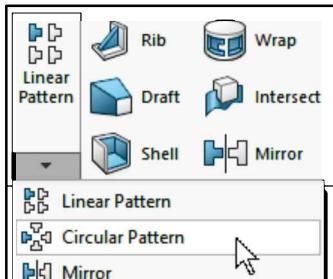


Figure 3-26. Adding a Reference Axis Using Two Planes.

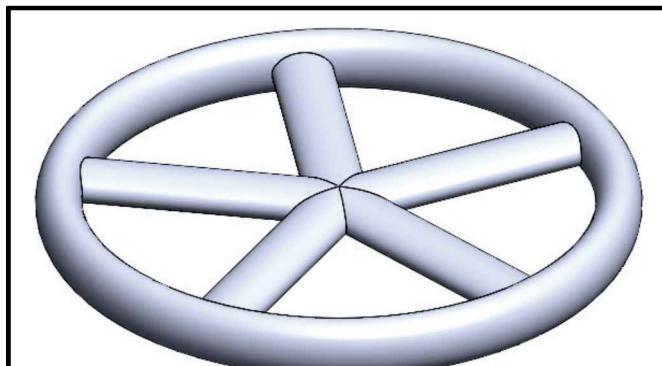


If the new **Axis1** feature is not visible on the screen, select the **Hide/Show Items** drop-down command in the **View** toolbar, and turn on the **Axis** features.

Now that you have the necessary reference, from the Features tab select the **Linear Pattern** drop-down and click on **Circular Pattern**, or from the menu **Insert, Pattern/Mirror, Circular Pattern**.



In the **Circular Pattern** command, add the **Axis1** feature in the **Direction 1** selection box from the fly-out FeatureManager, and add the spoke in the **Features and Faces** selection box. Set the number of copies to **5** and click **OK** to finish.



The complete circular pattern should look as shown in **Figure 3-27**. In the **Hide/Show** command turn off the **View Axis** option.

The next step is to add the hub of the Hand Wheel. Select the **Top** plane in the FeatureManager and add a new sketch on it.

Change to a **Top** view orientation for better visibility and draw a circle centered at the origin. Dimension its diameter **2.000"** to fully define it. Change to an **Isometric** view and select the **Extrude Boss/Base** command. Set the **Direction 1** end condition to **Mid Plane**, change the extrusion's depth to **1.500"** and click **OK** to finish.

Your part with the hub is shown in **Figure 3-28**. Notice the extrusion was made in both directions, symmetrical about the **Top** plane.

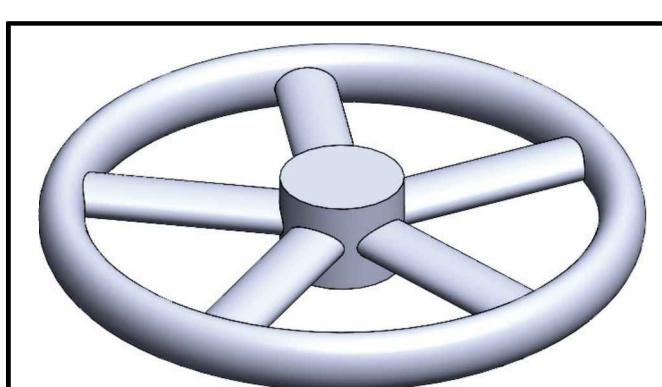
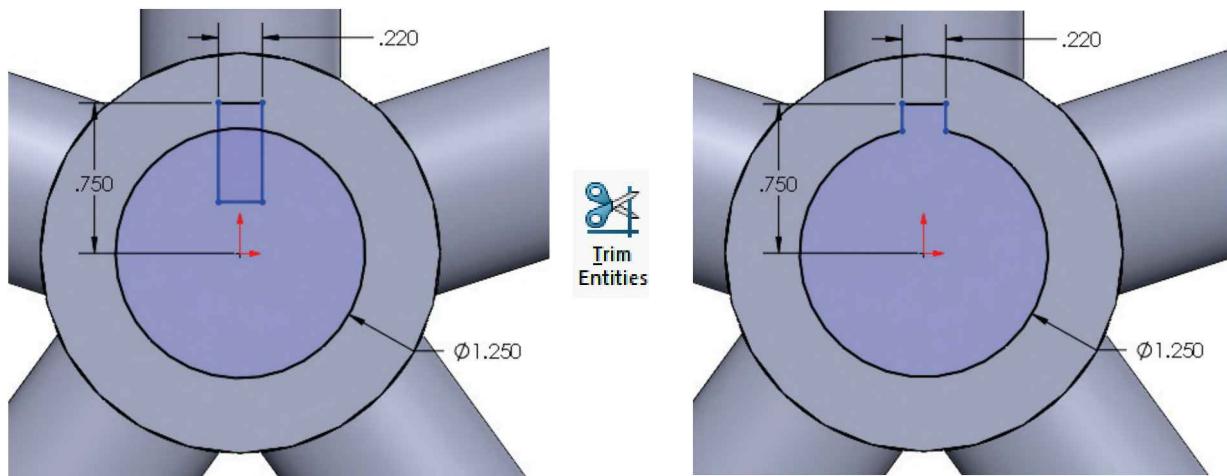


Figure 3-28. Central Hub Added.

Now you need to add a hole and keyway through the hub to fit a shaft and key. Change to a **Top** view for visibility, select the top face of the hub and add a new sketch. Draw a circle at the origin, and a rectangle as shown, and use the **Trim** tool to make the closed profile.



Since the keyway cut must be centered in the shaft, in order to maintain the design intent, you will use a **Midpoint** geometric relation to keep it centered. Draw a Vertical centerline starting at the origin going up almost to the circle. From the **Display/Delete Relations** drop down command select **Add Relations**. Select the top endpoint of the centerline and the horizontal line and add a **Midpoint** relation to fully define your sketch as shown in **Figure 3-29**, correctly maintaining the design intent.

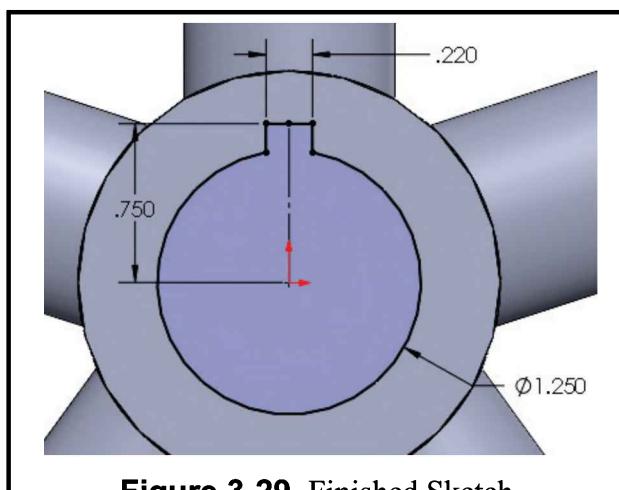


Figure 3-29. Finished Sketch.

With the sketch profile complete, select the **Extruded Cut** command, set the **Direction 1** end condition to **Through All** and click **OK** to finish.

Now you must add a fillet at the intersections of the spokes and the hub, and the spokes and the hand wheel.



Select the **Fillet** command from the **Features** tab, or the menu **Insert, Features, Fillet**. Set the radius to **0.125"** and select the faces of the **5** spokes. By selecting the faces, you will add a fillet to all 10 edges at the same time.

The finished Hand Wheel model is shown in **Figure 3-30**.

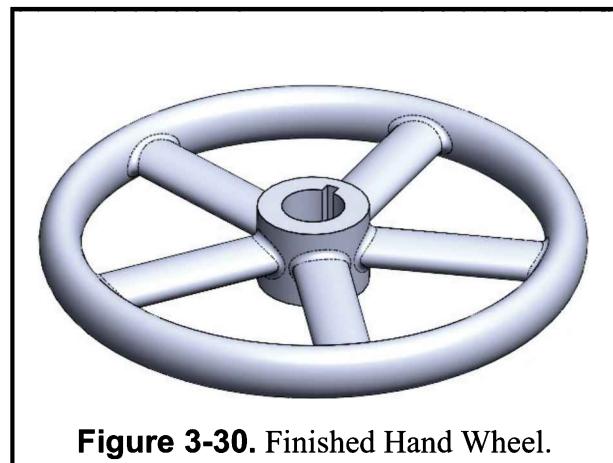


Figure 3-30. Finished Hand Wheel.

Select the part as **HAND WHEEL.sldprt** in your personal folder. Make a new drawing, add a shaded Isometric view as in previous exercises and add the missing notes. The finished drawing is shown in **Figure 3-31**. Finally print a copy for your lab instructor. Save your drawing as **HAND WHEEL.slddrw**.

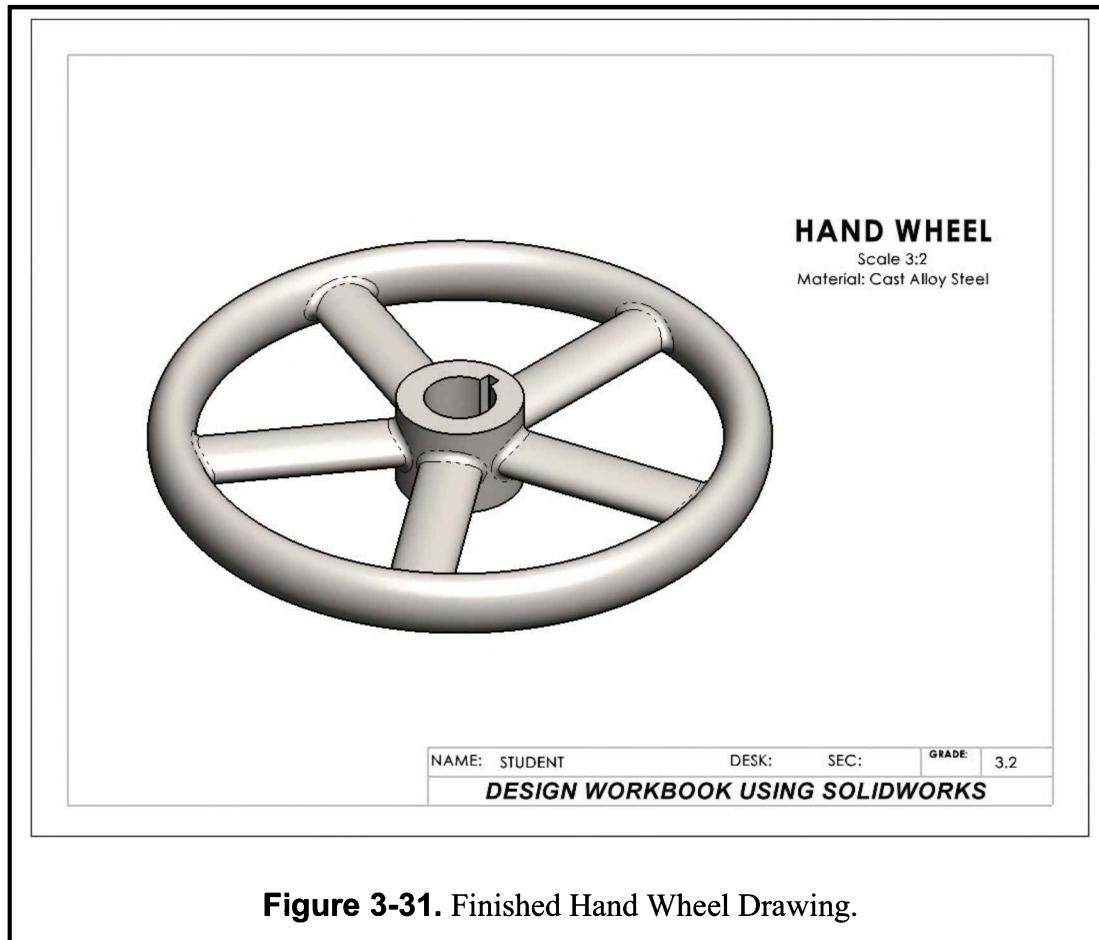


Figure 3-31. Finished Hand Wheel Drawing.

Exercise 3.4: TOE CLAMP

In this exercise you will practice previously learned concepts, and practice working on non-orthogonal planes while designing the Toe Clamp part.

Start by making a new part using the **ANSI-INCHES.prtdot** template and save the new part as **TOE CLAMP**. Select the **Front** plane in the FeatureManager and add a new sketch. Draw the profile shown in **Figure 3-32**.

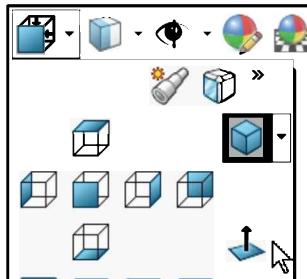
Adding these dimensions will fully define the sketch, with the origin in the lower left corner. To add the angle dimension, select the horizontal and inclined lines, and locate the dimension.

When the sketch is complete, select the **Extrude Boss/Base**

command, set the **Direction 1** end condition to **Mid Plane**, make the distance **2.00"** and click **OK** to finish. The base feature is shown in **Figure 3-33**.

Now you will cut the counterbore and hole on the inclined face. To make it easier to work on this face you want to orient the part parallel to the screen.

To get this orientation select the face and click in the **Normal to** command in the View Orientation box, or in the context menu after the face is selected.



Now it will be easier to add the counterbore and hole on the inclined face. After the part is reoriented, select the inclined face and add a new sketch. Draw a circle and dimension it as shown in **Figure 3-34**. To fully define the sketch, add a **Vertical** geometric relation between the center of the circle and the part's origin.

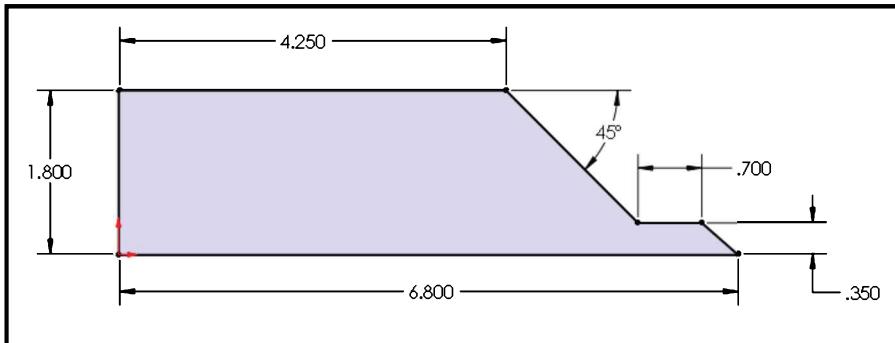


Figure 3-32. Initial Sketch for the Toe Clamp.

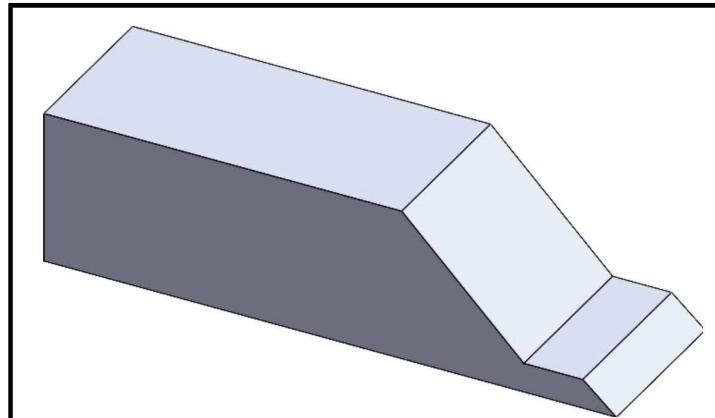


Figure 3-33. The Base Part for the Toe Clamp.

Select the **Extruded Cut** command, set the end condition to **Through all** and click **OK** to finish. The hole will be made perpendicular to the sketch face.

After adding the first hole, select the same angled face, add a new sketch and draw a circle concentric with the previous hole. Add a **0.80"** diameter dimension. Select the **Extruded Cut** command and make a cut **0.25"** deep using the **Blind** end condition to make the counterbore. Your part will look as **Figure 3-35**.

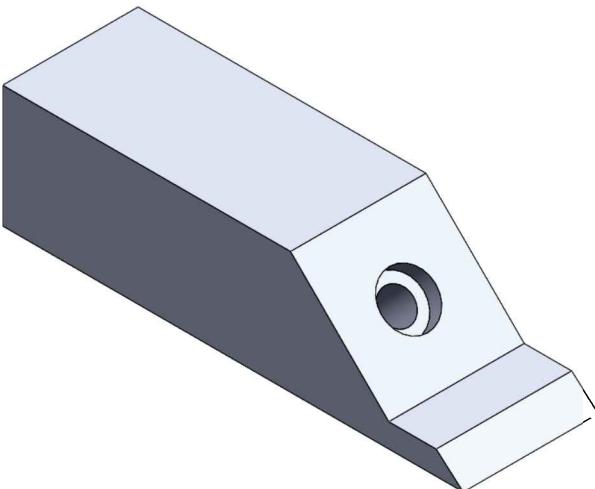


Figure 3-35. Completed Counterbore Hole.

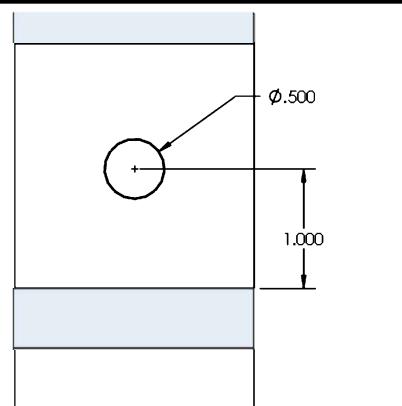


Figure 3-34. Sketch on Angled Face.

You can now make the V-cut to add stress relief for the Toe Clamp. This cut can be added to any of the horizontal faces, but for this exercise we'll use the flat face between the angled faces.

Change to a **Top** view orientation and add a new sketch on the flat face. To locate the cut centered in the part, draw a horizontal centerline starting at the middle of the edge, and then draw the triangle approximately as shown in **Figure 3-36**.

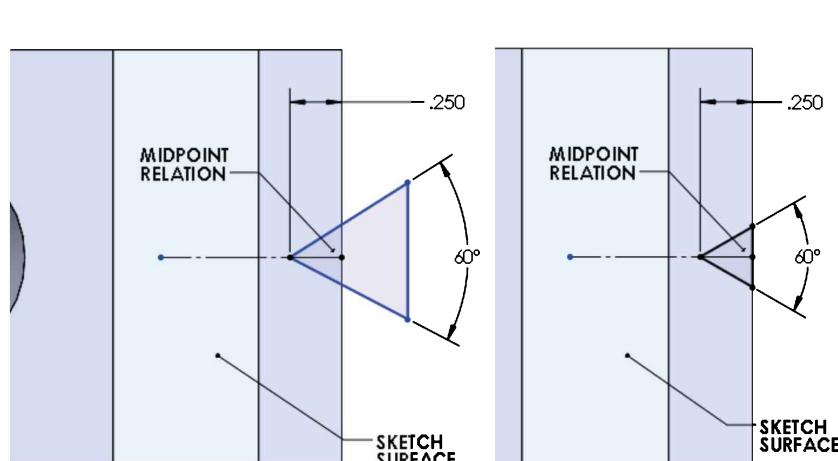


Figure 3-36. Sketch for the V-Cut.

The easiest option to fully define the sketch and maintain the design intent is to add a **Midpoint** relation between the vertical line of the triangle and the endpoint of the centerline you added first.

After you add the relation, the sketch is fully defined, and regardless of which dimension is changed, the V-Cut will remain at the center of the part and symmetrical.

Now you can make the V-Cut. Select the **Extruded Cut** command from the Features tab and use the **Through all** end condition. Make sure the cut goes into the part through the beveled end, and then click **OK** to finish.

The next step is to add the side slots. Change to a **Right** view orientation and add a new sketch on the **Right** plane. The design intent requires the slots to be symmetrical about the part's center. To accomplish this, you will use the **Dynamic Mirror** command. Draw a vertical centerline starting at the origin, select it and go to the menu **Tools**, **Sketch Tools**, **Dynamic Mirror**. Now use the Rectangle tool and draw and dimension one side of the sketch as shown in **Figure 3-37**. The other side will be mirrored automatically.

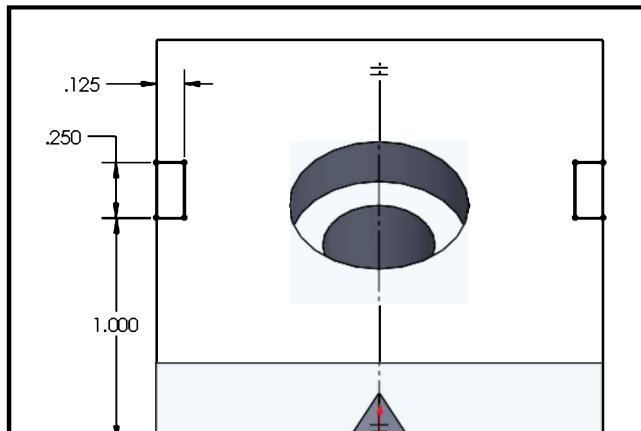


Figure 3-37. Sketch for Grooves.

Select the **Extruded Cut** command and use the **Through All** end condition. The resulting part is shown in **Figure 3-38**.

The last feature to create is the large counter slot on top of the Toe Clamp. Change to a **Top** view, select the top face, and add a new sketch. Using the **Slot** tool draw and dimension a slot as shown in **Figure 3-39**. To fully define the sketch, select the slot's centerline and the origin, and add a **Coincident** geometric relation.

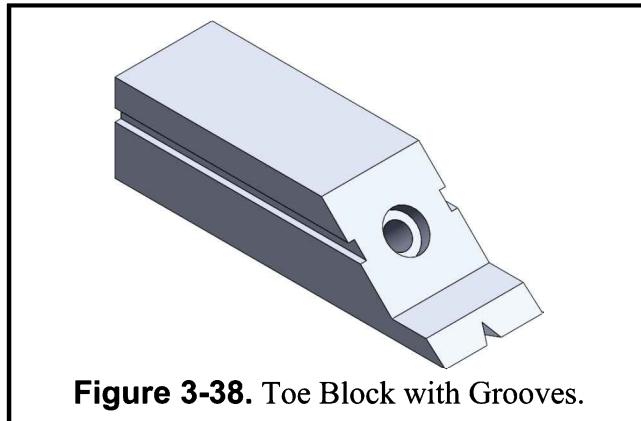


Figure 3-38. Toe Block with Grooves.

Use the **Extruded Cut** command and make a **0.25"** cut using the **Blind** end condition.

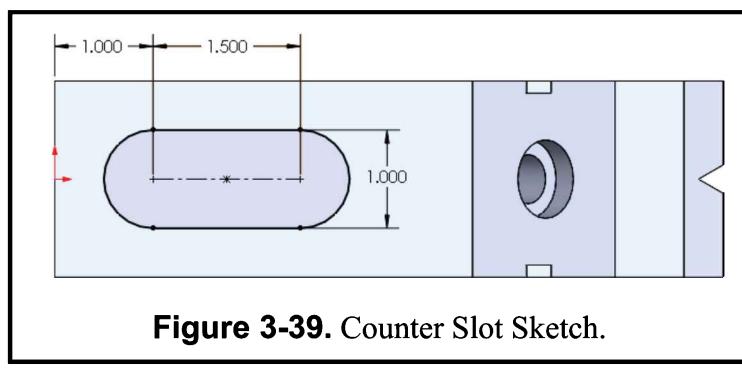


Figure 3-39. Counter Slot Sketch.

To add the last feature, select the bottom face of the slot you just made

and add a new sketch. With the slot face selected, click on the **Offset Entities**

command. Use the **Reverse Direction** option, set the distance to **0.20"** to make a smaller slot and click **OK**. Select the **Extruded Cut** command, set the **Through all** end condition and click **OK** to finish.

The finished Toe Clamp model is shown in **Figure 3-40**.



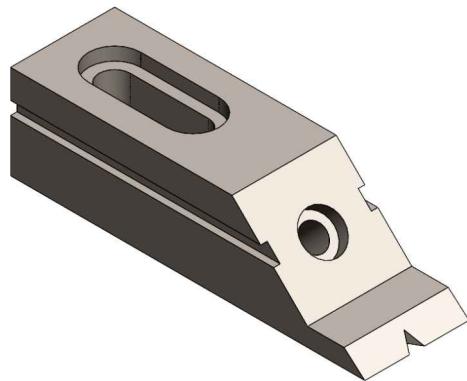


Figure 3-41. Completed Toe Block.

The last thing you need to do is to define the part's material. In the FeatureManager, right click on **Edit Material**, expand the **Steel** materials category, and assign **AISI Type A2 Tool Steel** to the **TOE CLAMP**.

Save your part as **TOE CLAMP**, make a new drawing, add a shaded isometric view and add the missing annotations. Save your drawing as **TOE CLAMP.slddrw**. Print a copy to submit to your lab instructor.

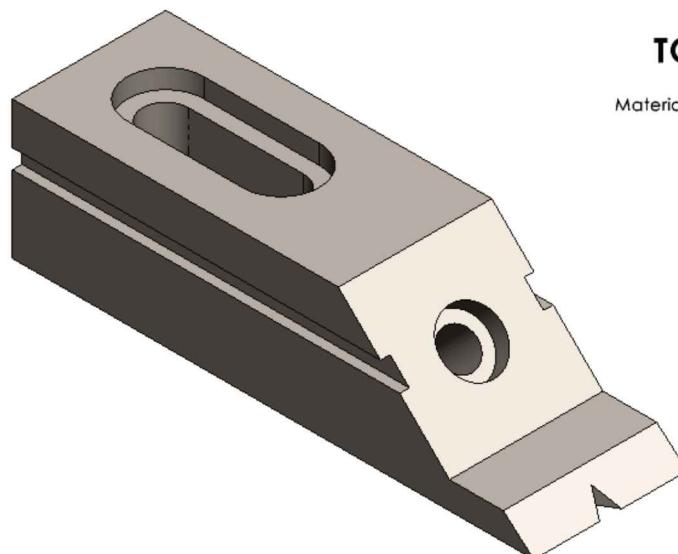
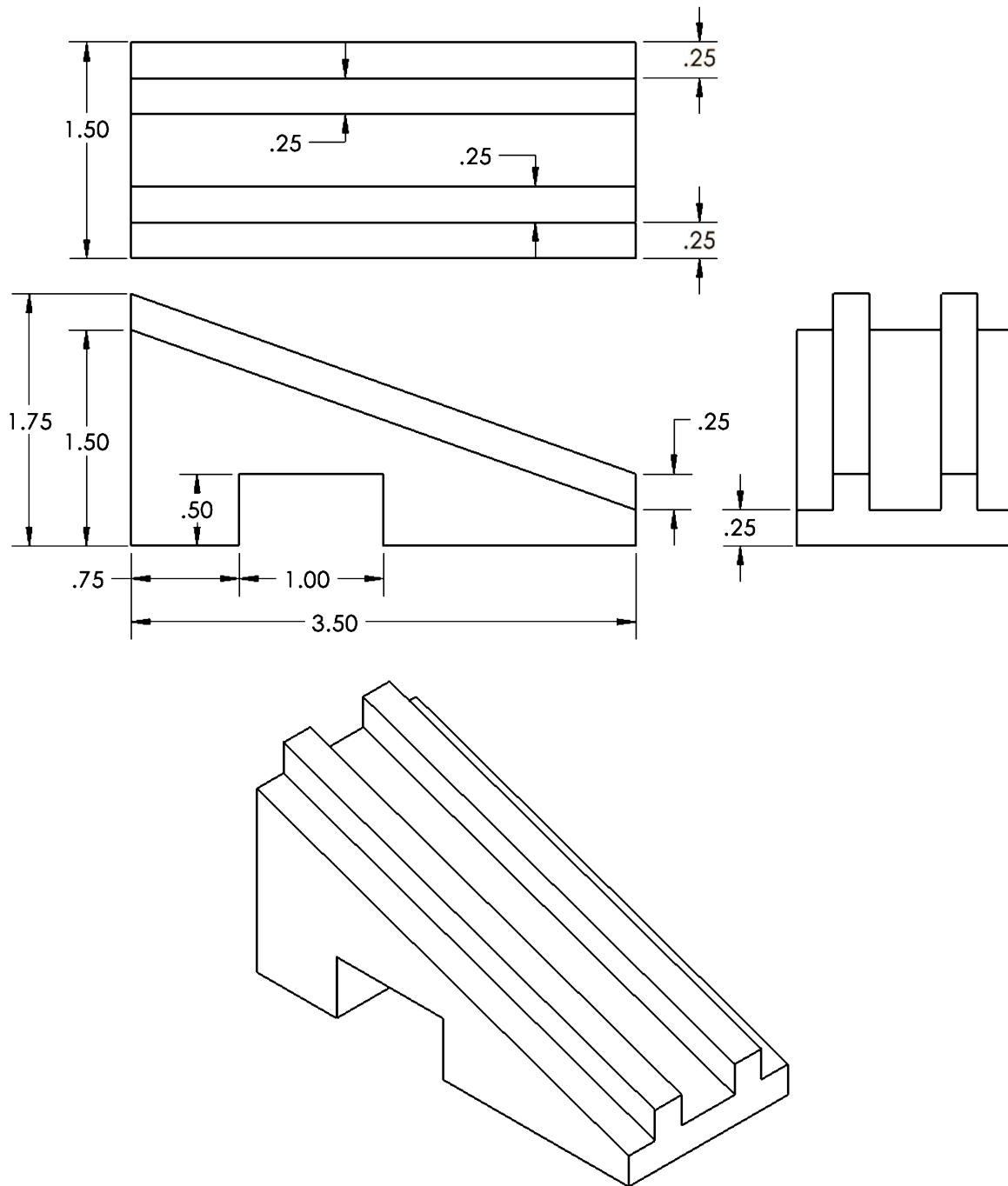


Figure 3-42. Finished Toe Clamp Drawing.

Supplementary Exercise 3-5: CONVEYOR RAMP GUIDE

Build a solid model of the figure below. Make a drawing with an **Isometric** view and name it **“CONVEYOR RAMP GUIDE.”** Use the dimensions indicated below.

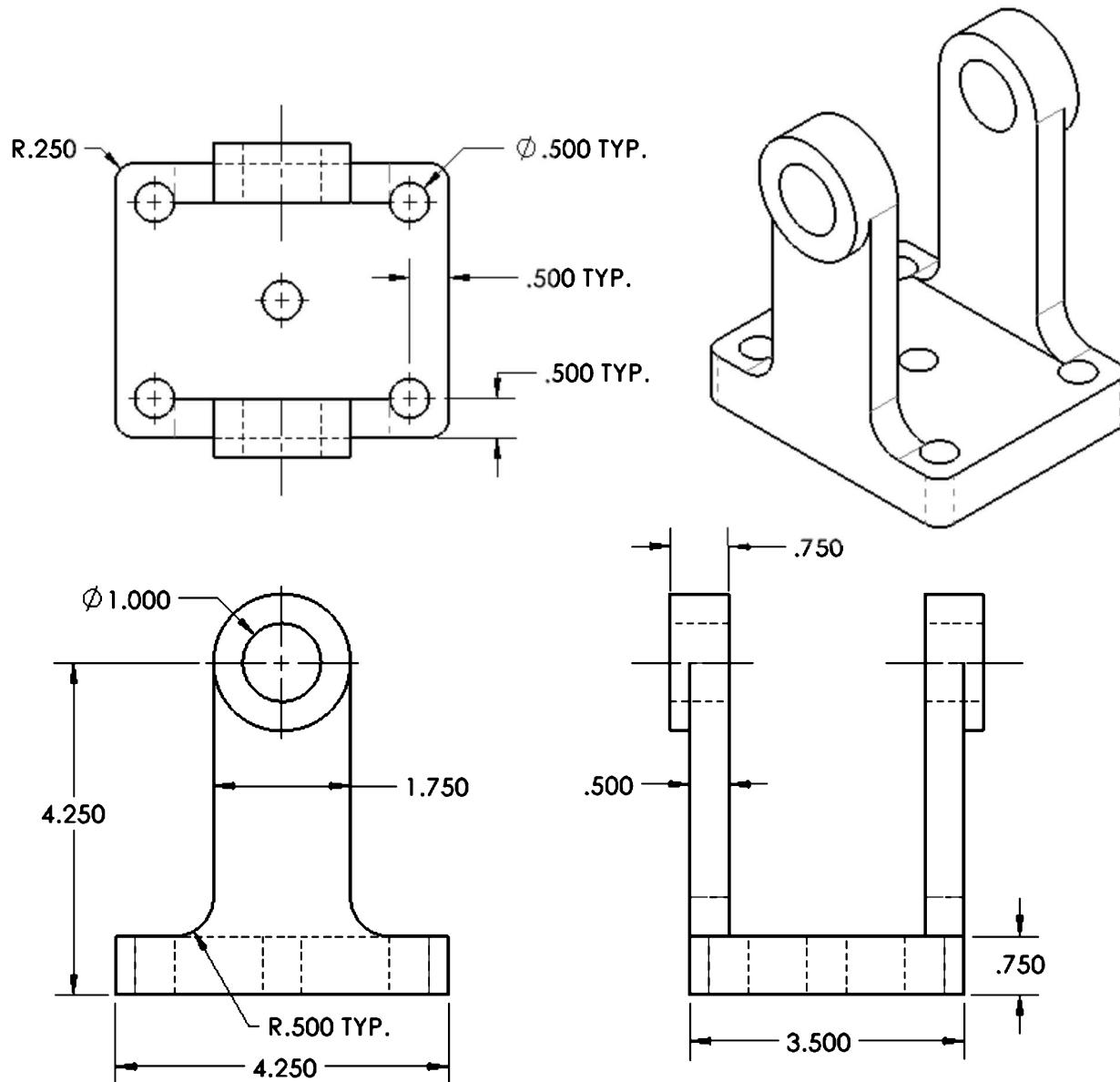


ISOMETRIC VIEW

Supplementary Exercise 3-6: DOUBLE SHAFT HANGER

Using the knowledge acquired so far, make a part of the figure below, make a drawing with an Isometric view and save it as **DOUBLE SHAFT HANGER**.

HINT: The part is symmetrical; make the first rectangle centered at the origin in the **Top** plane.



NOTES:

Design Workbook Lab 4: 3D Modeling Part II

INTRODUCTION TO LAB 4

In this lab you will learn to use advanced features to make 3D models that would be difficult or impossible to make with the prismatic shapes used previously. These are the commands you will be using:

Draft - Creates a taper to the selected model faces by a specified angle, using either a neutral plane or a parting line. **NOTE:** You can also apply a draft angle as a part of an extruded base, boss, or cut feature.

Offset Plane - You can create planes in parts or assemblies. You can use planes to sketch, to create a section view of a model, for a neutral plane in a draft feature, and so on.

Offset - You can create sketch curves offset from one or more selected sketch entities, edges, loops, faces, curves, set of edges, or set of curves by a specified distance. The selected sketch entity can be changed to construction geometry. The offset entities can be bi-directional.

Convert Entities - You can create one or more curves in a sketch by projecting an edge, loop, face, curve, external sketch contour, set of edges, or set of sketch curves onto the sketch plane. You can convert sketched entities into construction geometry to use in creating model geometry.

Face/Edge Fillet - A Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

Shell - The shell hollows out the part, leaving open the selected faces, and creates a thin-walled feature on the remaining faces.

Loft - Creates a 3D feature by making transitions between different profiles. A loft can be a base, boss, cut, or surface.

Dome - Creates a 3D feature that begins with the shape of the selected face and lofts up to a rounded face at a specified height.

Sweep - Creates a base, boss, cut, or surface by sweeping a profile (section) along a path, according to these rules:

- The profile must be closed for a solid sweep feature; the profile may be open or closed for a surface feature.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a curve, or a set of model edges.
- The start point of the path must lie on the plane of the profile.
- The section, the path, or the resulting solid cannot be self-intersecting.

Exercise 4.1: DRAWER TRAY

Start a new part based on the **ANSI-INCHES** template and save it as **DRAWER TRAY.sldprt**. Select the Top plane in the FeatureManager and a new sketch. Your part will be rotated to a Top view automatically.

Use the commands learned so far to draw the profile shown in **Figure 4-2**. Add a sketch Fillet in the corners to finish. Make sure the sketch origin is in the lower left corner.

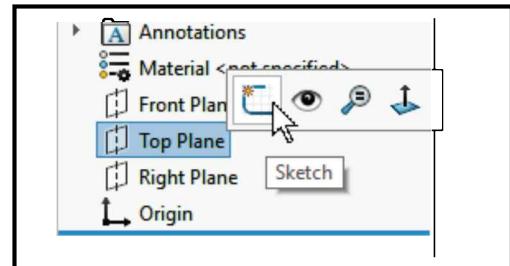


Figure 4-1. Add Sketch to Plane.

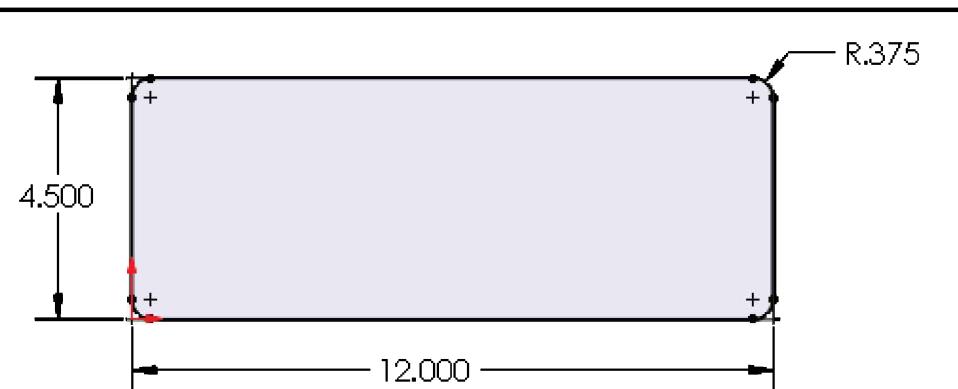


Figure 4-2. First Feature's Sketch.

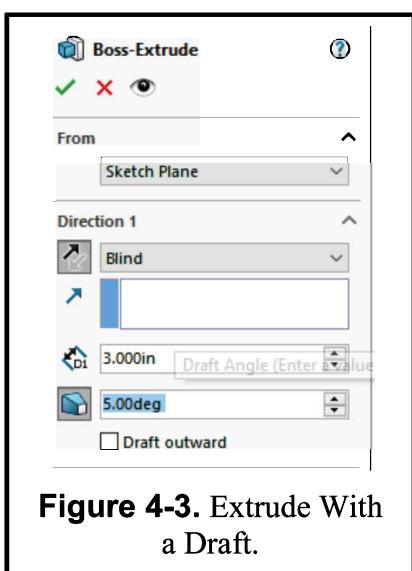


Figure 4-3. Extrude With a Draft.

From the Features tab select the **Extruded Boss** command and make the extrusion **3.000"** deep using the **Blind** end condition. If needed, activate the **Reverse Direction** option to make the extrusion go downward. Activate the **Draft Angle** option and set the angle to **5°**, as shown in **Figure 4-3**, and click **OK** to finish. The first feature looks like **Figure 4-4**.

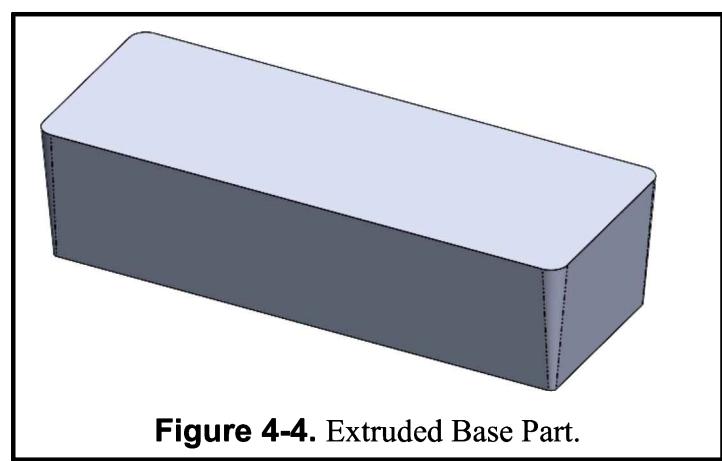


Figure 4-4. Extruded Base Part.

SOLIDWORKS allows you to rename features in the FeatureManager to make it easier to identify them later. To rename the **Boss-Extrude1** feature, slow double-click on it and rename it “**Tray Body**.”



The next step is to add fillets to the bottom of the part. To rotate the view, use the **Middle Mouse Button** and drag the mouse, or use the arrow keys until you see the bottom face. From the Features tab select the **Fillet** command, change the radius to **0.375"**, select the bottom face of the “**Tray Body**” feature and click **OK** to finish. By selecting the face, all its edges will be rounded.



To create a hollow part change to an **Isometric** view orientation, select the **Shell** command from the Features tab, in the Parameters section set the wall thickness to **0.10"** and select the top face of the part. This is the face that will be removed from the model, and every other face will have the specified thickness. Click **OK** to finish. Your shelled part will look like **Figure 4-5**.

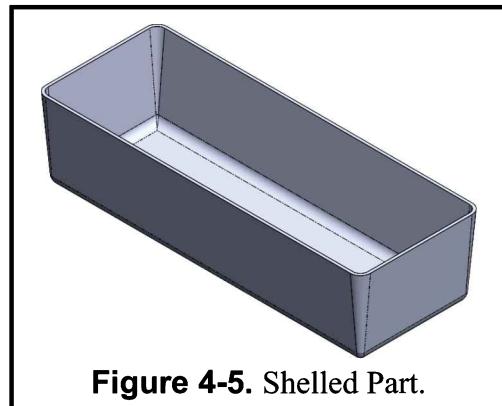


Figure 4-5. Shelled Part.



To reinforce the tray, you will add a rim around the top edge. To do this you have to add a new sketch in the top thin face of the part. To make the selection of this face easier, you can use the **Face Selection Filter**. Press the **F5** key (default shortcut) to display the selection filter toolbar and activate the **Faces** filter. Optionally you can press the **X** key (default shortcut) to activate/deactivate the **Face Selection Filter**.



As long as the selection filter is active you can only select faces. Your mouse cursor will have a small funnel to indicate a selection filter is active.

NOTE: If you don't use the selection filter you have to zoom in close to select the face.

After selecting the top thin face add a new sketch as seen in **Figure 4-6**, and immediately turn off the selection filter. If you don't turn it off, you will not be able to select other entities.

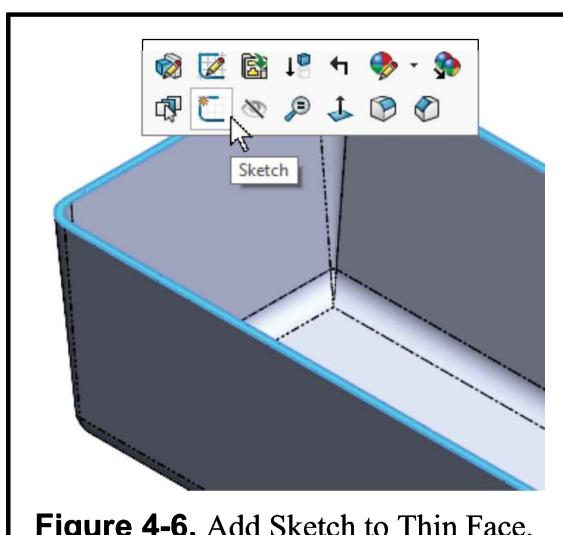


Figure 4-6. Add Sketch to Thin Face.



While the top face is selected, click on the **Convert Entities** command to project the outer edges of the selected face onto new sketch entities.



Now select the **Offset Entities** command and select one of the previously projected sketch lines. Make sure the **Select Chain** option is activated to select the entire profile. If the preview is shown outside, turn on the **Reverse** option to add the offset inside. Set the distance to **0.10"** as shown in **Figure 4-7** and click **OK** to finish.

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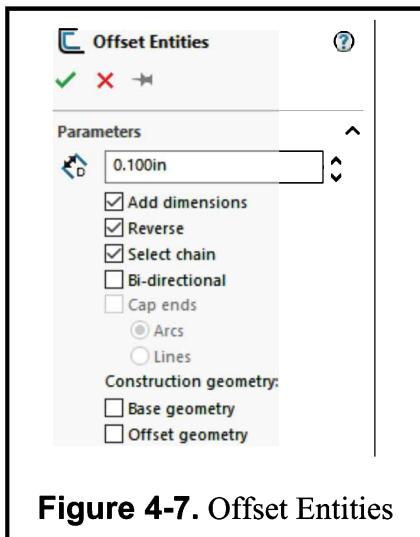


Figure 4-7. Offset Entities

select the **Plane** command.

Select the **Extruded Boss** command and make a **Blind** extrusion going down. Set the distance to **0.625"** and click **OK** to finish to create the vertical rim around the upper edge of the tray.

The next step is to add three equally spaced dividers inside the tray. They will be located **0.25"** below the top face of the tray.

From the Features tab, click in the **Reference Geometry** command and

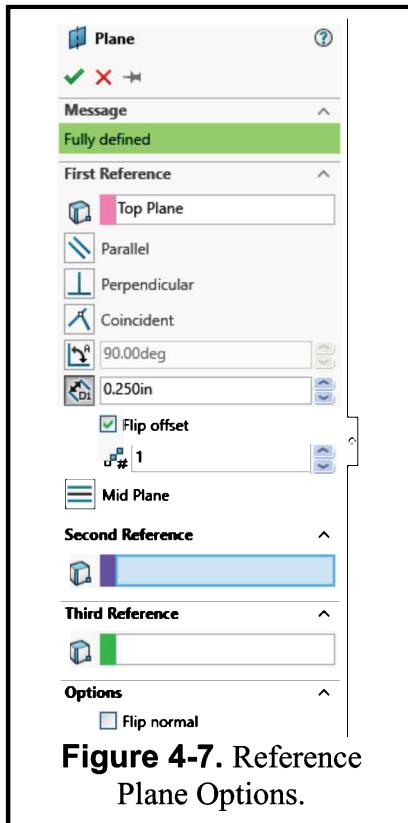


Figure 4-7. Reference Plane Options.

From the fly out FeatureManager select the **Top** plane, set the distance to **0.25"** and use the **Flip Offset** option to add the plane below the top face, as seen in **Figure 4-7**, and click **OK** to finish.

Now slow double click to rename the new plane in the FeatureManager and change its name to **Dividers Plane**.

To create the divisions, you will use the **Rib** command. This command uses a sketch with a **single open profile**, or **multiple open profiles**, and extrudes it in the designated direction, defining the width and direction of the extrusion and optionally adding a draft.

Change to a Top view and add a new sketch in the Dividers Plane just created. Draw a single line as indicated in **Figure 4-8**.

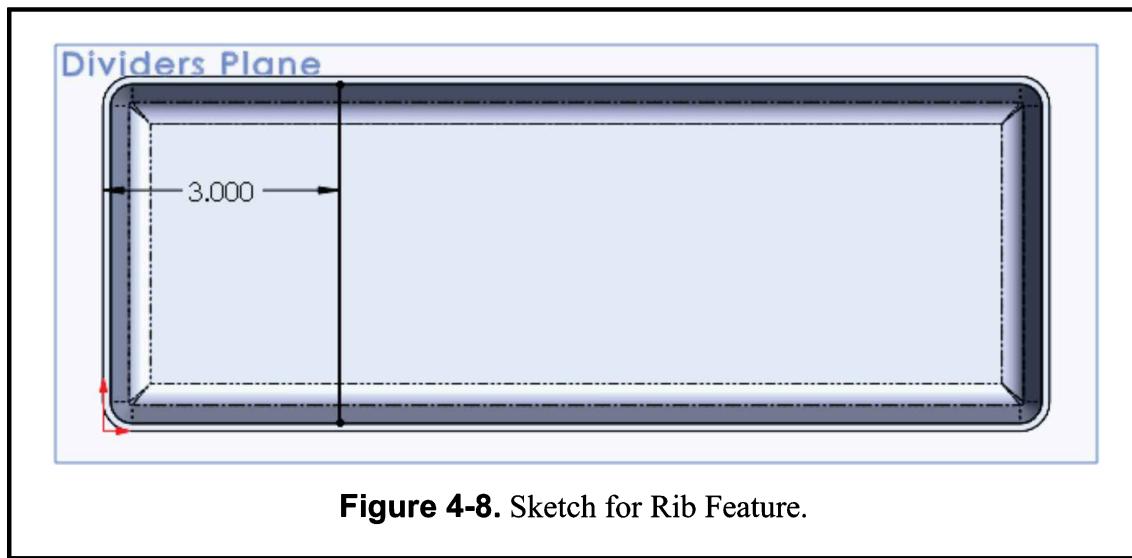


Figure 4-8. Sketch for Rib Feature.



Rib From the Features tab select the **Rib** command. In the Parameters section set the **Thickness** to Both Sides, set the **Rib Thickness** to **0.10"** and, if needed, change the **Extrusion Direction** to Normal to Sketch. Make sure the arrow is pointing down into the part. Turn on the **Draft** option, set the draft angle to **2°** and make sure the **Draft outward** option is checked, as shown in **Figure 4-9**.

Click **OK** to finish; the completed rib is seen next.

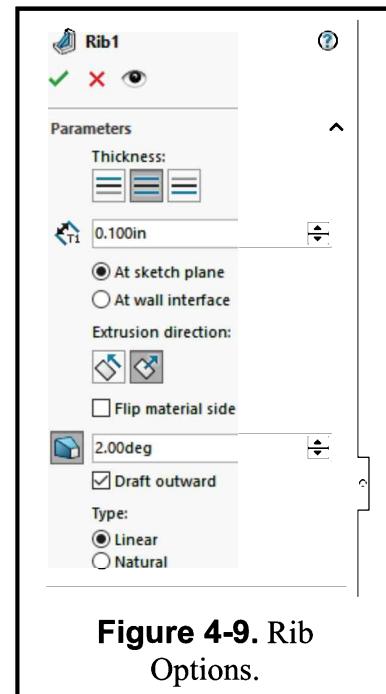
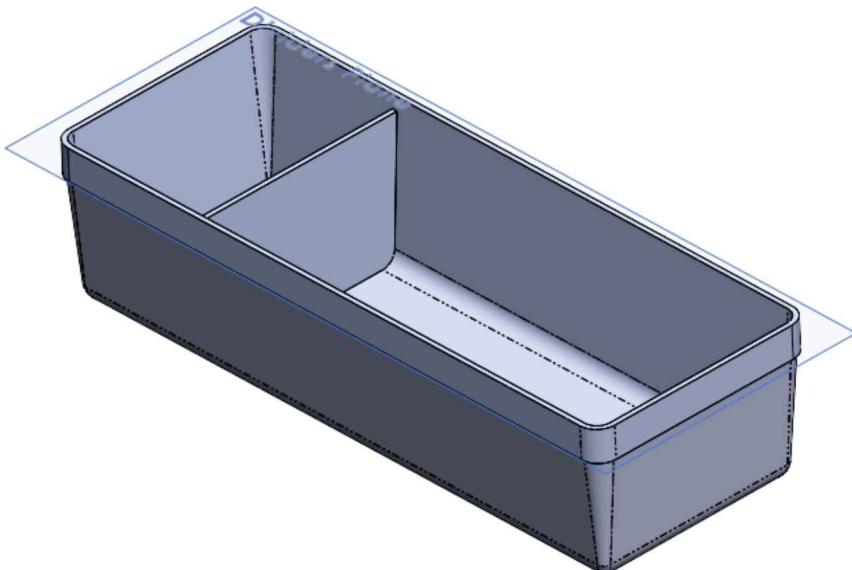


Figure 4-9. Rib Options.



To add the rest of the ribs, use the **Linear Pattern** command, select the long edge of the tray for the **Direction 1**, set the number of copies to **3** and make the spacing equal to **3.00"**. In the Features and Faces selection box add the Rib1 and click **OK** to finish. In the **Hide/Show** Items command turn off the auxiliary plane's visibility. The finished tray is shown in **Figure 4-10**.

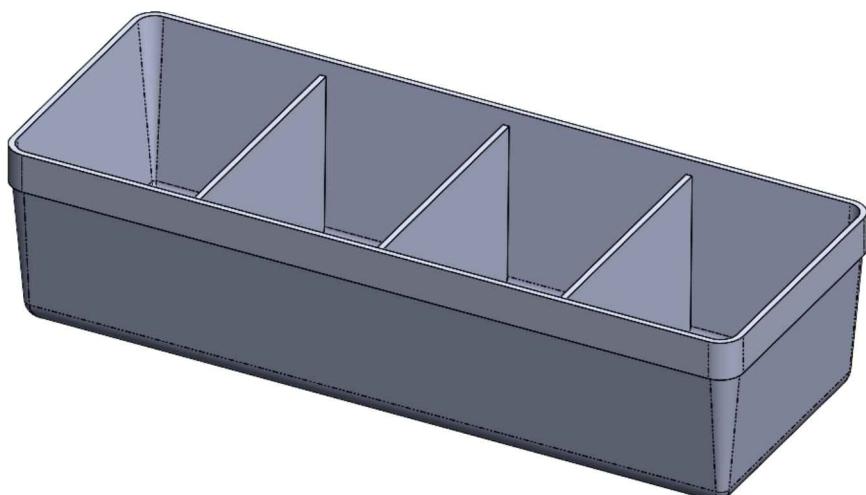
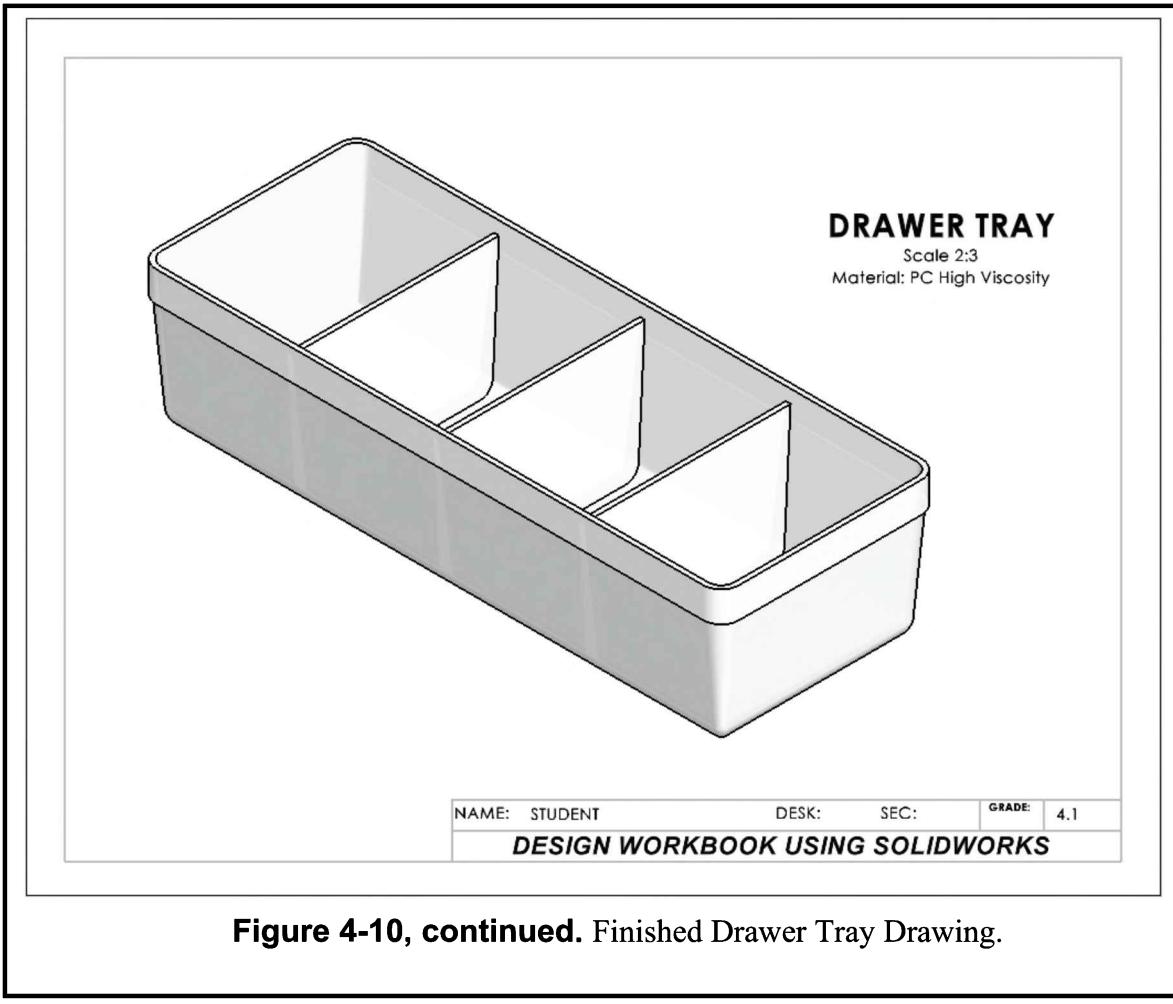


Figure 4-10. Completed Drawer Tray.

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Right click on **Material** in the FeatureManager. Select **Edit Material**, expand the **Plastics** Tab and apply the **PC High Viscosity** material for the part. Save your model as **DRAWER TRAY**.

To finish this exercise, make a new drawing, add an Isometric using the Shaded with Edges display mode and add the missing notes. Save your drawing as **DRAWER TRAY.slddrw**. Print a copy for your instructor. The drawing should look like **Figure 4-10**.



Exercise 4.2: TAP-LIGHT DOME

Start a new part using the **ANSI-INCHES** template and save it as **TAP-LIGHT DOME**. Add a new sketch in the **Top** plane and draw a circle centered in the origin with a diameter of **4.70"**. After the sketch is complete make an **Extruded Boss** command **0.375"** thick, as shown in **Figure 4-11**.

In the FeatureManager rename the **Boss-Extrude1** feature as “**Dome Base**.”

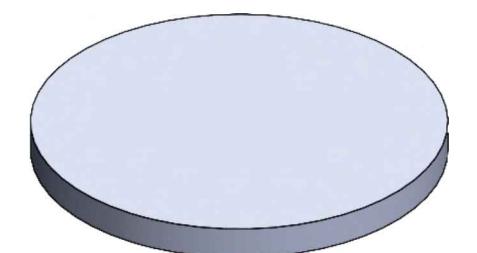


Figure 4-11. The Base Feature.

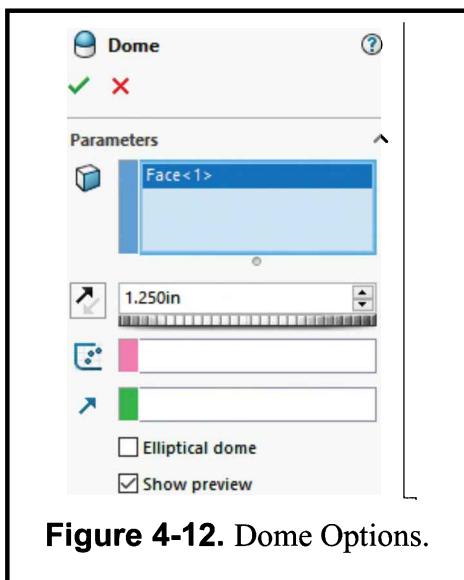


Figure 4-12. Dome Options.

Now you need to add the dome on top of the disk. Select the menu **Insert, Features, Dome**. In the **Faces to Dome** selection list add the top face of the base feature, set the **Dome Height** to **1.20"** as shown in **Figure 4-12** and click **OK**. Your part will look like **Figure 4-13**.

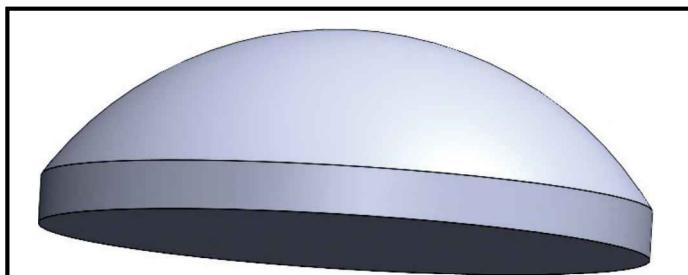


Figure 4-13. Dome Feature on Top of Base.

 **Shell** The next step is to make the part hollow. From the Features tab select the **Shell** command, rotate the part, and select the bottom flat face to remove it. In the Parameters section, set the shell thickness to **0.075"** and click **OK** to finish.

Now you need to add a lip at the bottom of the part. Change to a **Bottom** view orientation, select the thin face at the bottom of the part and add a new sketch. If needed, use the **Face** selection filter (Shortcut **X**) to make selection easier.



With the bottom face selected, click in the **Convert Entities** command to project the outside edge of the face.



Now click on the **Offset Entities** command, set the offset distance to **0.20"** and select the previously projected outside edge. Make sure the offset is added outside the part and click **OK** to finish. The sketch will look like **Figure 4-14**.

Select the **Extruded Boss** command and make a **0.075"** thick boss going up. Change to an **Isometric** view to visualize the extrusion's direction and click **OK**. The part so far will look like **Figure 4-15**.



Figure 4-15. Dome with the Finished Lip.

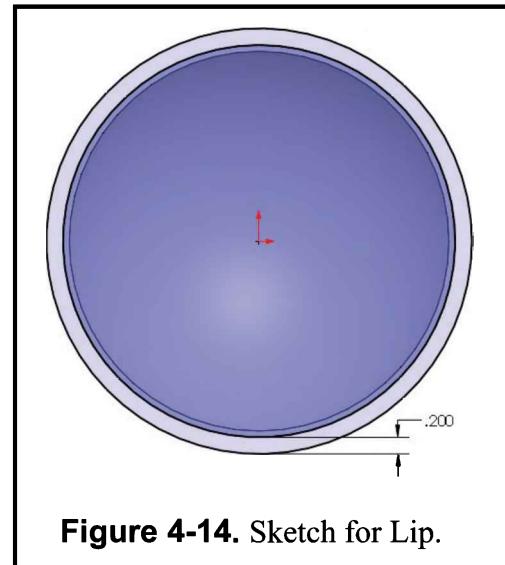


Figure 4-14. Sketch for Lip.

In the next step you will add four holes in the lip of the dome for fastening screws to hold the body and the lens together. The holes in the lip of the dome are placed at uneven angles. This is done to ensure that the dome can only be placed on the body of the light fixture in one position.

Change to a **Top** view, select the top face of the lip, add a new sketch, and draw a vertical centerline starting at the origin. Next add **four** construction lines starting at the origin radially out, making sure not to capture any automatic geometric relations; in other words, do not snap a line to any automatic reference and dimension their angle to the vertical as **Figure 4-16**.

Next draw a circle starting at the origin, and while it is selected, activate the **For construction** option in the properties, and finally dimension the diameter **5.05"**. This construction bolt circle will be used to locate the screw holes in the lip, since it is not the same diameter as the outer size of the lip.

Draw **four** circles at the intersections of the construction circle and the radial lines, after you select the **Circle** command you will see an inference icon next to the cursor when you approach an intersection to start each circle. Dimension one of them **0.230"** diameter and add an **Equal** relation between all four circles to fully define the sketch. Your sketch should look like **Figure 4-16**. Now use the Extruded Cut command and make a cut using the Through All end condition. The finished part is shown in **Figure 4-17**.

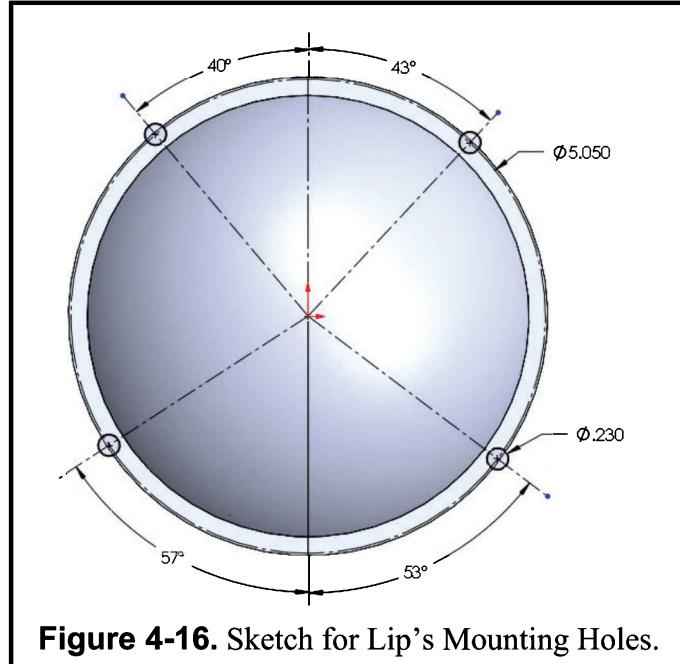
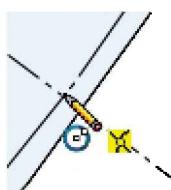


Figure 4-16. Sketch for Lip's Mounting Holes.

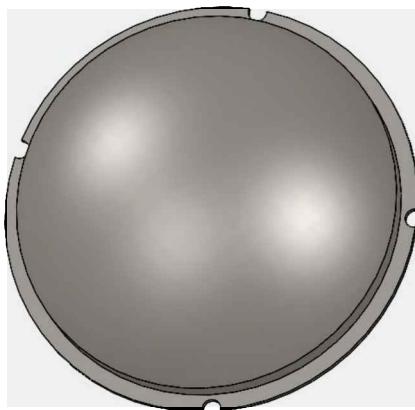
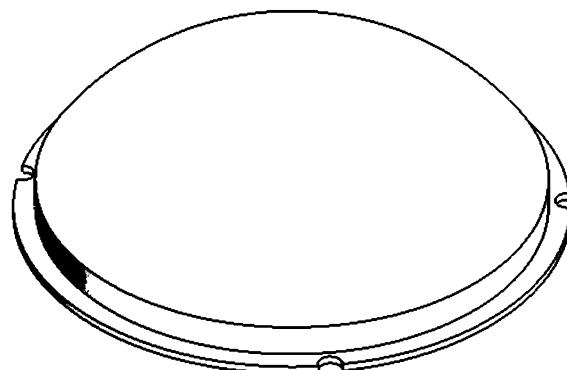


Figure 4-17. The Finished Tap-Light.

To assign a material to your part Right click on **Material**, select **Edit Material**, from the Plastics section select **ABS** and apply the material to your part.

Save your part as **Tap-Light Dome.sldprt**, make a new drawing with an **Isometric** view, add the missing annotations and save it as **Tap-Light Dome.slddrw**. Your final drawing should look like **Figure 4-18**.



TAP-LIGHT DOME

Scale 1:1

Material: ABS

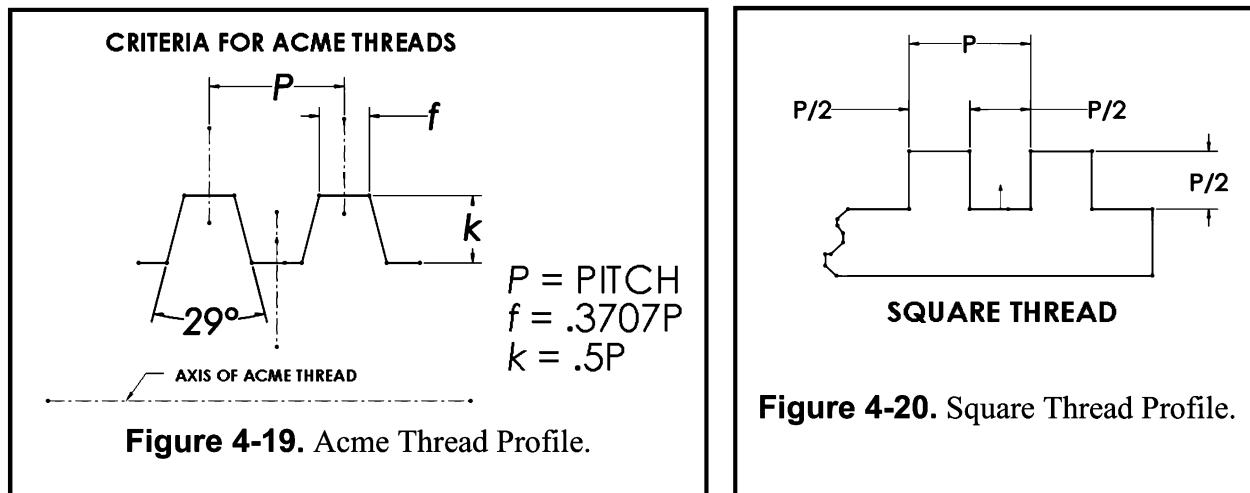
NAME: STUDENT	DESK:	SEC:	GRADE:	4.2
DESIGN WORKBOOK USING SOLIDWORKS				

Figure 4-18. Finished Tap-Light Dome Drawing.

Exercise 4.3: THREADS AND FASTENERS

THREAD TYPES

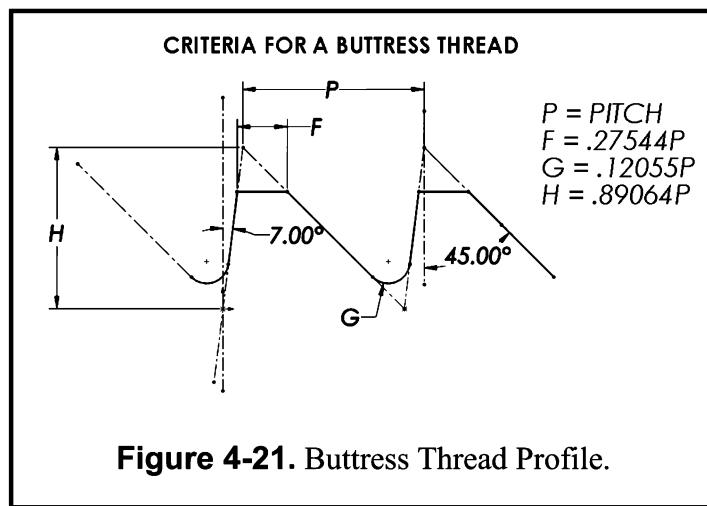
SQUARE THREADS & ACME THREADS These Threads have been used for many years to transmit power either from a turning motion or a linear motion. The uses have increased as more and more processes require this type of translation. Since the square thread is difficult to manufacture, the Acme thread has become the more commonly used thread for the purpose of transmitting power. **Figure 4-19** shows the profile of an Acme Thread and **Figure 4-20** shows a Square Thread profile.



BUTTRESS THREAD This thread form is a nonsymmetrical thread that is used where exceptionally high stresses lie along the axis of the threaded shaft. An example of this thread form is shown in **Figure 4-21**.

STANDARD V-THREADS - This thread form is the basis for most nuts and bolts; however, within this thread form there are many deviations that are used for special applications. Several of the most common standards are the American Standards and the Metric

Standards. Within the American standards the most common series are “UNC” or Unified National Course (also known as “NC” National Course); “UNF” or Unified National Fine (also known as “NF” National Fine); and “UNEF” or Unified National Extra Fine (also known as “NEF” National Extra Fine). Within the Metric standards there is a coarse series and a fine series that are distinguished only by the different values of the pitch.



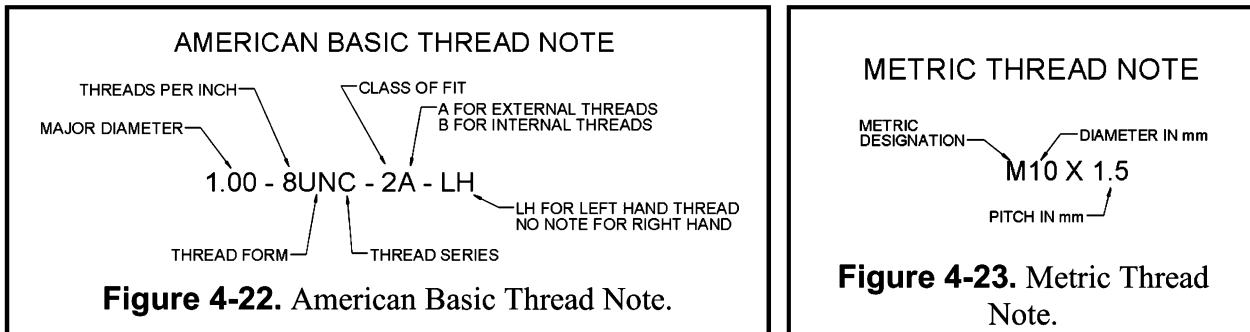
THREAD TERMINOLOGY

Following is a list of terms that are used extensively in association with threads:

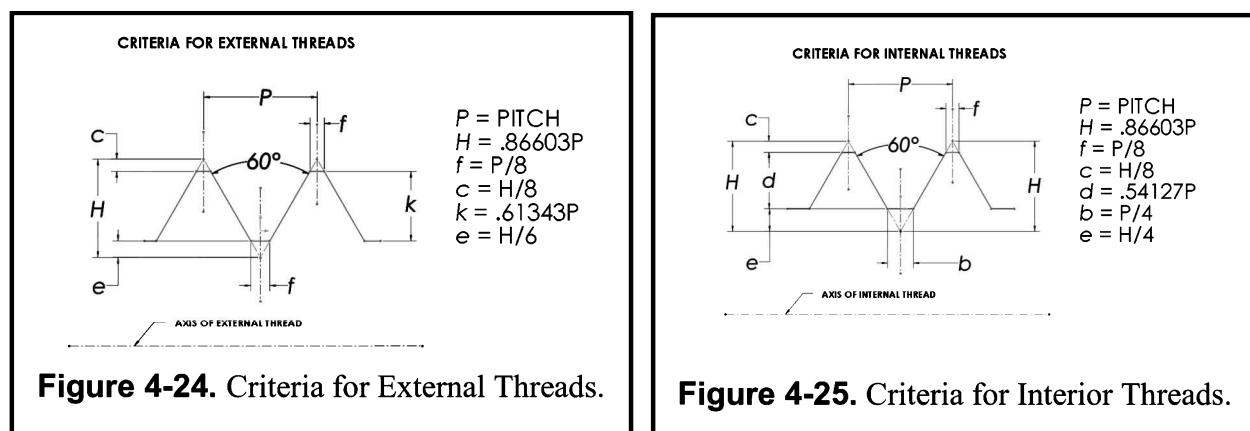
- Major Diameter – the largest diameter of an internal or external thread.
- Pitch – The distance from one crest of a thread to the next crest.
- Pitch Diameter – the theoretical diameter at the point where the tooth width and gap width are equal.
- Minor Diameter - the smallest diameter of an internal or external thread.
- Threads per Inch – The number of threads in an inch that determines the pitch.
- Thread Depth – generally the difference between the major diameter and minor diameter divided by 2.

THREAD NOTES

Thread notes are the most critical portion of a thread representation. It gives all the information necessary for a machinist to produce the thread or the manufacturer to select the correct fasteners for assembly. In **Figure 4-22** an American thread note is illustrated and in **Figure 4-23** is a metric thread note.

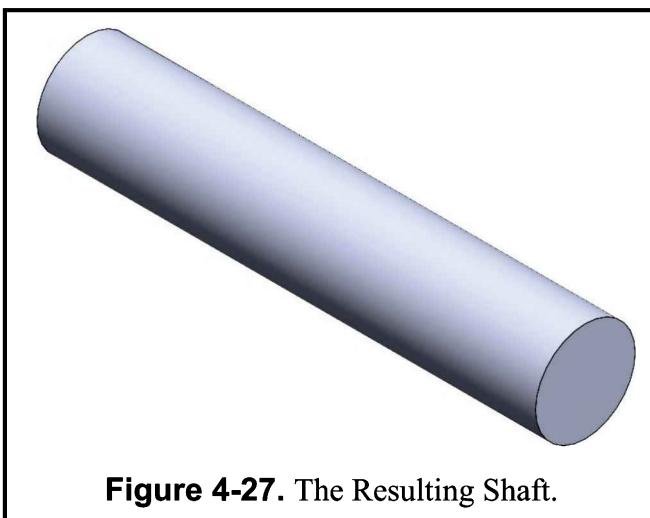


The dimensions of threads are based primarily on the number of threads per inch in the American system. The number of threads divided into one inch produces the pitch, which is also the basis in the metric system. In the exercise we will be using the criteria for internal and external threads. **Figure 4-24** shows the criteria required for external threads and **Figure 4-25** gives the specifications for the internal thread. The values for the different formulas are given in the various tables provided in Machinist or Engineering Handbooks. **Figure 4-26** is an example of such a table and includes the criteria that will be used in **Exercise 4.3**.



UNIFIED COARSE THREAD SERIES PARTIAL TABLE					
Fractional Sizes	Basic Major Diameter	Threads per Inch	Pitch	Minor Diameter Internal Threads	Thread Depth
1/4	0.250	20	0.0500000	0.1959	0.0325
5/16	0.313	18	0.0555555	0.2524	0.0361
3/8	0.375	16	0.0625000	0.3073	0.0406
7/16	0.438	14	0.0714286	0.3602	0.0464
1/2	0.500	13	0.0769231	0.4167	0.5000
9/16	0.563	12	0.0833333	0.4723	0.0541
5/8	0.625	11	0.0909091	0.5266	0.0590
3/4	0.750	10	0.1000000	0.6417	0.0650
7/8	0.875	9	0.1111111	0.7547	0.0722
1	1.000	8	0.1250000	0.8647	0.0767
1 - 1/8	1.125	7	0.1428571	0.9704	0.0928
1 - 1/4	1.250	7	0.1428571	1.0954	0.0928
1 - 1/2	1.500	6	0.1666667	1.3196	0.1083

Figure 4-26. Unified Coarse Thread Table for Selected Sizes.



From the Features Tab, select the **Extruded Cut** command, use the **Blind** end condition and make a cut **3.25"** deep. Activate the **Flip Side to Cut** option to make the cut outside the profile, as shown in **Figure 4-28**.

Now you need to add a **Chamfer** on the face of the threaded end. First you need to calculate the chamfer size using the formula for **Thread Depth** found in **Figure 4-24**. Per the table, a **1.00"** diameter thread has a **0.125"** pitch.

$$\text{Chamfer size} = (0.61343 * 0.125) \text{ or } 0.0767"$$

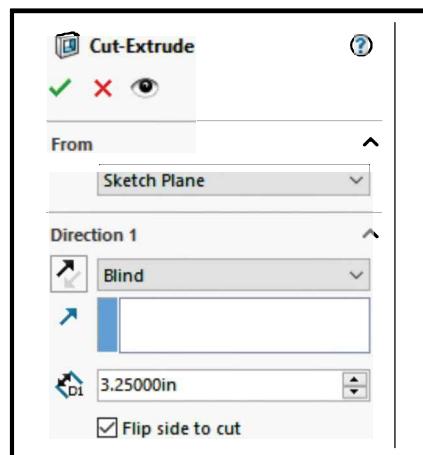
Start a new part with the **ANSI-INCHES** template, and save it as **Threaded Bolt.sldprt**.

Designing a thread requires high precision. Select the menu **Tools**, **Options**, **Document Properties**, **Units** and change the units to **5 Decimal Places**.

Add a new sketch in the Right plane, draw a circle at the origin, dimension its diameter **1.00"** and finally extrude it **5.00"** to the right. The result will look like **Figure 4-27**.

For proper clearances between interior and exterior threads, the exterior thread for a **1"** thread is undersized by a tolerance of between **0.9966"** and **0.9744"**. In our case we will use the mid-value of **0.9855"**.

Change to a **Left** view orientation, select the face on the left side of the shaft, add a new sketch and draw a circle with a **0.9855"** diameter.



From the Features tab select the **Chamfer** command, select the left side face of the threaded end, select the face and enter the **0.0767"** chamfer size. Click **OK** to finish.

The thread in this part will be **3.125"** long from the left side. To make the thread, it needs to start away from the left face the distance of the thread's **Pitch**. Per the table **Figure 4-26** it should be **0.125"**. In this case you need to add a Reference Plane at this location.

From the Features tab, click on **Reference Geometry** and select **Plane**. Select the left face of the threaded end, change the distance **0.125"**, make sure the plane is located on the left and click **OK** to finish. Your part should look like **Figure 4-27**.

To make the thread you will need to make a **Sweep** feature, which requires a **Path** or curve, and a **profile**. The path will be a helix that will define the thread, and a profile with the thread's profile.

To make the helix, first you need to make a sketch with a circle. Select **Plane1** in the FeatureManager and add a new sketch. Select the larger diameter in the chamfer and click in the **Convert Entities** command.

After the circle is projected, select the Features tab, click on the **Curves** command, and select **Helix/Spiral**, or go to the menu **Insert, Curves, Helix/Spiral**. In the **Helix/Spiral** options use the **Height and Pitch** definition and select the **Constant pitch** option. Set the **Height** to **3.25"** and the Pitch to **0.125"**.

Activate the **Reverse Direction** checkbox to make the helix go into the part, set the **Start angle** to **270°** and make sure you are using the **Clockwise** option to create a right-hand thread as shown in **Figure 4-28**. After entering all the parameters click **OK** to create the Helix.

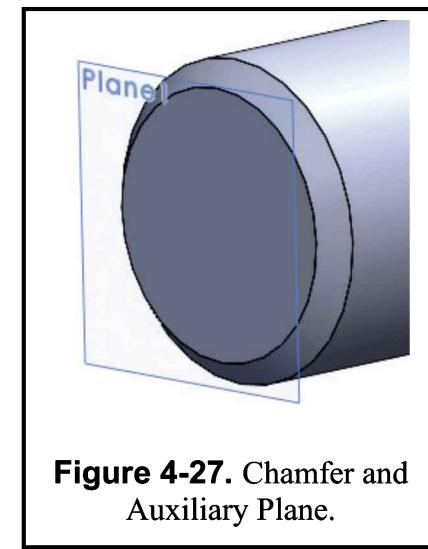


Figure 4-27. Chamfer and Auxiliary Plane.

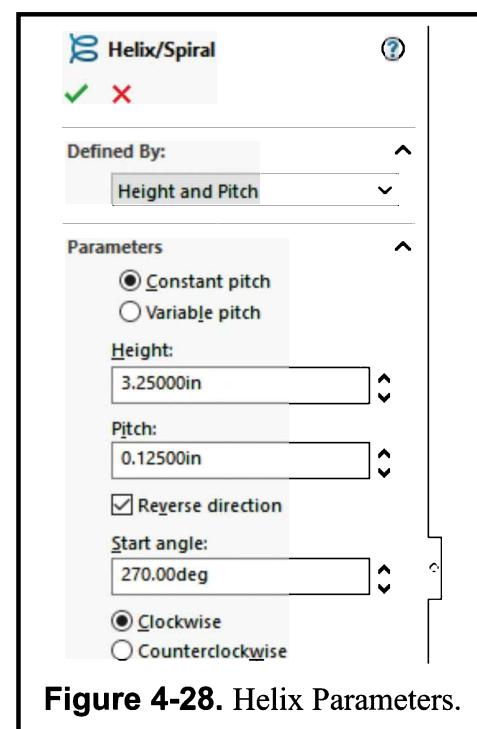


Figure 4-28. Helix Parameters.

The reason to start the helix at **270°** is to make the start of the curve coincide with the **Front** plane; this way you can add the thread's profile in the **Front** plane.

Change to a **Front** view, select the **Front** plane, and add a new sketch. **Zoom** in the left side of the helix, add a horizontal centerline coincident to the bottom of the threaded end, and a vertical centerline coincident to **Plane1** (seen in **Figure 4-29** from the side). Select the vertical centerline and go to the menu **Tools, Sketch Tools, Dynamic Mirror** to make the profile symmetrical. Draw and dimension the sketch as shown in **Figure 4-30**.

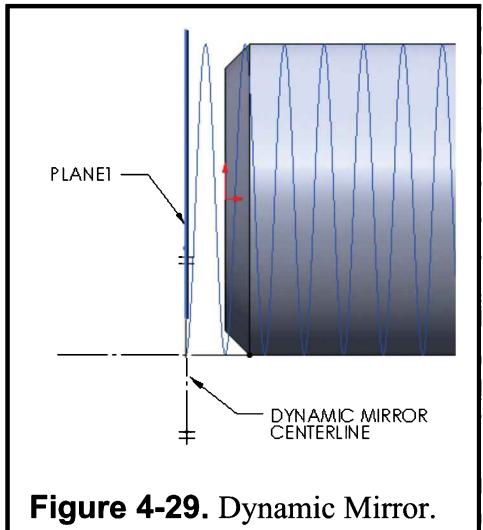


Figure 4-29. Dynamic Mirror.

With the **Dynamic Mirror** active, draw one half of the profile, and the other half will be automatically created. To dimension the sketch, you must calculate the size of the thread profile using the formulas in **Figure 4-24**, since this is an external thread. Remember the Pitch (P) is **0.125"**.

$$f = P/8 \rightarrow 0.125"/8 = \mathbf{0.0156"}$$

$$k = 0.61343 * P \rightarrow 0.61343 * 0.125" = \mathbf{0.07670"}$$

The finished sketch is shown in **Figure 4-30**. Make sure the lower horizontal line is located slightly below the bottom of the threaded end. The reason is to make sure the **Sweep Cut** feature extends past the solid body.

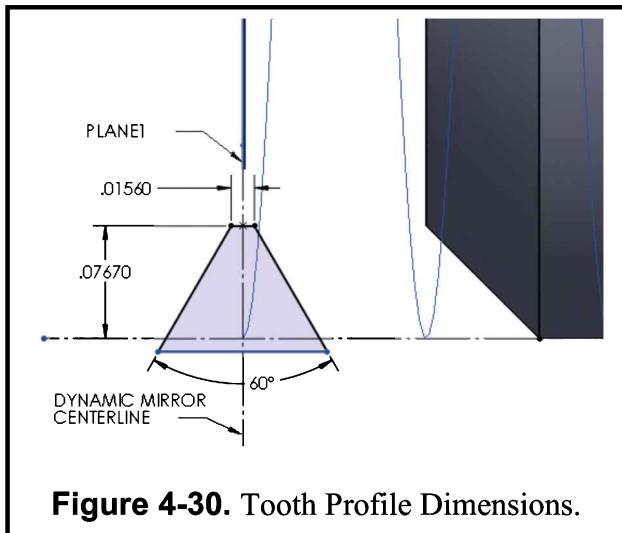


Figure 4-30. Tooth Profile Dimensions.



After the sketch is complete click on **Exit Sketch** or the **Rebuild** command. The Helix and thread profile are shown in **Figure 4-31**.



From the Features tab select the **Swept Cut** command and change to an **Isometric** view for visibility. In the **Profile** selection box add the thread profile sketch, and in the **Path** selection box add the Helix. Review the preview and click **OK** to finish. To hide the helix curve, select it either in the FeatureManager or the graphics area, and click on **Hide** from the context toolbar. The finished thread is shown in **Figure 4-32**.

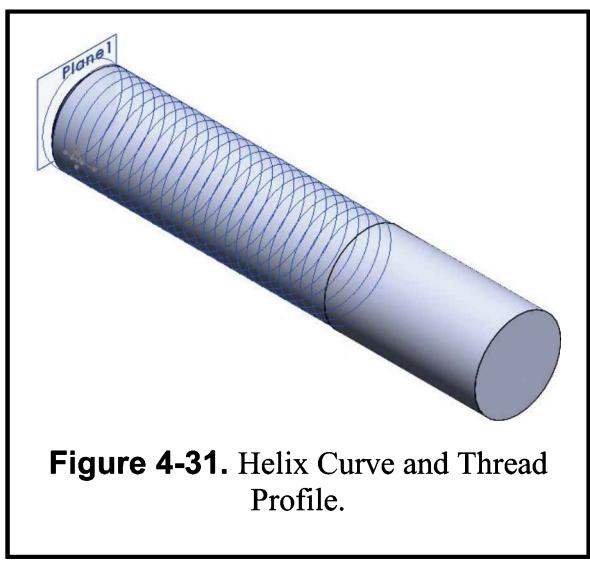


Figure 4-31. Helix Curve and Thread Profile.

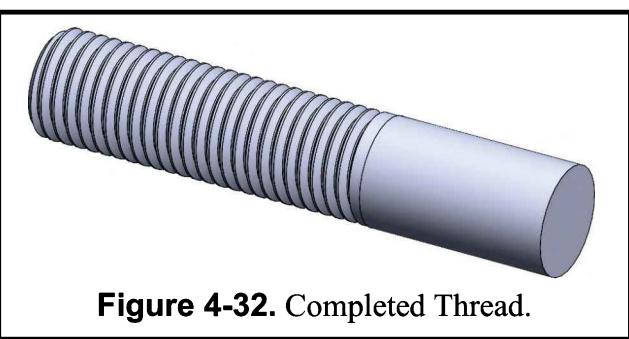
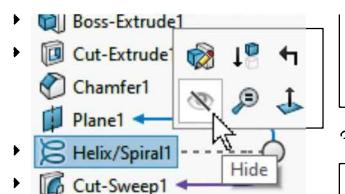


Figure 4-32. Completed Thread.

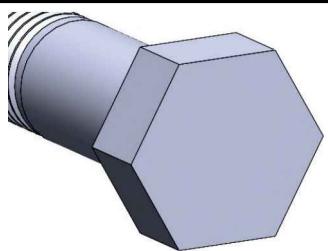


Figure 4-33. Bolt Head.

Now you need to make the head of the bolt. Change to a **Right** view orientation, add a sketch in the right face and draw a **Hexagon** at the origin. Select one of the hexagon's lines, add a **Horizontal** geometric relation and dimension the distance between the top and bottom sides **1.50"**. The standard height for a **1"** screw head is **39/64"**. With the fully defined sketch make an **Extruded Boss**, in the extrusion's distance box enter **39/64"**. SOLIDWORKS will calculate the fraction and change it to **0.609375"**. Click **OK** to finish as shown in **Figure 4-33**.

 **Tangent** To complete the bolt's head, add a new sketch on the right face of the bolt, draw a circle and add a **Tangent** relation between the circle and the edge. Click on **Exit Sketch** or **Rebuild** to continue. Your sketch will look as **Figure 4-34**.

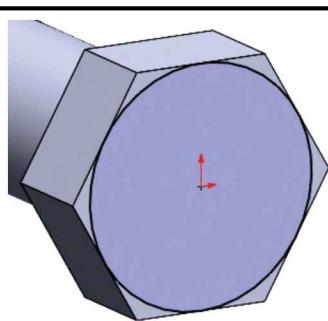


Figure 4-34. Circle Sketch.

Now change to a **Top** view orientation, select the **Top Plane**, and add a new sketch. Draw the profile approximately as shown in **Figure 4-35**, including a horizontal centerline going through the bolt's axis.

To finish the sketch, slightly rotate the part, select the  previous sketched **circle** and the bottom **endpoint** of the triangle, and add a **Pierce** relation as shown in **Figure 4-36**. From the Features tab select the **Revolved Cut** command, the centerline is automatically selected and click **OK** to finish. The finished bolt is shown in **Figure 4-37**. Save your part as **Threaded Bolt.sldprt**.

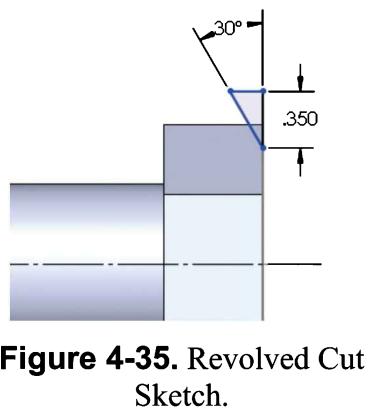


Figure 4-35. Revolved Cut Sketch.

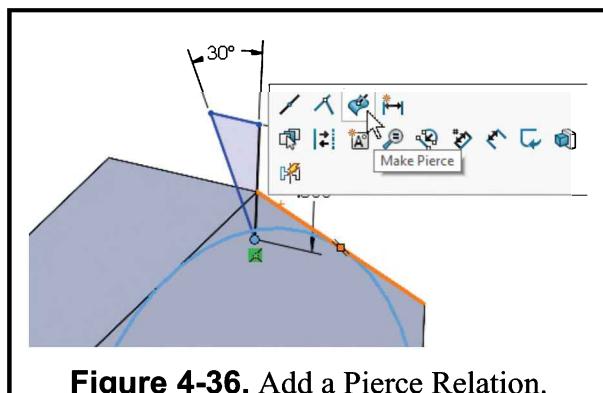


Figure 4-36. Add a Pierce Relation.

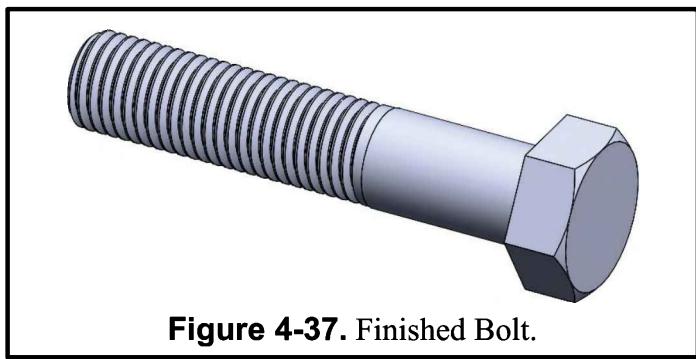


Figure 4-37. Finished Bolt.

CONSTRUCTION OF A CORRESPONDING NUT

There are many standards to produce threads and fasteners. When you design fasteners, it is advisable to consult official standards for accuracy. In the case of a 1" diameter nut to fit the previous bolt, the hexagon of the nut is 1.5 times the major diameter of the shaft. Make a new part using the **ANSI-INCHES** template and change the units to **5** decimal places.

Select the **Right Plane**, add a new sketch and draw a hexagon. Select one side, add a horizontal relation and dimension the distance between sides **1.500"**. Referring to **Figure 4-26** on **Page 4-12**, draw a circle at the origin; its diameter will be the Minor Diameter for Internal Threads, equal to **0.8647"**. To complete the first feature, make an **Extruded Boss** using the **Midplane** end condition, and using the dimension from the standards table set the depth to **55/64"**, which SOLIDWORKS resolves to

0.859375", as shown in **Figure 4-38**.

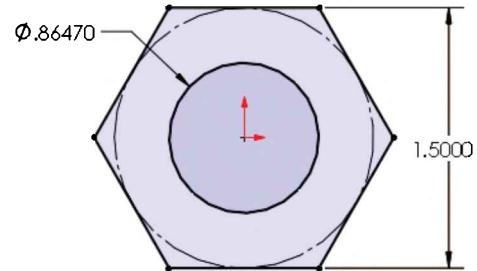


Figure 4-38. Initial Nut Profile.

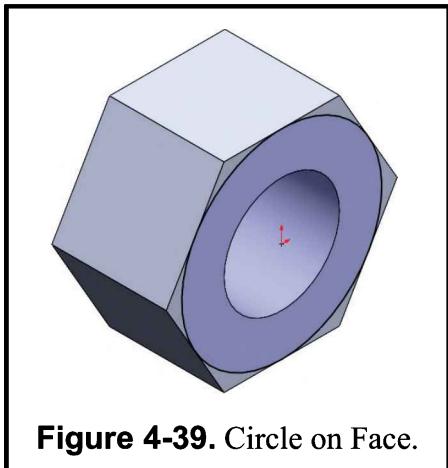


Figure 4-39. Circle on Face.

Select the right face of the base feature, add a sketch, and draw a circle at the origin. Just like with the bolt's head, make the circle **Tangent** to an edge, and **Rebuild** the model, or **Exit the Sketch**, as shown in **Figure 4-39**.

As before, change to a **Top** view orientation, select the **Top Plane**, and add a new sketch. Draw and dimension the sketch profile approximately as shown in **Figure 4-40**, including the centerline at the origin.

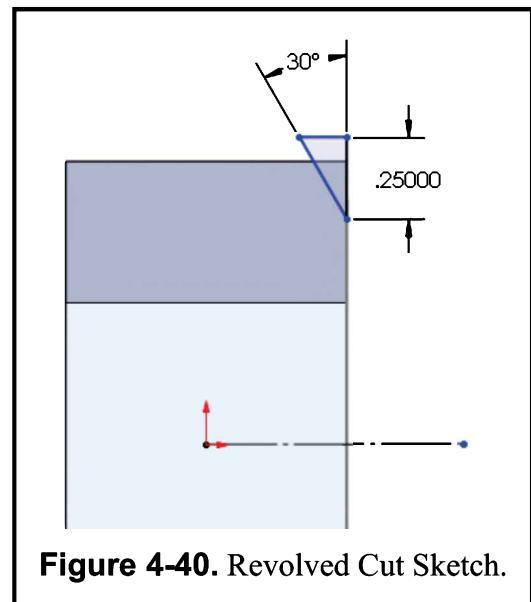
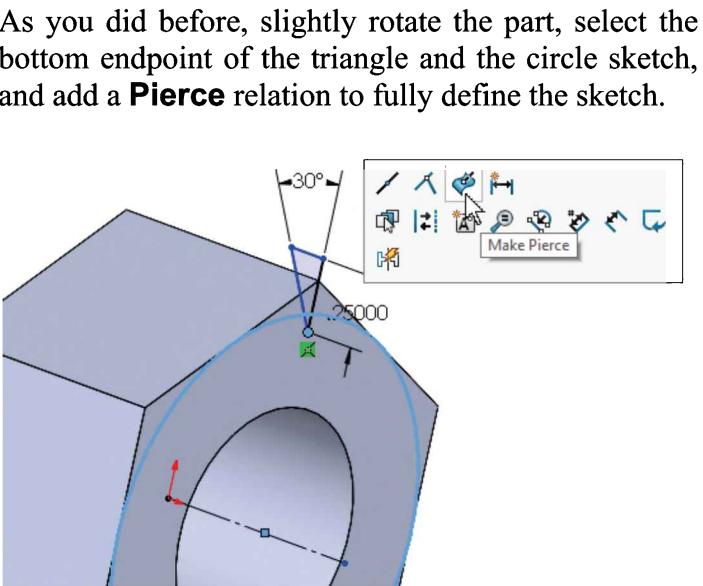


Figure 4-40. Revolved Cut Sketch.



Select the **Revolved Cut** command and make a **360°** cut around the centerline. To duplicate the cut on the other side, select the **Mirror** command from the Features tab and mirror the **Cut-Revolve1** about the **Right Plane**, as seen in **Figure 4-41**.

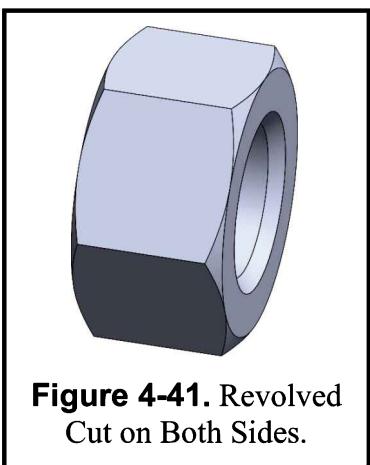


Figure 4-41. Revolved Cut on Both Sides.

The next step is to add a **Chamfer** in both ends of the hole equal to the Thread Depth. According to the chart in **Figure 4-26 (Page 4-12)** for a **1"** diameter thread it must be **0.0767"**. Select the **Chamfer** command, set the size, and to add both edges at the same time, select the inside face of the hole. Click **OK** to finish.

Now, as you did before, add a reference plane 1 Pitch distance (**0.125"**) away from the nut. Select the **Reference Geometry** and click on **Plane**. Select the right face of the nut, set the distance to **0.125"** and click **OK**.

To make the thread's **Helix**, add a new sketch on **Plane1**, select the smaller edge inside the nut and click on **Convert Entities** to project it. Activate the Features tab, select the **Curves** command and click on **Helix and Spiral**.

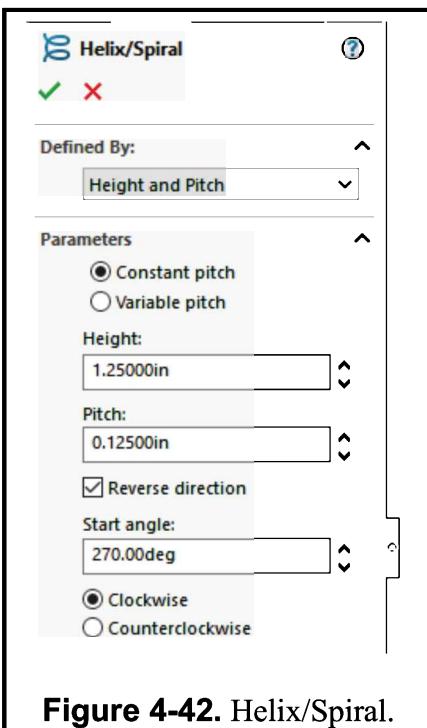


Figure 4-42. Helix/Spiral.

Just as you did with the bolt previously, in the **Helix/Spiral** options use the **Height and Pitch** definition and select the **Constant pitch** option. Set the **Height** to **1.25"** and the **Pitch** to **0.125"**. Activate the **Reverse Direction** checkbox to make the helix go into the part, set the **Start angle** to **270°** to make the start of the helix at the **Front Plane** and make sure you are using the **Clockwise** option as shown in **Figure 4-42**. Click **OK** to create the Helix.

The next step is to draw the thread's profile sketch. Change to a **Front** view and set the Display Style to **Hidden Lines Visible**. Add a new sketch in the **Front Plane** and draw the sketch shown in **Figure 4-43**.

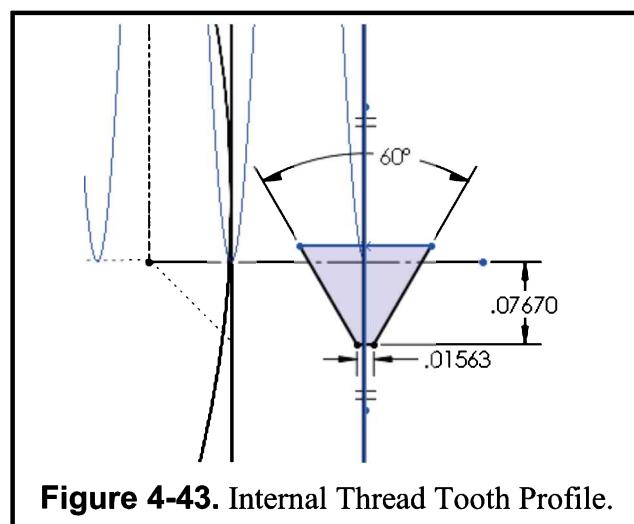
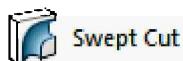


Figure 4-43. Internal Thread Tooth Profile.

Remember to add a horizontal and vertical centerline. In order to make the sketch symmetrical, activate the **Dynamic Mirror** command about the vertical centerline, which is coincident to the **Plane1**. The sketch dimensions were calculated from the formulas provided in **Figure 4-24**.

When the sketch is complete click on **Exit Sketch** or **Rebuild** the part to continue.



Change back to a **Shaded with Edges** display form for visibility, and select the **Swept Cut** command from the Features tab. In the **Profile** select the thread profile sketch, and in the **Path** select the helix curve. Click **OK** to finish and hide the helix and the auxiliary plane. Save your part as **Threaded Nut.sldprt**. Your finished part should look like **Figure 4-44**.

To finish this exercise, make a new drawing using the template in Inches. Add an **Isometric** view of the **Threaded Bolt** and the **Threaded Nut** parts as shown in **Figure 4-45**.

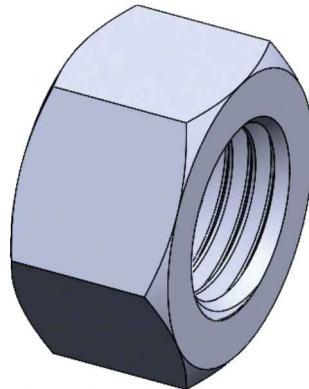


Figure 4-44. Finished Threaded Nut.

Save your drawing as **SCREW THREADS.slddrw** and print a copy to submit to your lab instructor.

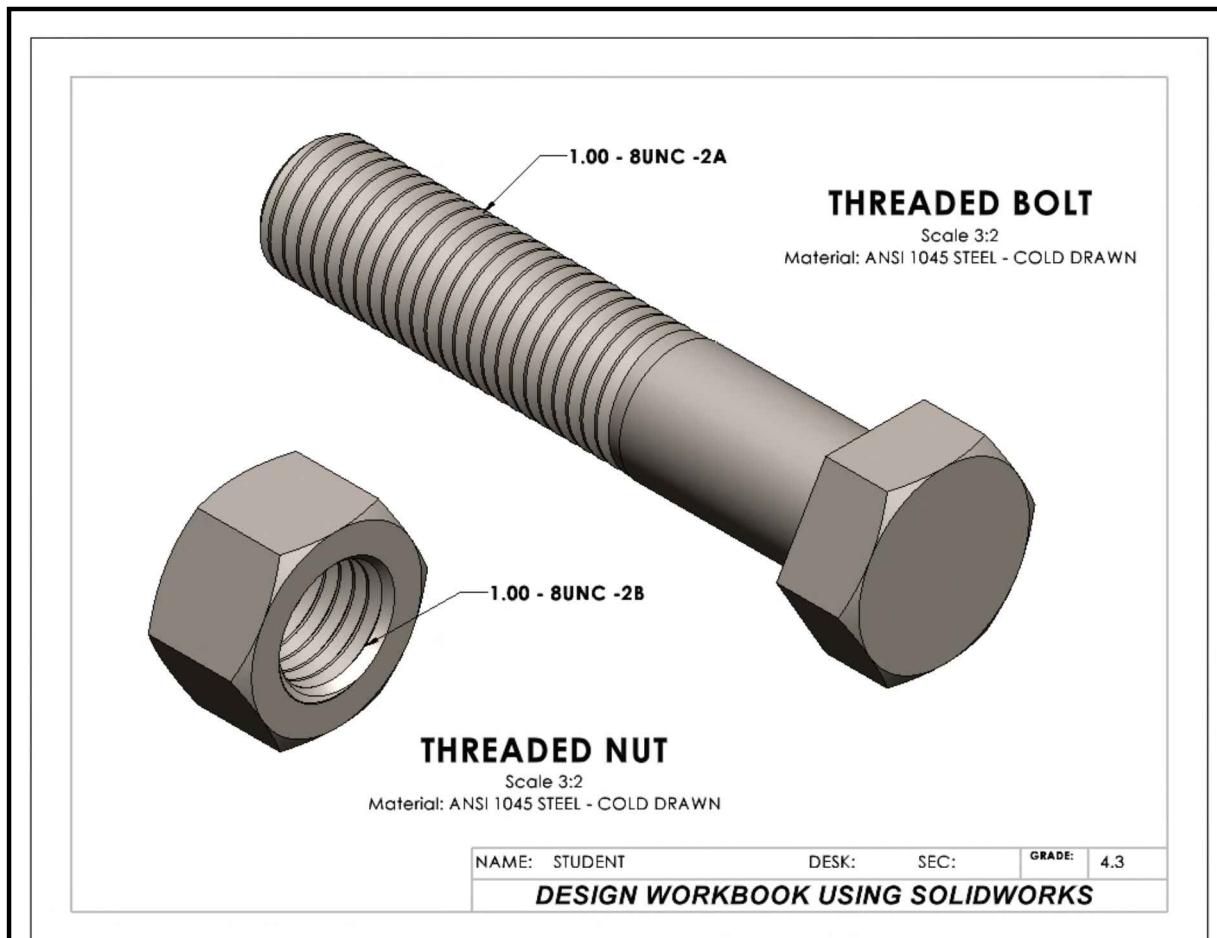


Figure 4-45. Completed Threaded Shaft and Nut.

Exercise 4.4: JACK STAND

In this Exercise 4 you will learn the basic options to create a **Lofted Boss** feature. A loft is a feature that changes shape between multiple different cross sections. The sections can be model faces or sketch profiles and optionally can have guide curves to better control the resulting feature. To learn this feature you will create a Jack Stand.

Make a new part using the **ANSI-INCHES** template and save it as **Jack Stand.sldprt**. Select the **Top Plane** and add a new sketch. Using the **Center Rectangle** command and the **Sketch Fillet**, draw the  **Center Rectangle** profile shown in **Figure 4-46**.



When you finish the sketch, click on **Exit Sketch** or **Rebuild** the model to continue. 

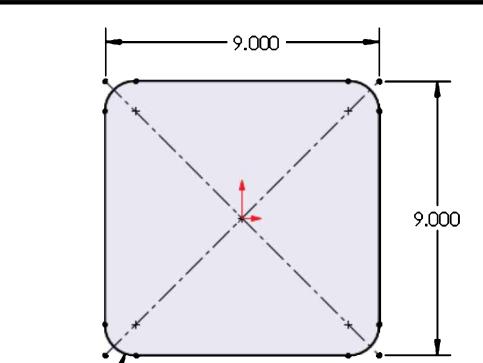


Figure 4-46. Center Rectangle Sketch.

To create the second section of the **Loft** feature you need to make a new **Auxiliary Plane**. Change to an **Isometric** view for better visibility, click on **Reference Geometry** command and select **Plane**.

From the fly-out FeatureManager select the **Top Plane**, set the offset distance to **6.50"** above the existing sketch and click **OK** to create it.

Select the newly made **Plane1**, change to a **Top** view orientation and add a new sketch. Draw a circle centered on the origin, dimension it **4.0"** diameter and click on **Exit Sketch** or **Rebuild**.

For the third section of the **Loft** add a new **Auxiliary Plane**. Change to an **Isometric** view, click on **Reference Geometry** and select **Plane** again. Select the **Top Plane** for reference, set the offset distance to **10.0"** above the **Top Plane** and click **OK** to finish.

Add a sketch on **Plane2** and change to a **Top** view for visibility. Draw a circle centered in the origin, dimension it **3.0"** in diameter and **Exit** the sketch. Change to an **Isometric** view to see the resulting sketch profiles as shown in **Figure 4-47**.



Lofted Boss/Base

With the profiles completed, activate the Features tab, and click on the **Lofted Boss** command.

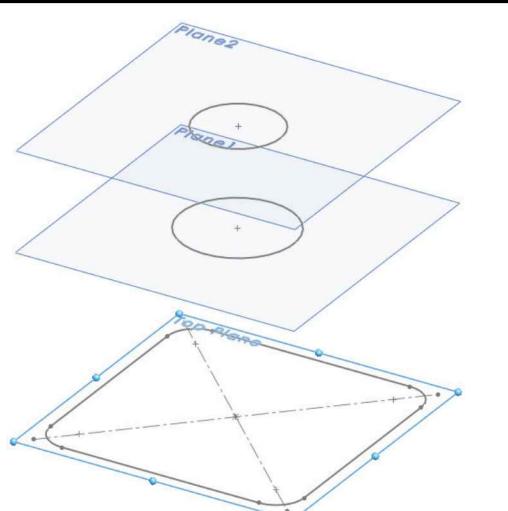


Figure 4-47. Sketches for Loft.

To better visualize the **Loft** operation change to an **Isometric** view. In the **Profiles** section select the three profiles made starting from the bottom and going up, as shown in **Figure 4-48**.

When you select the profiles, remember the location where you select a profile will be connected to the location where you select the next profile. For example, if you select the profiles on opposite sides, this will cause the **Loft** feature to twist.

As you select the sketches, a preview of the loft will be shown on the screen. If it looks similar to **Figure 4-49**, click **OK** to complete the loft.



To make the jack stand lighter select the **Shell** command from the Features tab.

Set the wall thickness to **0.1875"** and rotate the part to select the bottom face of the part to remove it. Click **OK** to finish.

The next operation is to reinforce the top face and add a hole for a threaded adjustment. Change to a **Top** view, select the top face, and add a new sketch.

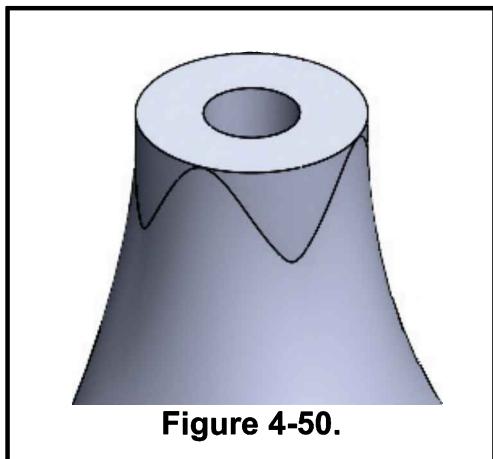


Figure 4-50.

Use **Convert Entities**

to project the edges of the top face and make a **Boss Extrude** going down **3.0"**. Select the top face again, add a new sketch and draw a circle of **1.25"** diameter. Use the **Extruded Cut** command with the **Through All** end condition and click **OK**. The top of the part should look like **Figure 4-50**.

The last feature to add is a triangular cut on the sides of the jack stand to make it lighter. Change to a **Front** view and add a new sketch on the **Front Plane**.

To make the profile symmetric, first draw a triangle and then add a vertical centerline from the origin to the bottom line. Add a **Midpoint** relation between the top endpoint of the centerline and the horizontal line, making the bottom line symmetric. Now add a **Coincident** relation between the top endpoint of the triangle and either the origin, or the vertical centerline.

Add the dimensions indicated in **Figure 4-51**. Select the **Extruded Cut** command, use the **Through All-Both Directions** end condition and click **OK** to finish.

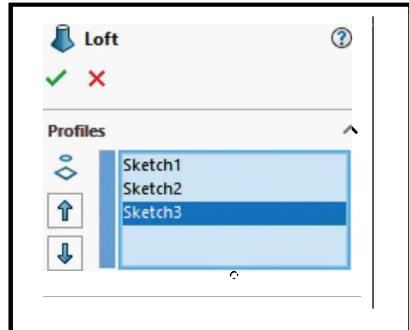


Figure 4-48. Loft Sections.

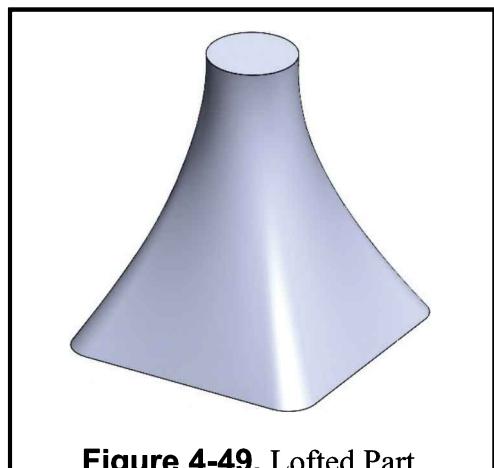


Figure 4-49. Lofted Part, Auxiliary Planes Hidden.

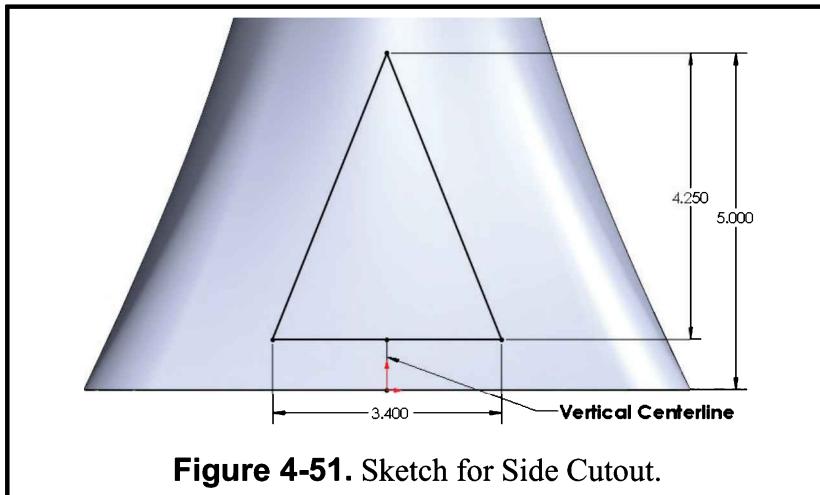
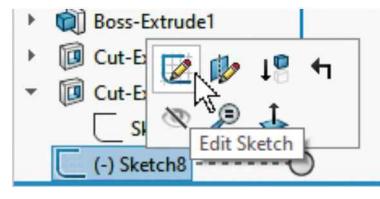


Figure 4-51. Sketch for Side Cutout.

Now you need to add the same cut extrude feature perpendicular to the **Right Plane**.

To reuse the same sketch you just made, expand the last **Cut-Extrude** in the FeatureManager, select the Sketch and from the menu click **Edit, Copy** (default shortcut **Ctrl+C**).



Now select the **Right Plane** in the FeatureManager, change to a **Right** view orientation and click in the menu **Edit, Paste** (default shortcut **Ctrl+V**). Select the new sketch in the FeatureManager or the graphics area, and from the context toolbar select **Edit Sketch**.

The reason to edit the sketch is because some information was lost in the **Copy-Paste** process. Even if the sketch *looks* correct, the coincident relation between the bottom endpoint of the centerline and the origin was lost, as well as the distance between the bottom of the part and the top of the sketch, making the sketch **Under defined** and colored *blue*, which is *not* the desired sketch state.

To fix it **click-and-drag** the bottom endpoint of the centerline *away* from the origin and move it back onto the origin. By doing this, you will be adding a **Coincident** relation again. Add the missing **5.0"** dimension from the top of the triangle to the bottom of the part. Now that the sketch is Fully Defined, make the **Extruded Cut** using the end condition **Through All-Both Directions** to finish. Your finished model should look similar to **Figure 4-52**.

To complete the design, change the material to **ANSI-1020 Steel-Cold Rolled** and save your model as **JACK STAND.sldprt**.

To finish this exercise, make a new drawing, add an **Isometric** view and the missing annotations. Save your drawing as **JACK STAND.slddrw** and print a copy for your lab instructor.

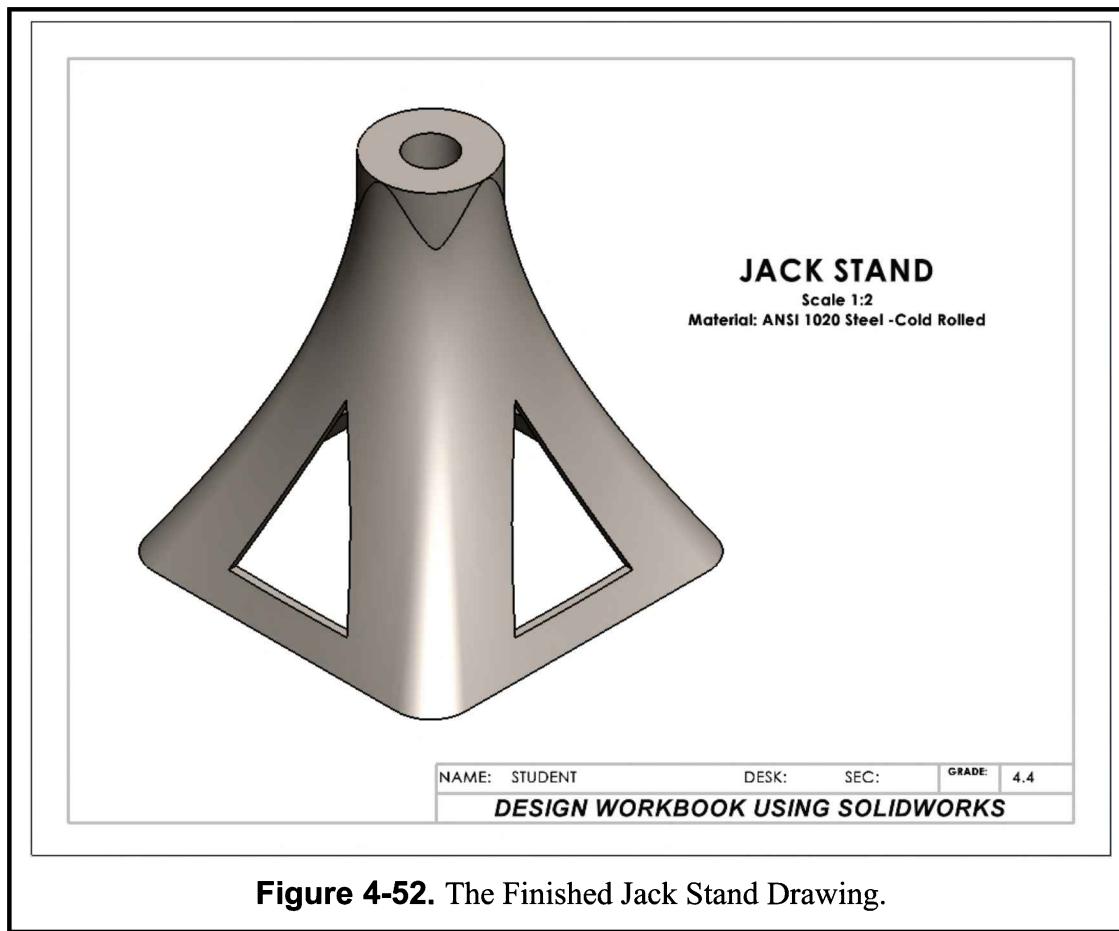
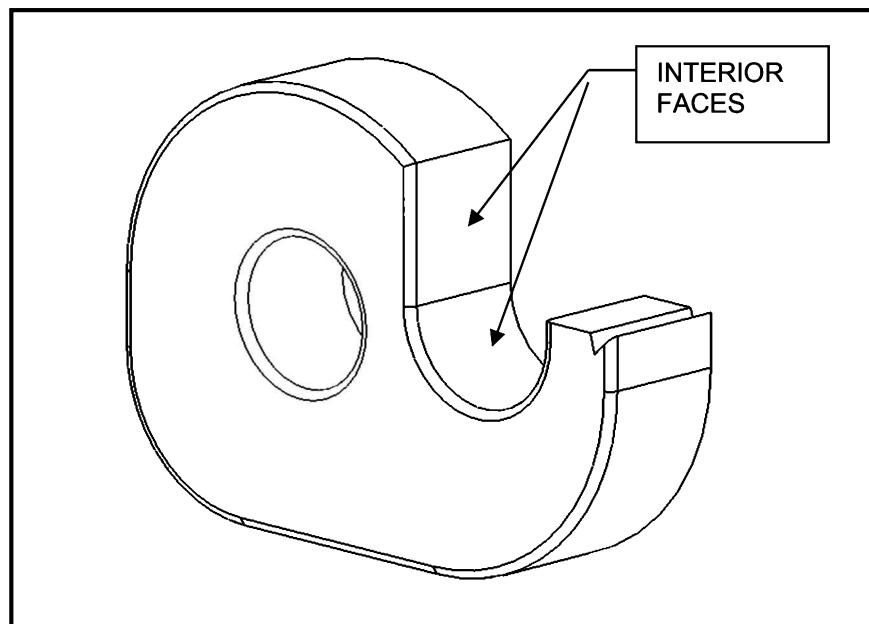
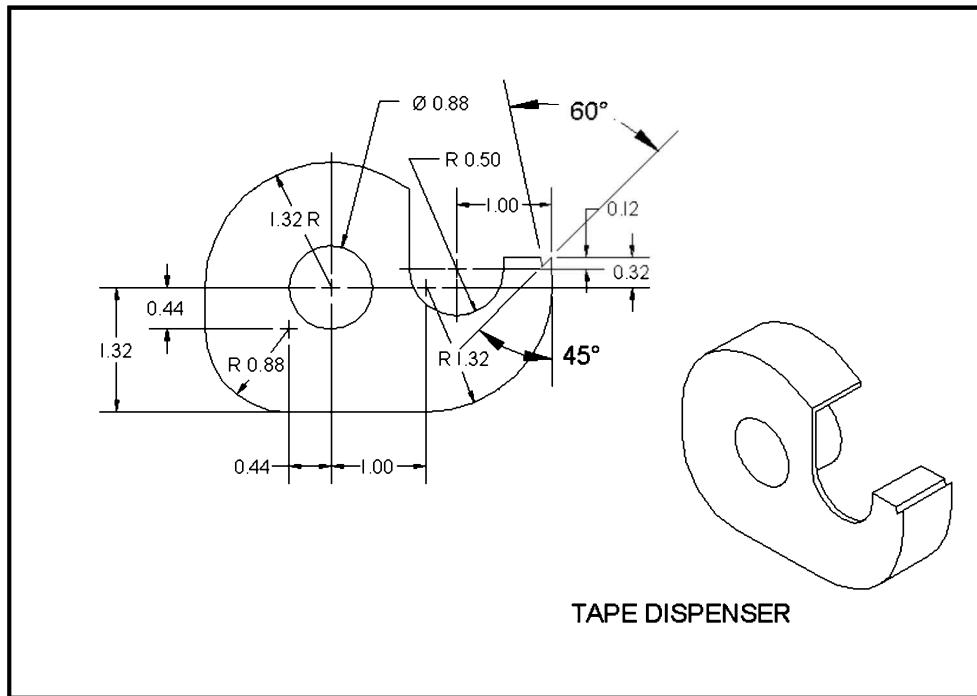


Figure 4-52. The Finished Jack Stand Drawing.

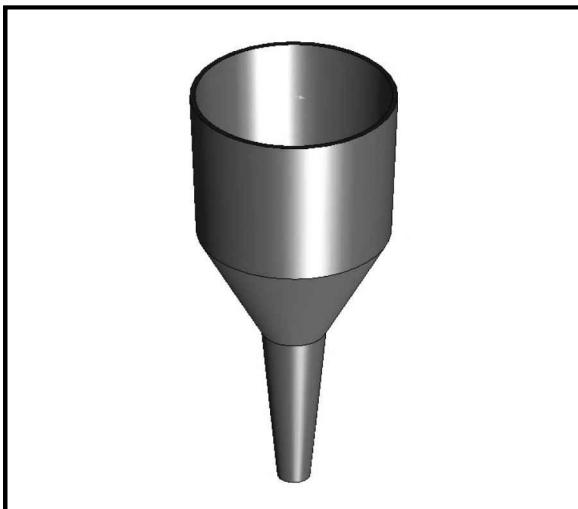
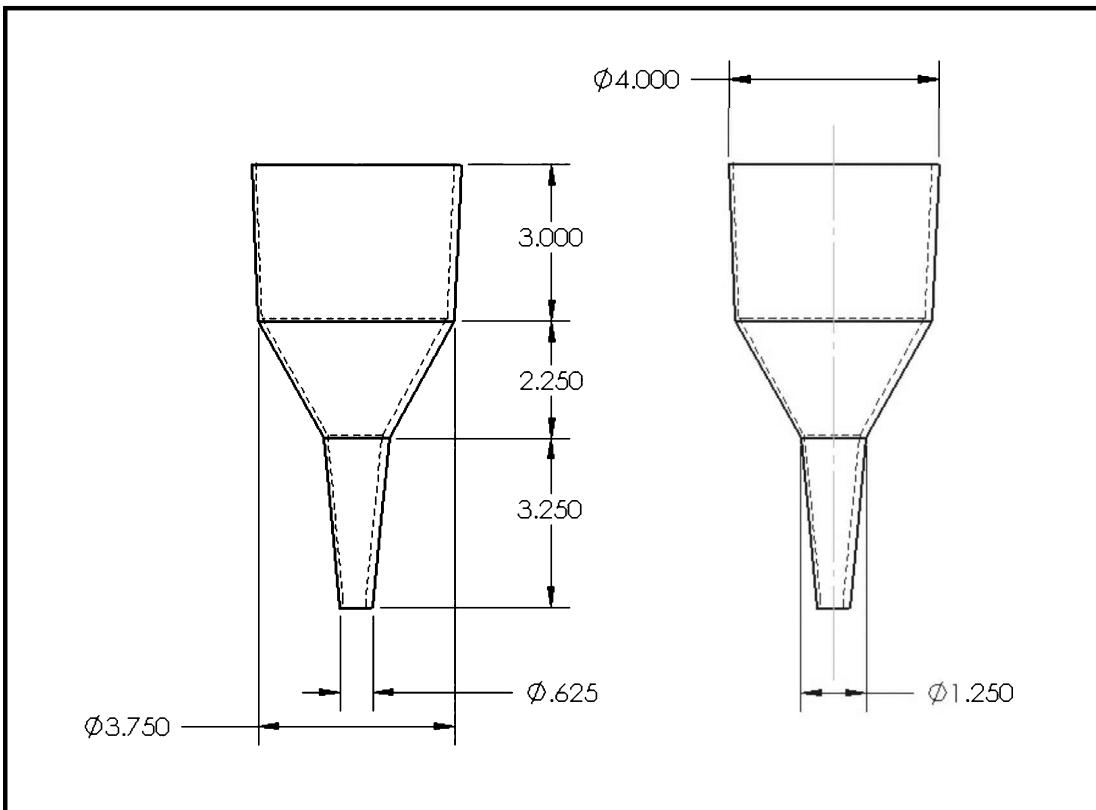
Supplementary Exercise 4-5: TAPE DISPENSER

Using the **ANSI-INCHES** part template draw the profile of the Tape Dispenser in the **Front Plane** and extrude it **0.75"**. Select the front face and add a fillet of **0.05"**. Select the back face and the two interior faces as indicated in the second image and Shell the part with a thickness of **0.10"**. Make a new drawing with a Shaded with Edges **Isometric** view and save it as **Tape Dispenser.slddrw**.



Supplementary Exercise 4-6: FUNNEL

Build a model of the Funnel below. Begin in the **Top Plane**. To get the desired model, each section must be lofted separately. Create auxiliary planes at the indicated distances and add the profiles needed for each Loft feature. When the loft features are complete, select the top and bottom faces and shell the model to a wall thickness of **0.10"**. Make a drawing with an **Isometric** view and save it as **FUNNEL.slddrw**.

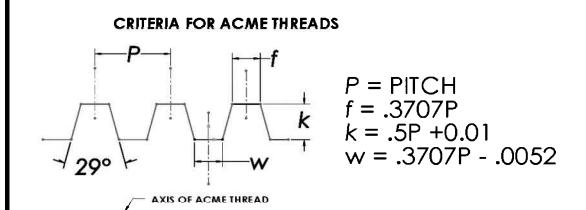


Supplementary Exercise 4-7: TWO INCH THREAD - UNC

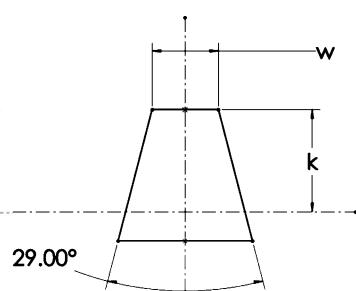
1. Make a 2" diameter shaft 5 inches long.
2. The major screw thread diameter is from 1.9868 – 1.9424. Cut the shaft down to 1.9646 for three inches.
3. Screw thread note - 2-4 ACME-2A X 3.
4. Add a chamfer equal to the Thread Depth (the "k" value in the Definition of Terms) on the end of the shaft that will be threaded.
5. Add a reference plane one pitch "P" distance away from the end of the shaft.
6. Select the reference plane and project the 1.9646 diameter circle using Convert Entities.
7. Make a Helix using the following parameters – height and pitch. The height will be 3.25" and the pitch will be 0.25", with the starting angle being at 270 degrees – clockwise.
8. Select the Front Plane and draw a centerline along the lower edge of the shaft and a vertical centerline coincident with the auxiliary plane. Select the vertical Centerline and use the Dynamic Mirror function.
9. Sketch the thread's profile according to the given parameters.
10. Use the Features – Swept Cut function to cut the thread.

ACME THREAD SPECIFICATIONS					
Nominal Size	Basic Major Diameter	Threads per Inch	Pitch	Thread Depth	Width of Space at Bottom of Thread
2	1.9646	4	0.25	0.135	0.087475

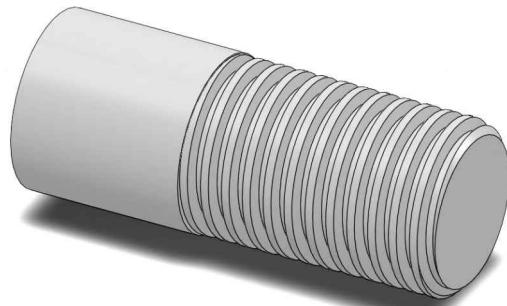
Values from Thread Table



Definition of Terms



Thread Profile for the Cut Sweep



Finished Threaded Shaft

NOTES:

Design Workbook Lab 5: Assembly Modeling

In this exercise you will learn how to make an assembly from multiple components, including how to use the basic tools available, mate components, add Exploded Views, and make assembly drawings with Bill of Materials and identification balloons.

ASSEMBLIES

So far you have been making individual parts; however, your designs will likely be made of multiple assembled components. To make a new assembly, just like a part or drawing, you must select an assembly template, add components to it and join them together using mates to reference how one component's geometry is connected to another. Just like parts and drawings, you can make multiple assembly templates with different options like units, precision, dimensioning standard, etc., following the same procedure used to create **Part Templates** in **Page 1-5**, except you will save them with the Assembly Template type (*.asmdot) to your templates folder.

To start a new assembly, select the **New** command, or use the menu **File**, **New** and select the “Templates” tab if you are using the **Advanced** option, or the Assembly document type if you are using the **Novice** option, as shown in **Figure 5-1**.

In an assembly, every component has six **Degrees of Freedom (DOF)**: three **translations** about the **X, Y and Z axis**, and three **rotations** about the **X, Y and Z axis**, and each of them can be either free or constrained based on how a component is related (**mated**) to other components. Partially constraining a part's DOF is the basis for assembly motion in SOLIDWORKS. The first component added to an assembly has all six degrees of freedom automatically constrained, or **Fixed**.

Making a new assembly using the default **Assembly** template will automatically start the **Begin Assembly** (or **Insert Component**) command as shown in **Figure 5-2** and ask you to browse and select the first assembly component.

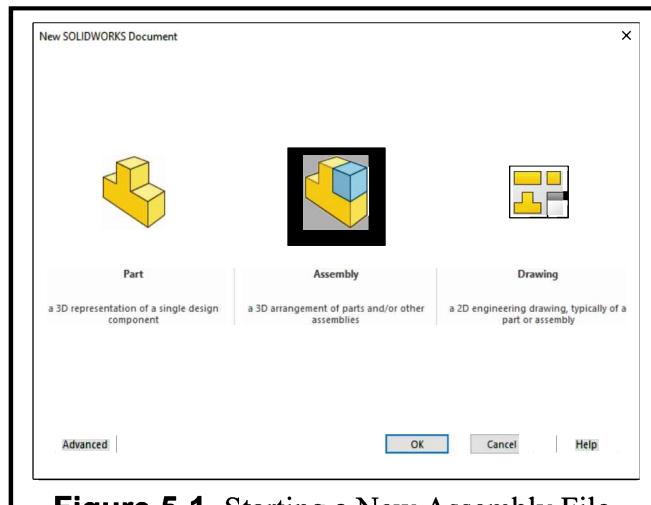


Figure 5-1. Starting a New Assembly File.

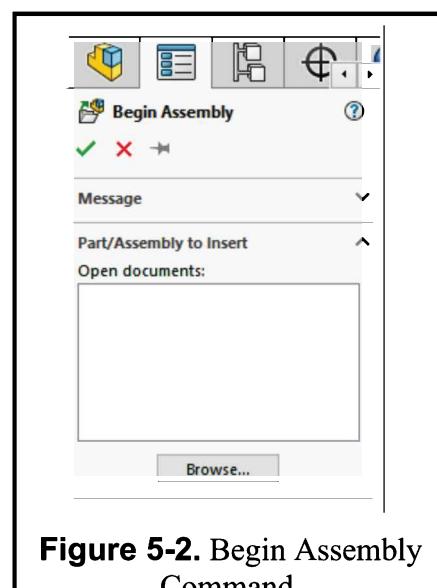


Figure 5-2. Begin Assembly Command.

After you have selected the first component, click **OK** to add the component to the assembly. By doing this, the *part's origin* will be coincident with the *Assembly's origin*, the part's **Front** plane will be coincident with the Assembly's **Front** plane, and likewise the **Top** and **Right** planes.

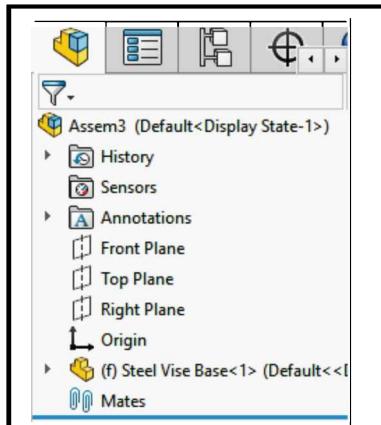


Figure 5-3. First Assembly Component.

It is usually good practice and convenient to align the first part's default Planes to the assembly's Planes for reference, especially when mating other components in the assembly.

After the first component is added, its name will have the prefix “(f)” in the FeatureManager, as seen in **Figure 5-3**, indicating to you that it is **Fixed** in place, meaning all six **Degrees of Freedom (DOF)** are constrained in space, and the component cannot be moved or rotated. This is the reason why it is a good idea for the first component to be the main part of the assembly, or the one other parts and sub-assemblies will be attached to. For example, if you are assembling a bicycle, the frame would be the best option as a first component because every other part will be attached to it and will move relative to it.

Assembly Commands

When you are working in the assembly environment, the CommandManager automatically changes to include assembly specific tools. Some of the most commonly used commands include:

	Edit Component – Used to modify a component or sub-assembly while working in the assembly. Useful to make changes to match other assembly components.
	Insert Components – Add an existing part or sub-assembly to the assembly.
	Mate – Position two components relative to each other using component faces, edges, planes, axes, vertices, sketch geometry, etc.
	Linear Component Pattern – Patterns components in one or two linear directions. Includes a drop-down menu with additional pattern types including Circular, Curve Driven, Mirror components, etc.
	Move Component – Moves a component within the degrees of freedom defined by its mates. Includes a drop-down menu to also rotate the component.
	Show Hidden Components – Temporarily hide all visible components and reveal hidden components. Select a hidden component to make it visible.

 Assembly Features	Assembly Features – Creates different assembly features, including cuts, simple holes, fillets, chamfers, hole series, weld beads, etc.
 Reference Geometry	Reference Geometry – Allows you, just like in components, to add reference geometry including Plane, Axis, Coordinate System, Point, and Mate Reference.
 Exploded View	Exploded View – Creates a view of the assembly components in an exploded state.

MATE TYPES

To make an assembly, you have to add multiple components (parts and/or sub-assemblies) and MATE them together. To add Mates, you can select component's faces, edges, vertices, sketch elements, planes, auxiliary geometry (planes, axes, etc.), origins, etc. Based on the different component entities selected, the valid mate references will be presented.

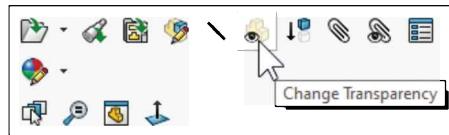
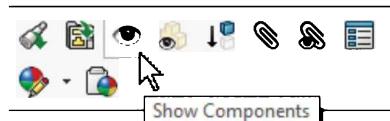
The most common Mates include:

Mate Type	Selections
 Coincident	Places the selected Faces, Edges, Planes, Vertices, etc. coincident and parallel. Two Coincident vertices will share the same point.
 Parallel	Places the selected flat Faces, linear Edges or Planes Parallel to each other, but not necessarily coincident.
 Perpendicular	Places the selected flat Faces, Linear Edges, or Planes at 90° to each other.
 Tangent	Places the selected entities Tangent to each other. At least one of the selected items must be a circular edge, cylindrical, conical, or spherical face.
 Concentric	Places the two selected entities so that they share the same axis.
 1.000in	Distance: Places the selected entities Parallel at a defined distance from each other.
 135.00deg	Angle: Places the selected entities at the specified angle from each other.
 Symmetric	Places the selected items at an equal distance from a plane of symmetry.
 Width	Places two selections of one component centered between two selections of a second component.

VIEWING ASSEMBLIES

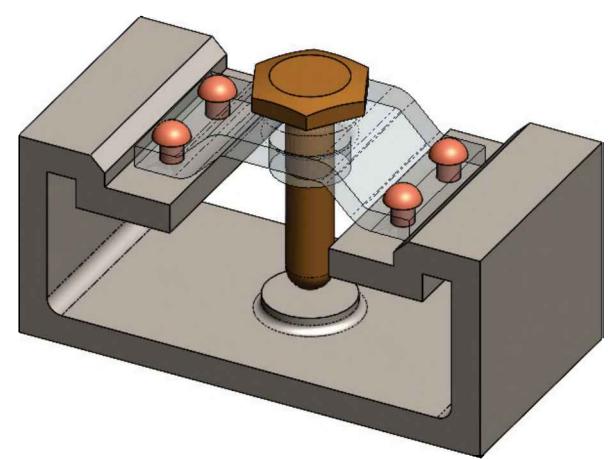
When you work with assemblies including multiple components, viewing specific components, assembly details or making selections may be difficult. One way to improve component visibility is by hiding components or making them transparent.

To hide a component select it in the FeatureManager or the screen, and from the context toolbar select the **Hide Component** command.



To Show a hidden component, select it in the FeatureManager and click in the **Show Component** command.

Figure 5-4. Transparent Component in an Assembly.



ASSEMBLY EXPLODED VIEW

Assembly components can be exploded using the **Exploded View** command from the assembly toolbar for visualization and documentation purposes. Using the **Exploded View** command, you can define the parameters for each step of the exploded view, as shown in **Figure 5-5**.

You will learn more about exploding assemblies in **Unit 8**, including animating an exploded view.

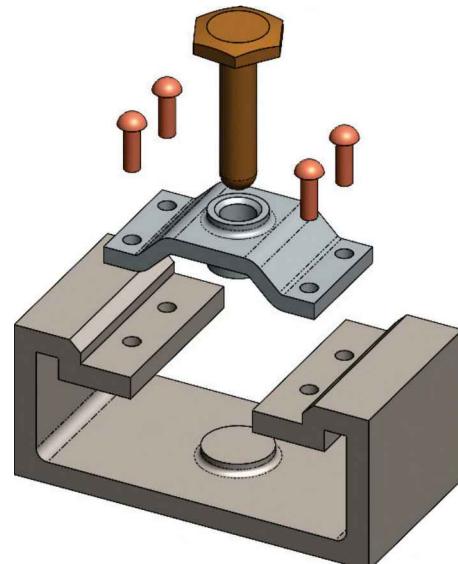


Figure 5-5. An Exploded Assembly.

Exercise 5.1: TERMINAL SUPPORT ASSEMBLY

The Terminal Support Assembly has four components: The **Frame**, the **Wing Base**, a **Pin** and four **Rivets**. You will start by designing the parts, starting with the **Frame**. When all parts are completed, you will make a new assembly, add the components, and assemble them using the **Mate** command.

FRAME

Make a new part using the **ANSI-INCHES** template and save it as **Frame.sldprt**. Add a new sketch in the **Front Plane**, draw a vertical centerline starting at the origin, activate the **Dynamic Mirror** command and draw the profile shown in **Figure 5-6**. When the profile is complete, make an **Extruded Boss**, and set the depth to **4.00"** using the **Mid Plane** end condition.

Change to a **Top** view, select the top-most face of the model and add a new sketch. Draw a horizontal and vertical centerline at the origin, draw and dimension a circle, and use the **Mirror Entities** command to create the other three holes as indicated in **Figure 5-7**. Make and **Extruded Cut** using the **Up to Next** end condition; using this option will only cut the part until the next face.

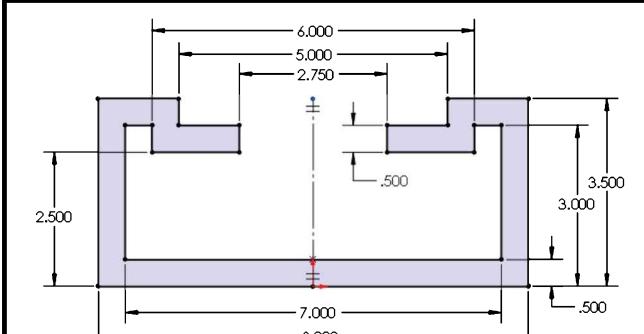


Figure 5-6. Sketch of First Feature.

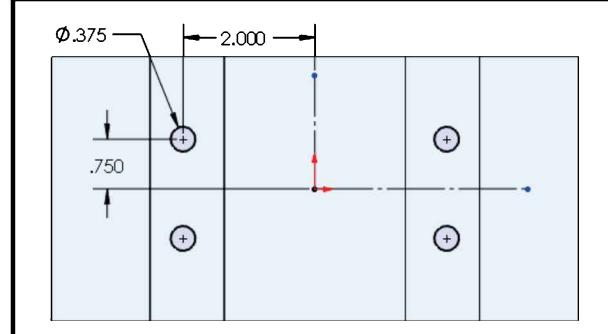


Figure 5-7. Sketch for Holes in the Top Face.

If needed, change back to a **Top** view, select the face at the bottom of the Frame part, add a new sketch and draw a circle at the origin. Dimension its diameter **1.50"** and make an **Extruded Boss 0.25"** going up.

Finish the part by adding a **0.125"** fillet at the base of this extrusion, a **0.25"** **Chamfer** to the two top inner edges and a **0.25"** fillet to the bottom inner edges of the Frame. The complete part is shown in **Figure 5-8**.

Change the part's material to **AISI 4340 Steel, annealed** and save the finished part as **Frame.sldprt**.

WING BASE

Make a new part using the **ANSI-INCHES** template and save it as **WING BASE.sldprt**.

Add a new sketch on the **Front** plane and draw the profile shown in **Figure 5-9**. Be sure to add a **Parallel** geometric relation to the angled lines.



Use the **Sketch Fillet** command to round the outside corners with a **0.125"** radius, and the inside corners with a **0.500"** radius, as shown in **Figure 5-10**.



To complete the sketch, window-select the entire sketch geometry including the centerline going through the origin and select the **Mirror Entities** to finish the sketch.

Select the **Extrude Boss** command and make a **2.500"** extrusion using the **Mid Plane** end condition and click **OK** to continue. Your part will look like **Figure 5-11**. Rename the new feature as "Base" in the FeatureManager.

Change to a Top view, select the top face of the "Base" feature and add a new Sketch. Draw a circle centered at the origin with a diameter of **1.250"**, select the **Extrude Boss** command and set **Direction 1** (up) to **0.25** inch. Set **Direction 2** (down) to **1.00**" and click **OK** to finish. Rename this feature "Round Boss" to continue.

Select the top face of this "Round Boss", add a new sketch and draw a circle concentric with the round face, centered at the origin, with a diameter of **0.75"**.

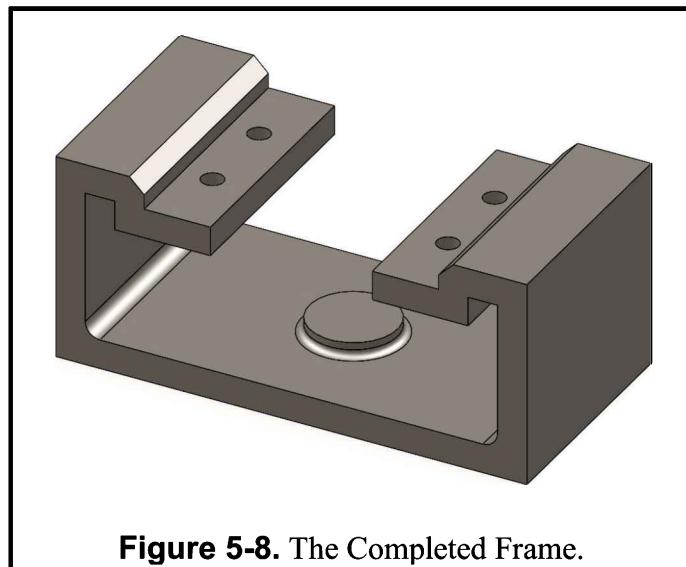


Figure 5-8. The Completed Frame.

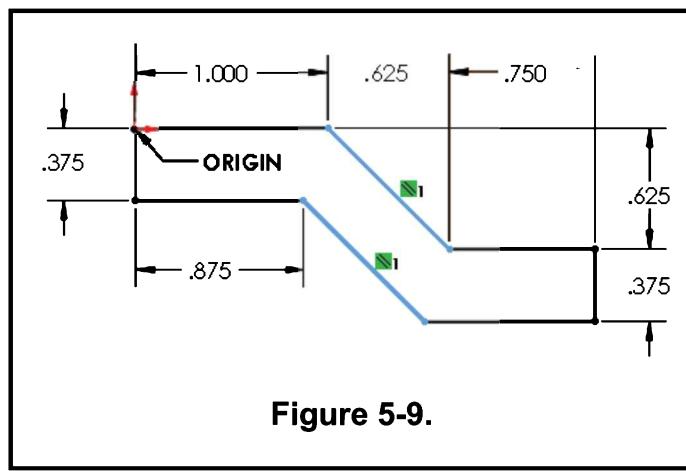


Figure 5-9.

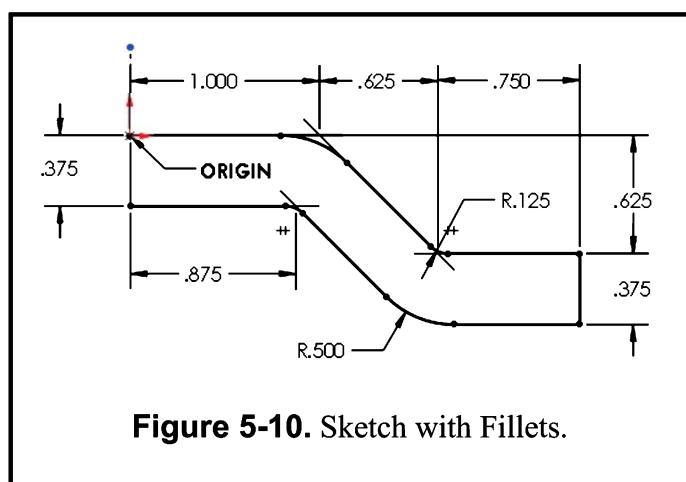


Figure 5-10. Sketch with Fillets.

Select the **Extrude Cut** command, use the **Through All** end condition and click **OK** to finish. Rename this new hole feature as "Center Hole". Your model should now look like **Figure 5-11**.

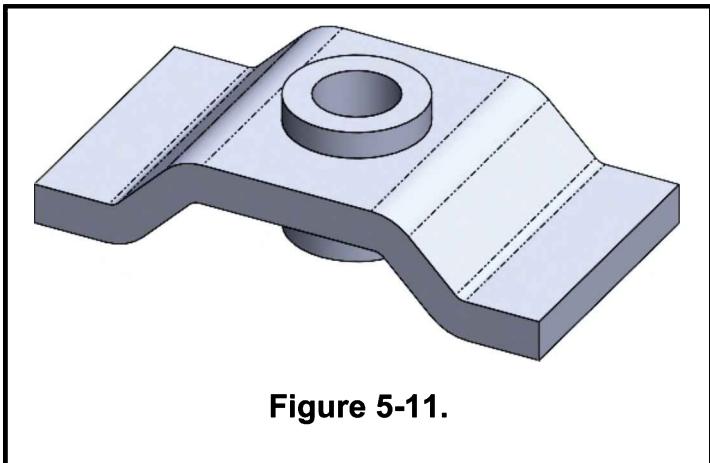
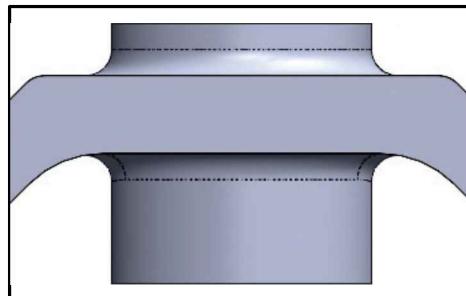


Figure 5-11.



Now you need to add a Chamfer and fillets to your model. Select the Fillet command from the Features tab, set the radius to 0.125" and select the two edges indicated next. If needed, change the Display Style to Hidden Lines Visible to make selection easier.



 To add the **Chamfer**, select the top inside edge of the "Center Hole" and from the context toolbar select the **Chamfer** command. Set the **Chamfer Type** to Angle-Distance, change the distance to **0.100"** and the angle to **45°**, and click **OK** to finish. Your model should now look like **Figure 5-12**.

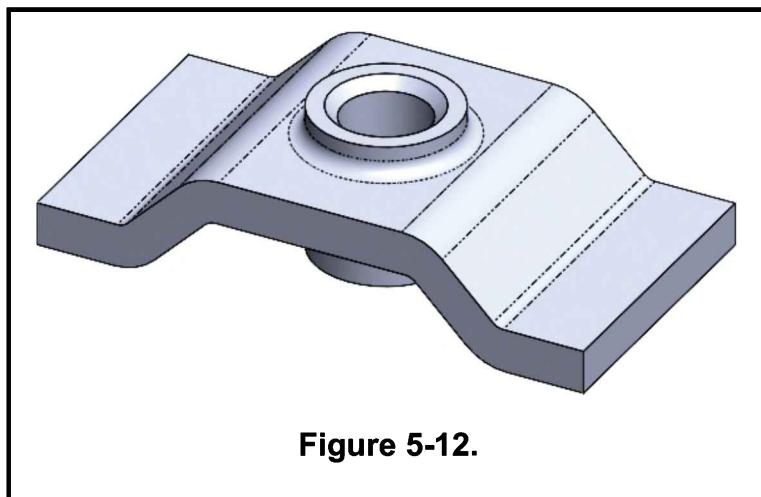


Figure 5-12.

The last feature to add to the model are the four holes for the rivets. Change to a **Top** view orientation, select the top face of either wing side and add a new sketch.

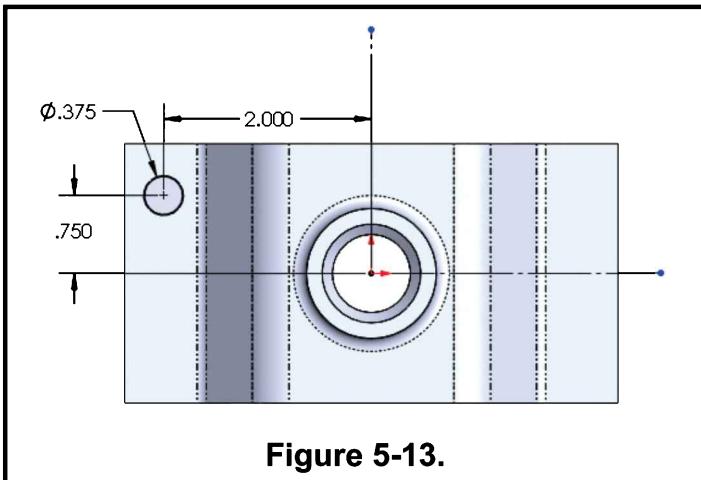


Figure 5-13.

Draw and dimension the profile shown in **Figure 5-13**, including a horizontal and a vertical centerline starting at the origin.



To better maintain the design intent to make

sure the part remains symmetrical, even if we introduce a dimensional design change, use the **Mirror Entities** command to mirror the circle about the vertical centerline first, and then select both circles and mirror them about the horizontal centerline. The finished, fully defined sketch should look like **Figure 5-14**.

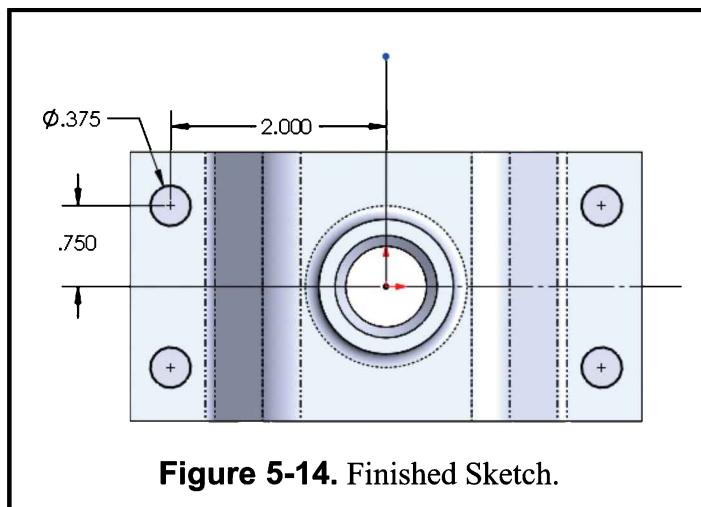


Figure 5-14. Finished Sketch.

To complete this feature, select the **Extruded Cut** command, use the **Through All** end condition and click **OK**. Rename the feature as “Rivet Holes.” Your finished part will look like **Figure 5-15**.

Set the part’s material to **Chrome Stainless Steel** and save it as **Wing Base.sldprt**. Close the part to continue.

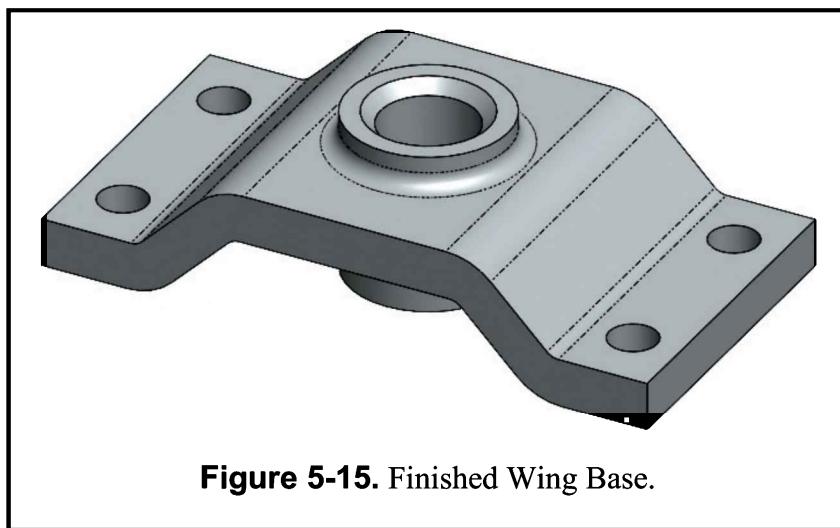


Figure 5-15. Finished Wing Base.

PIN

Next you will design the center pin. Make a new part using the **ANSI-INCHES** template and save it as **Pin.sldprt**. Add a new sketch in the **Front** Plane and draw the profile indicated in **Figure 5-16**. Remember to add a vertical centerline at the origin.



Select the **Revolve Boss/Base** command and revolve the sketch profile **360** degrees. Rename this feature as **Pin Base**.

Now you need to add the hexagonal head feature to the top of the Pin. Change to a **Top** view, select the top flat face of the Pin and add a new sketch.

Using the **Polygon** tool draw a hexagon by changing the number of sides to 6. Center the hexagon at the origin and drag a vertex horizontally out to the right to make it coincident with the edge at the perimeter of the Pin, as shown in **Figure 5-17**. Finally select the top (or bottom) horizontal line and add a **Horizontal** relation to fully define the sketch.

Change to an **Isometric** view and select the **Extruded Cut** command. Set the end condition to **Through All** and activate the **Flip side to cut** option to cut outside of the sketch profile. Click **OK** to complete the Cut Extrude and rename this feature **Hex Cut**. The finished part will look like **Figure 5-18**.

Finally, change the part's material to **2.1020 (CuSn6)** from the **DIN Copper Alloys** material section. Save your part as **PIN.sldprt** and close the file.

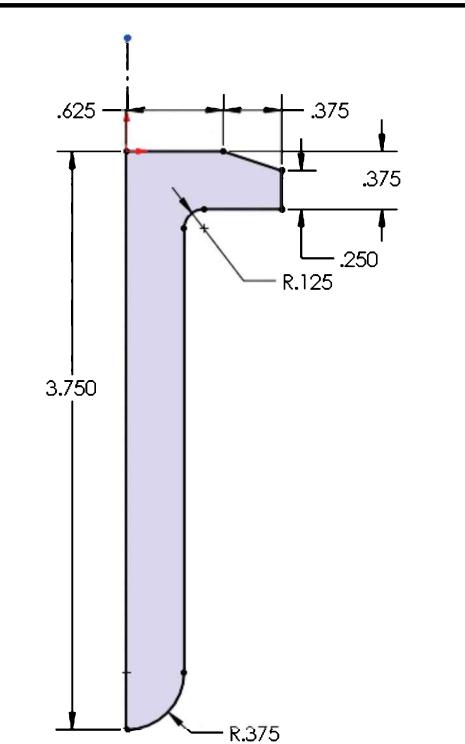


Figure 5-16. Sketch for the Pin.

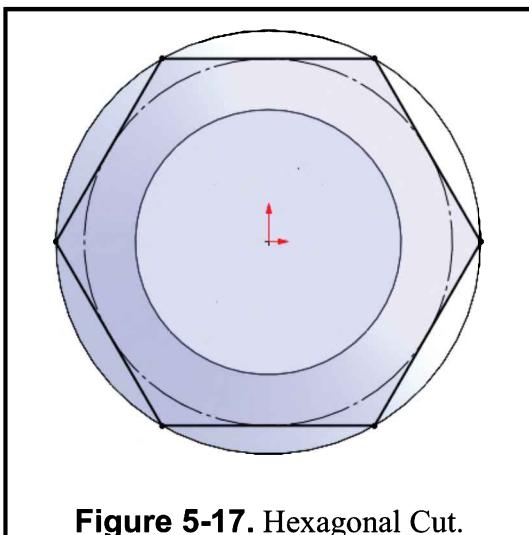


Figure 5-17. Hexagonal Cut.

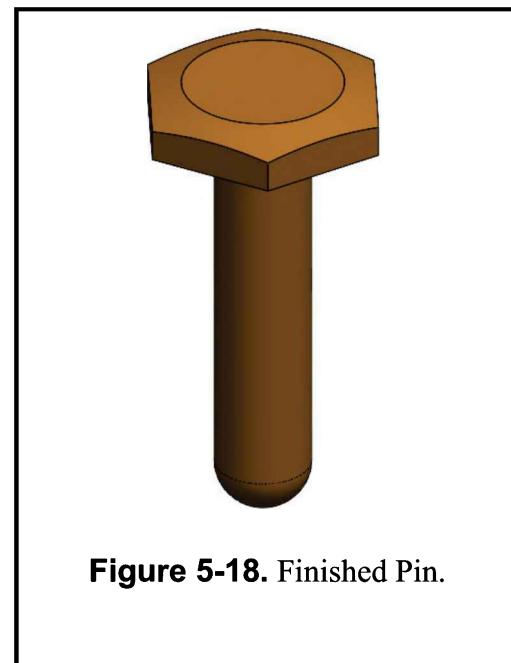


Figure 5-18. Finished Pin.

RIVET

The **Rivet** is a simple part and you can create it in a number of ways. The method chosen here is to simply sketch a half-profile and then revolve it around a centerline. There are different types of rivets (flat head, pan head, etc.). In this exercise, you will design a “Button Head” rivet.

Start a new part using the **ANSI-INCHES** template and save it as **Rivet.sldprt**. Add a sketch in the **Front Plane** and draw the profile shown in **Figure 5-19**; remember to add the centerline.

Use the **Centerpoint Arc** tool to draw the arc, making sure the arc’s center is located below the head of the rivet.

Select the **Revolve Boss/Base** command and revolve the sketch 360 degrees. Rename this feature as **Rivet Base** in the FeatureManager and change the part’s material to **2.0090 (Cu-DHP)** from the **DIN Copper Alloys** Materials list.

Save the part as **RIVET.sldprt** and close the part. Now that you have made all the parts needed you are ready to start the assembly.

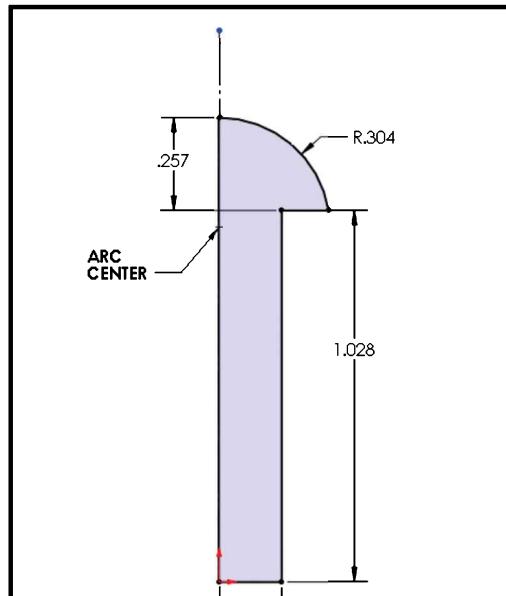


Figure 5-19. Sketch for Rivet.

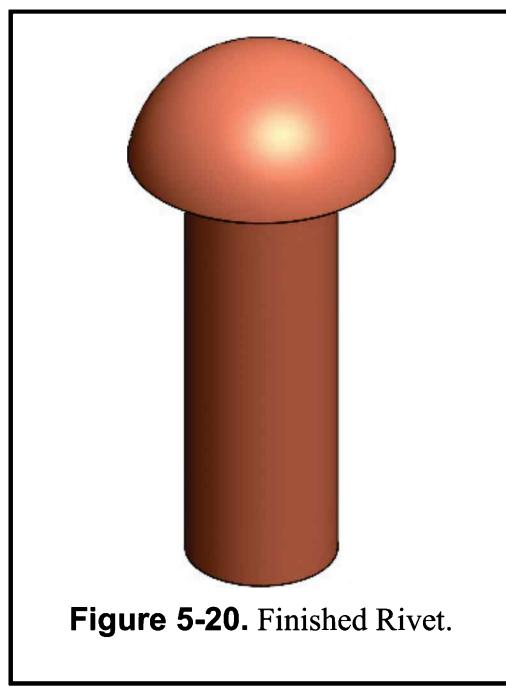


Figure 5-20. Finished Rivet.

TERMINAL SUPPORT ASSEMBLY

There are multiple ways to make a new assembly. In this lesson we'll use the new assembly's automatic command to add components to the assembly. Before you start, open the **Frame**, **Wing Base**, **Pin**, and **Rivet** parts. The reason to open the parts before starting the assembly is to facilitate their selection.

Select the **New** command from the main toolbar, or the menu **File**, **New**. In the **New SOLIDWORKS Document** window select the **Assembly** template and click **OK** to continue.

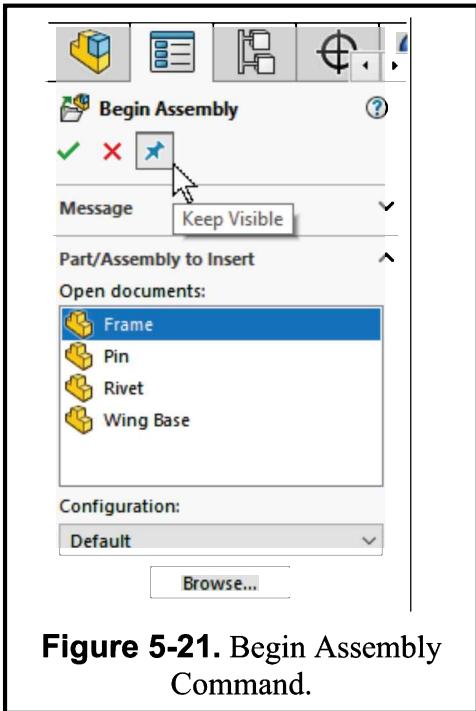


Figure 5-21. Begin Assembly Command.

When the new assembly is created, the **Begin Assembly** command is automatically loaded. At the top of the command options, select the **Keep Visible** option to keep the dialog open. In the **Open Documents** list select the **Frame** part (or **Browse** to locate it if it is not currently open) and click **OK**, as seen in **Figure 5-21**.

This way the selected part's default planes are automatically aligned with the assembly's default planes. Remember the first part added to the assembly is **FIXED** and cannot be moved unless its constraints are deleted. It will be the base to which all other parts are mated.

The reason why this is desirable, is to be able to leverage the assembly's planes to maintain your design intent, especially if the assembly has symmetry about any of those planes.

With the **Begin Assembly** command still open, select the **Wing Base** part, rotate the assembly, and click in the screen to locate it above the **Frame**. Repeat the same process with the **Pin** and add the **Rivet** four times, locating them approximately as shown in **Figure 5-22**.

After you have added all components to the assembly click **OK** to continue. The next step is to accurately locate all the components using the **Mate** command.

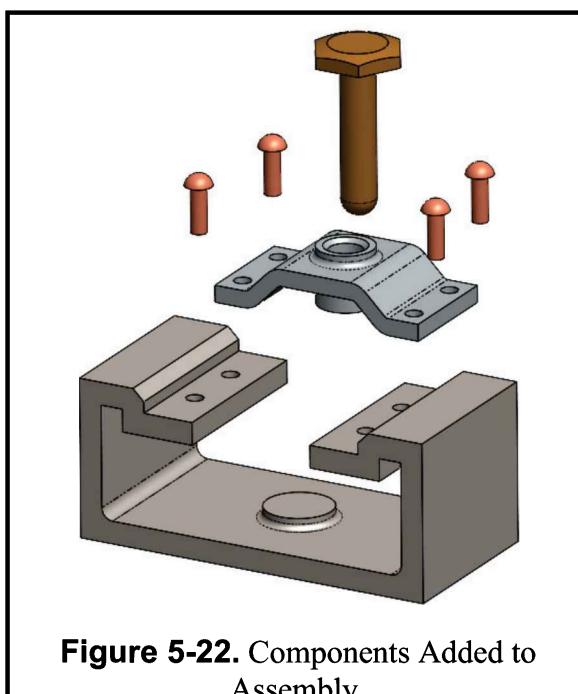


Figure 5-22. Components Added to Assembly.



The first component to be assembled to the **Frame** will be the **Wing Base**. In the Assembly tab select the **Mate** command. First select the inside face of a hole of the **Wing Base** (not the edge) and the inside face of the corresponding hole in the **Frame**. Both faces are added to the Mate Selections list.



By selecting two cylindrical faces, SOLIDWORKS automatically pre-selects the **Concentric** mate as the most likely option and moves the **Wing Base** (the part that can move) towards the **Frame** (the Fixed part). Click **OK** to add this mate and continue.



Click and drag the **Wing Base**; notice how it rotates about the hole's axis and moves up and down. The next mate you will add is to restrict the vertical motion of the **Wing Base**. With the **Mate** command still visible, select the top flat face of the Frame with the holes, and the bottom face of the **Wing Base**. Since both faces are planar, the **Coincident** mate type is pre-selected. Click **OK** to add the mate to continue.



When you click and drag the **Wing Base**, notice it only rotates about the hole's axis. To restrict this motion, select the right side faces of the **Wing Base** and the **Frame**, and select the **Parallel Mate**.

Optionally, you could select the cylindrical faces of a second pair of holes to add a **Concentric** mate. The potential problem with this approach is that if the hole's centers are not *exactly* at the same distance, SOLIDWORKS would not be able to add the mate, over defining the assembly. Instead, the **Parallel** mate is more forgiving and can absorb small dimensional differences.

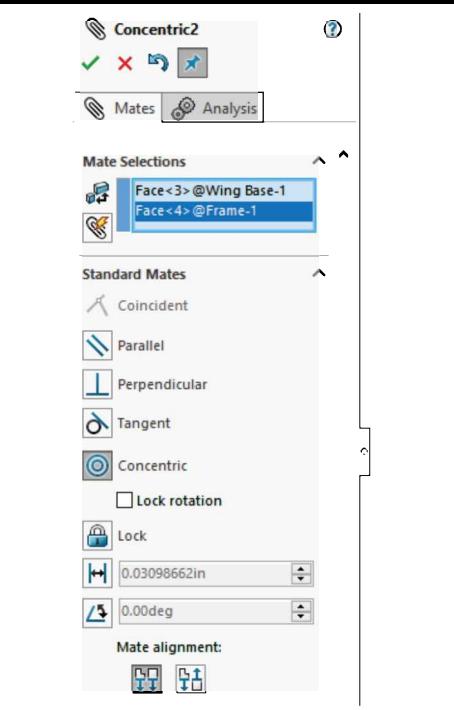
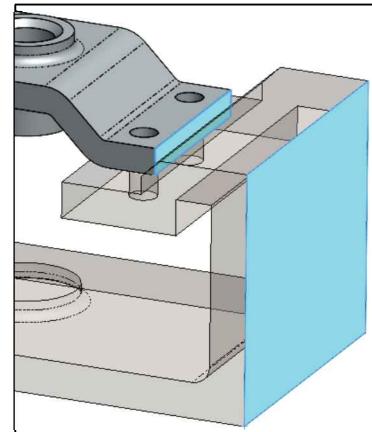


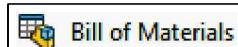
Figure 5-23. Mate Command.



Now you need to mate the Pin to the **Wing Base**. Select the inner cylindrical face of the Center Hole and the outer cylindrical face of the **Pin**. Just as before with the holes, the Concentric mate is pre-selected. Click **OK** to add the mate. If you click and drag the **Pin**, it will move along the axis. Now select the face at the top of the **Wing Base** and the matching face at the bottom of the **Pin**. Zoom and rotate the assembly as needed to select the faces to be mated.

While still using the **Mate** command, add four more **Concentric** mates between the **Rivets** and their corresponding holes, and finally add a **Coincident** mate between the bottom face of each **Rivet** head to the **Wing Base**. Close the **Mate** command; the finished assembly is shown in Figure 5-24.

Save the assembly as **TERMINAL SUPPORT.sldasm** and make a new drawing with an Isometric view, using the **Shaded with Edges** display style.



To complete the drawing, you need to add a **Bill of Materials** and **Identification balloons**. Pre-select the Isometric view and click in the menu **Insert, Tables, Bill of Materials**.

Select the options **Parts Only** and **Display all configurations of the same part as one item**. Click **OK** to accept the parameters and click to locate the **Bill of Materials** in the upper left corner of your drawing.

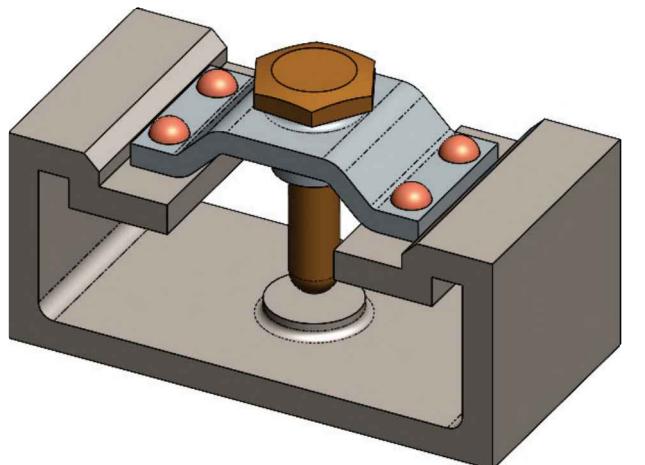


Figure 5-24. Finished Assembly.

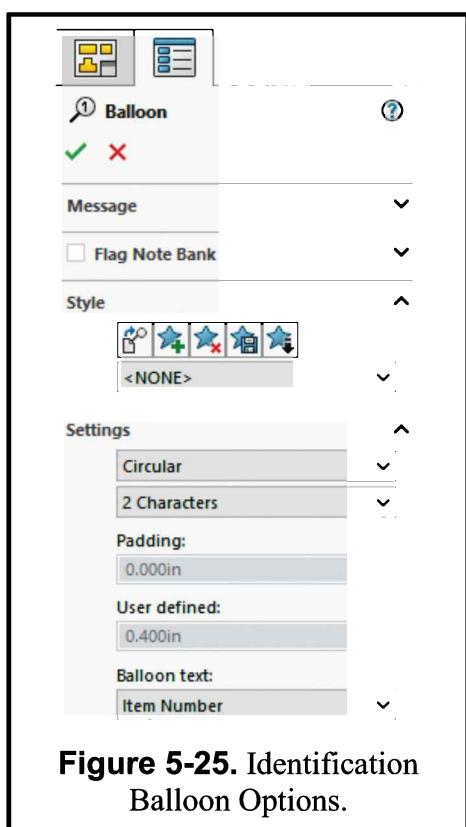
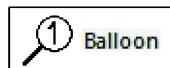


Figure 5-25. Identification Balloon Options.



To add identification balloons, select the Annotation tab and click in the **Balloon** command, or from the menu **Insert, Annotations, Balloon**. In the Settings section, select the **Circular** option, and **Item Number**, as shown in **Figure 5-25**.

After setting these options, click on each component of the assembly and locate the corresponding balloon in the drawing. You can select a part's face, edge or vertex as needed.

Optionally change the balloon's font size. Select the four balloons and click in “**More Properties...**”, turn off the “**Use document font**” option and use the **Font** command to change the balloons to use a **16-point** size font.



After completing the balloons, select the **Note** command from the Annotations tab, and add a note next to each balloon with the component's name. Set the font size to also use a 16-point.

As an alternative to changing the font style individually, you can select the menu **Tools, Options, Document Properties**, and select the corresponding section to set the default font for Annotations, Balloons, Notes, etc. *Remember that every option changed in the Document Properties section can be saved to an assembly template.*

Design Workbook Using SOLIDWORKS 2020

Save your drawing as **TERMINAL SUPPORT.slddrw** and **Print** a copy of the finished drawing for your lab instructor.

Do not discard your assembly files; they will be used in a later lesson to learn how to create an **Exploded View** and animation.

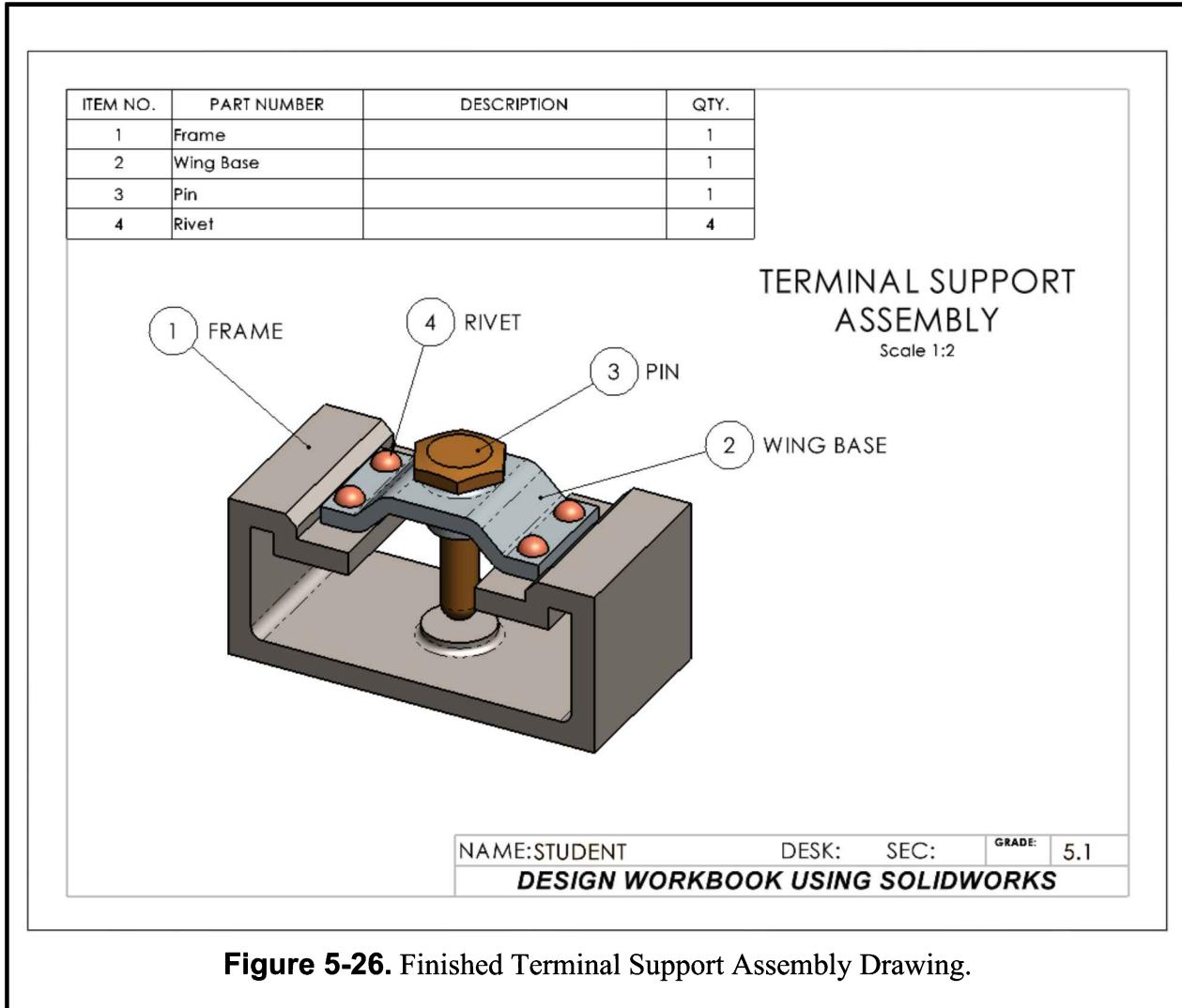


Figure 5-26. Finished Terminal Support Assembly Drawing.

Exercise 5.2: PULLEY ASSEMBLY

The Pulley Assembly is a simple design used to hoist objects with a mechanical leverage advantage. It has four major components: An **Eye Hook**, a **Pulley Sheave**, a **Spacer**, and a **Base Plate**. It is assembled using small rivets that are peened on one end to secure them and hold the components together. You will start by designing the **Base Plate**, which is the part that drives the overall dimensions of the other components.

BASE PLATE

Start a new part using the **ANSI-Inches** template. Add a new sketch in the **Front Plane** and draw the profile shown in **Figure 5-27**. Since it is symmetrical, you

 **Dynamic Mirror** command. Remember to add Tangent relations between the lower arc and the inclined lines.

Select the **Extruded Boss** command and extrude the profile to **0.125"**. Change the part's material to **AISI 4340 STEEL, NORMALIZED** and save your part as **Base Plate.sldprt**.

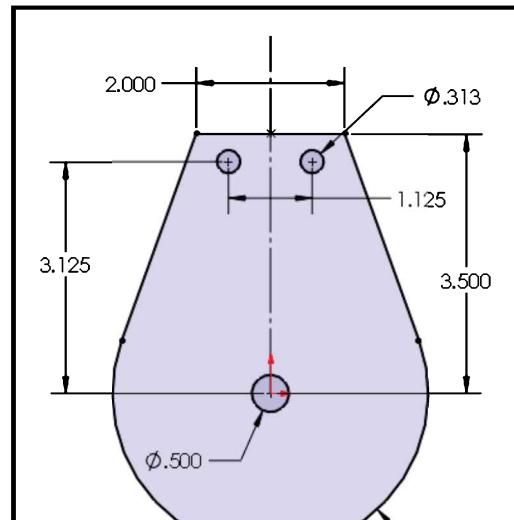


Figure 5-27.

PULLEY

Start a new part using the **ANSI-INCHES** template, add a sketch in the **Right Plane** and draw the sketch shown in **Figure 5-29**.

Draw a vertical Centerline for the **Dynamic Mirror** command, and a horizontal centerline to use it as the axis of rotation. Use the Dimension tool to fully define the sketch. Use the menu Tools, Options, Document Properties, Units to change the number of significant digits to four decimal places.

Now select the **Revolve Boss** command and make the revolved feature **360** degrees about the horizontal centerline. You now have the pulley part of the assembly.

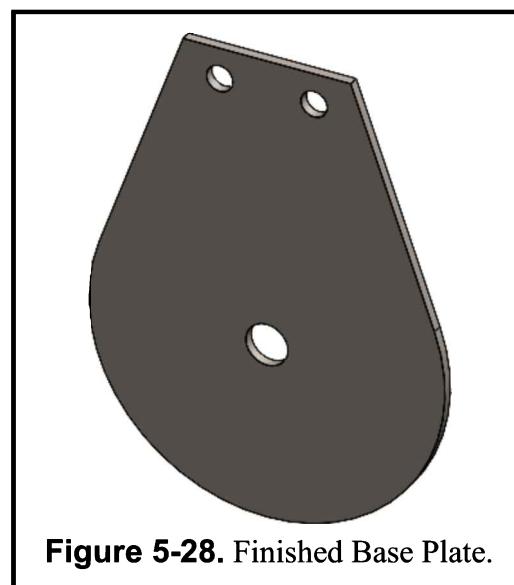


Figure 5-28. Finished Base Plate.

Set the part's material to **2.0375 (CuZn36Pb3)** from the **DIN Copper Alloys** section. Finally, save your part as **Pulley.sldprt** to finish.

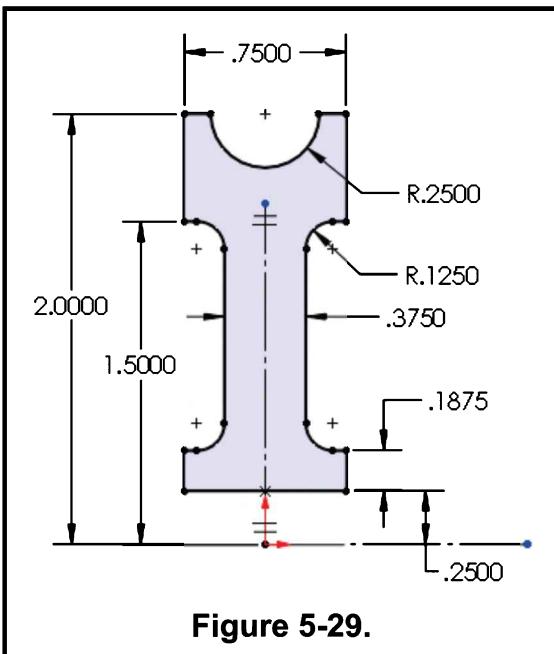


Figure 5-29.

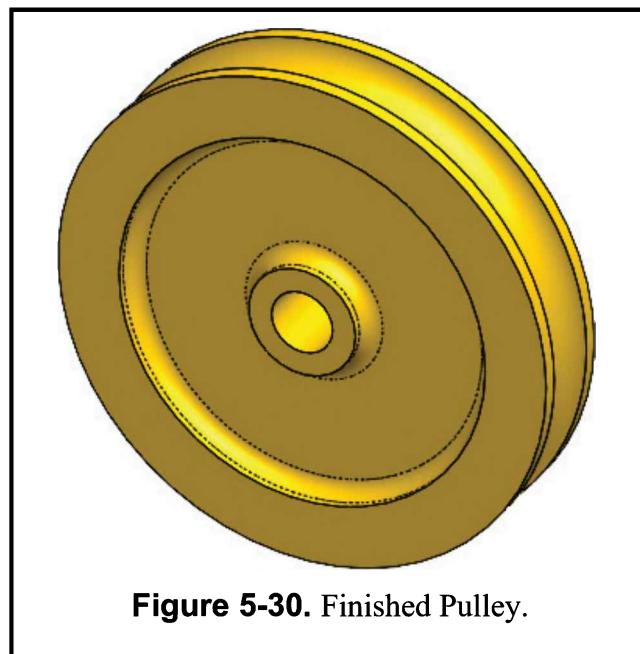


Figure 5-30. Finished Pulley.

SPACER

Start a new part using the **ANSI-INCHES.prtdot**, add a new sketch in the **Front Plane** and draw the profile shown in **Figure 5-31**. Use the **Center Rectangle** and **Circle** commands and dimension as indicated.

Use the **Dynamic Mirror** to make the two circles symmetric and add a **Horizontal** relation between the center of one circle and the origin to fully define your sketch. When the sketch is complete, extrude it using the **Mid Plane** end condition a distance of **1.00"**.

Remember the **Mid Plane** option will extrude the sketch half of the specified distance in each direction, making the extrusion symmetrical about the sketch plane.

Next, change to a **Top View** and add a new sketch on the top face. Draw a circle in the origin with a diameter of **0.350"** and make an **Extruded Cut** using the **Through All** end condition, as shown in **Figure 5-32**.

Set the material to **AISI Type A2 Tool Steel** and save your part as **Spacer.sldprt** to finish.

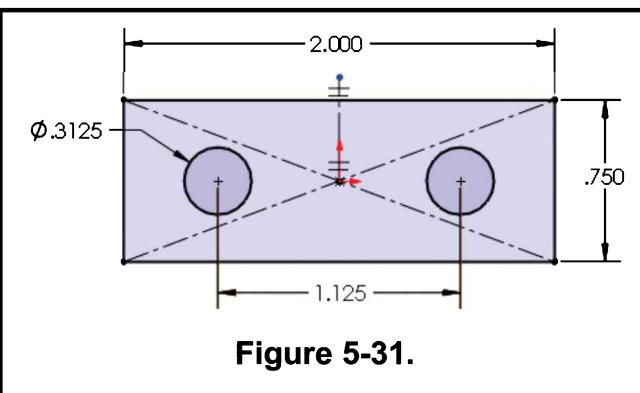


Figure 5-31.

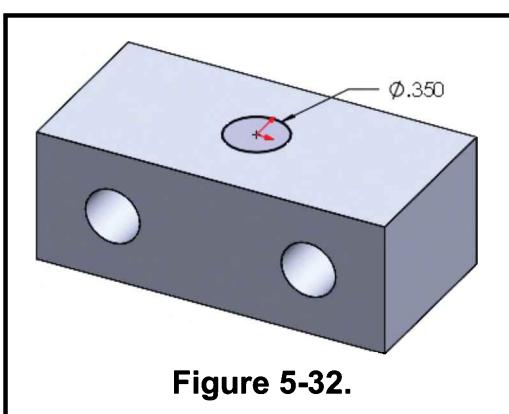


Figure 5-32.

EYE HOOK

The next part for the assembly is the **Eye Hook**. It can be made by making a revolved boss and an extrusion.

Start a new part in inches and set the dimensions to four decimal places. Add a new sketch in the **Right Plane** and draw the profile shown in **Figure 5-33**. Add a **Vertical** relation between the origin and the circle's center to fully define your sketch.

Select the **Revolved Boss** command and revolve the circle **360** degrees to create the top part of the Eye Hook.

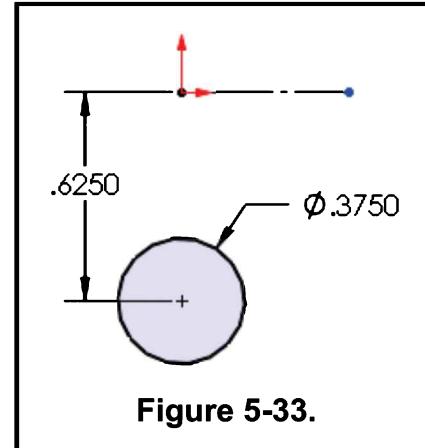
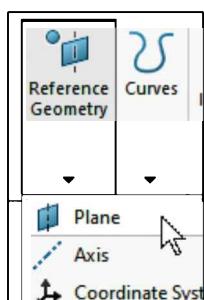


Figure 5-33.



To make the bottom half you need to add an auxiliary plane. Select **Reference Geometry** and click on **Plane**. From the fly-out FeatureManager select the **Top Plane** (or pre-select it before starting the **Plane** command), set the distance to **0.625"** below and click **OK** to complete it.

Select the new plane, add a new sketch and draw a circle in the origin with a **0.3475"** diameter, as shown in **Figure 5-34**. Make an Extruded Boss **1.25"** downward. Your completed **Eye Hook** part should look like **Figure 5-35**.

Change the part's material to **Tin Bearing Bronze** from the **Copper Alloys** library and save it as **Eye Hook.sldprt**.

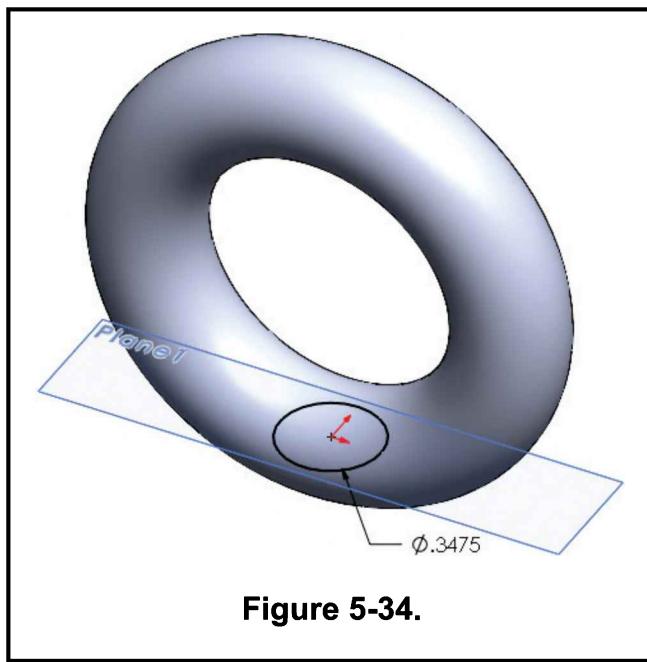


Figure 5-34.

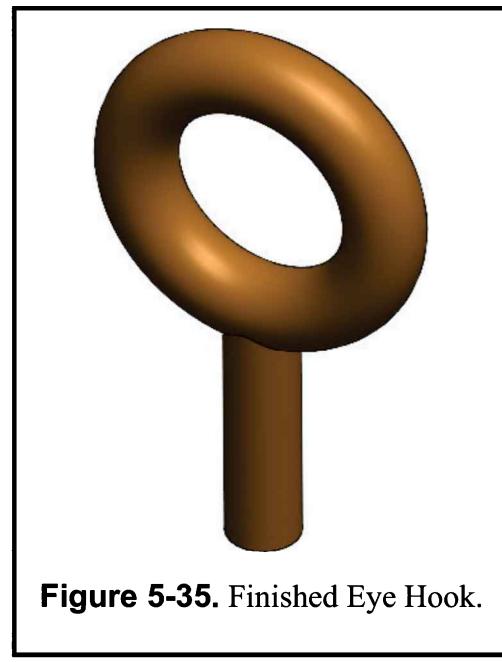


Figure 5-35. Finished Eye Hook.

RIVETS

The last parts you have to make are the **Small** and **Big Rivets**. These are standard components whose dimensions can be found in engineering handbooks.

For both components start a new part using the template in Inches, add a new sketch in the **Front Plane** and draw the corresponding profile shown in **Figure 5-36** and **5-37** for the **Big Rivet** and **Small Rivet**, respectively. Remember to add a centerline and make a 360° **Revolved Boss** to complete the parts.

Set the material on both parts to **2.0060 (Cu-ETP)** from the DIN Copper Alloys list found in SOLIDWORKS DIN Materials. To finish, save each part as **Big Rivet.sldprt** and **Small Rivet.sldprt**, respectively.

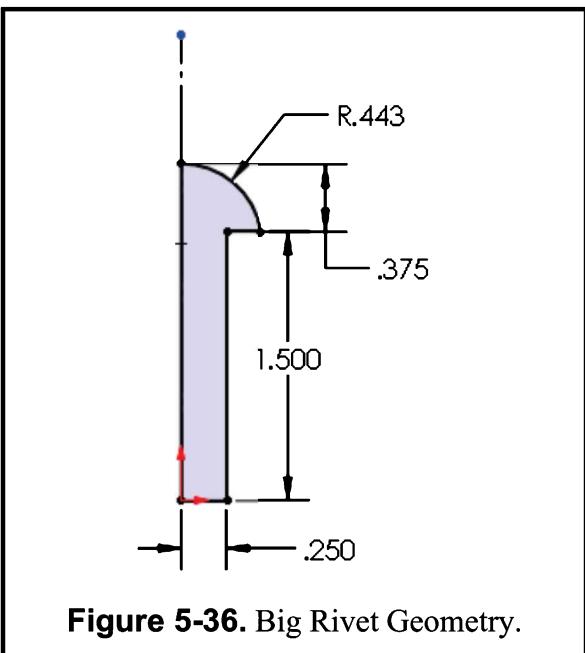


Figure 5-36. Big Rivet Geometry.

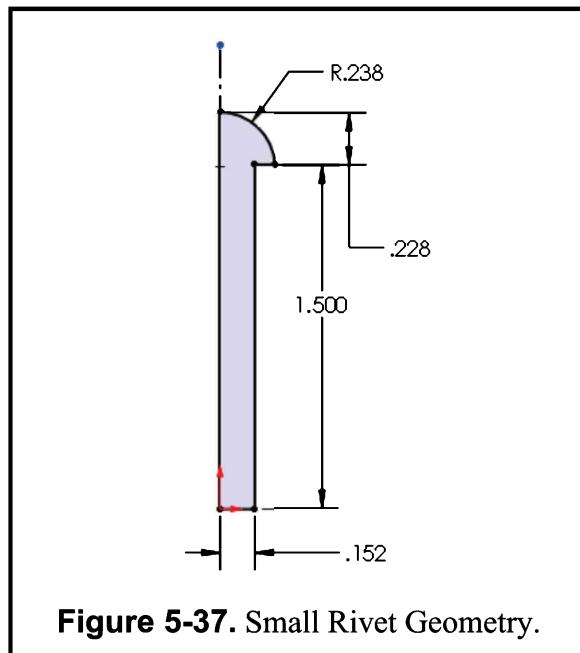
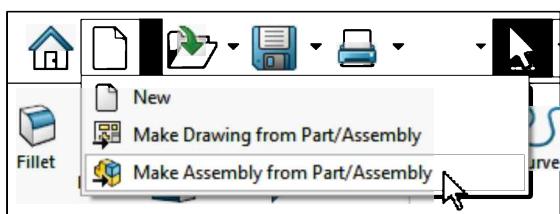


Figure 5-37. Small Rivet Geometry.

SWIVEL EYE BLOCK ASSEMBLY

Now you are ready to start the new assembly. In the previous exercise you opened all the parts needed, and then added them from the **Begin Assembly** command. In this exercise you will use a different approach.



The first component added to this assembly will be the **Spacer**. Open the **Spacer** part, and select the menu **File, Make Assembly from Part**, or from the **New** command's drop-down list select **Make Assembly from Part/Assembly**.

Select the default **Assembly** template and click **OK** to continue (if you made an assembly template in Inches you can use it, too).

You are immediately presented with the **Begin Assembly** command. Make sure the **Spacer** is selected in the **Open Documents** list and click **OK**. Remember the **Spacer** will be added to the assembly and its default planes will be aligned with the assembly's default planes.

If needed, go to the menu **Tools, Options, Document Properties** and set the dimensioning standard to **ANSI**. Go to the **Units** section and change the assembly's units of measure to **Inches** with three decimal places and save your assembly as **Swivel Eye Block.sldasm**.



In this exercise we'll add the components to the assembly and mate them one at a time. From the Assembly tab select the **Insert Component** command, select the **Base Plate** from the Open Documents list, or browse to locate it. Then click in the graphics area to add it to the assembly, as seen in **Figure 5-38**.

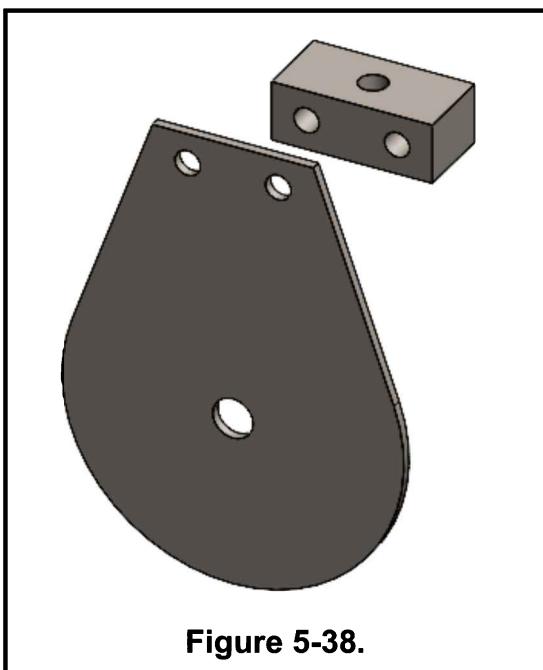


Figure 5-38.



Now you need to mate the **Base Plate** to the **Spacer**. Select the **Mate** command from the Assembly tab or the menu **Insert, Mate**. Select the upper left hole's cylindrical face of the **Base Plate** and the corresponding **Spacer's** hole. Since both faces are cylindrical, the **Concentric** mate is automatically pre-selected. Click **OK** to add this mate. Repeat the same mate using the faces of the other pair of holes.

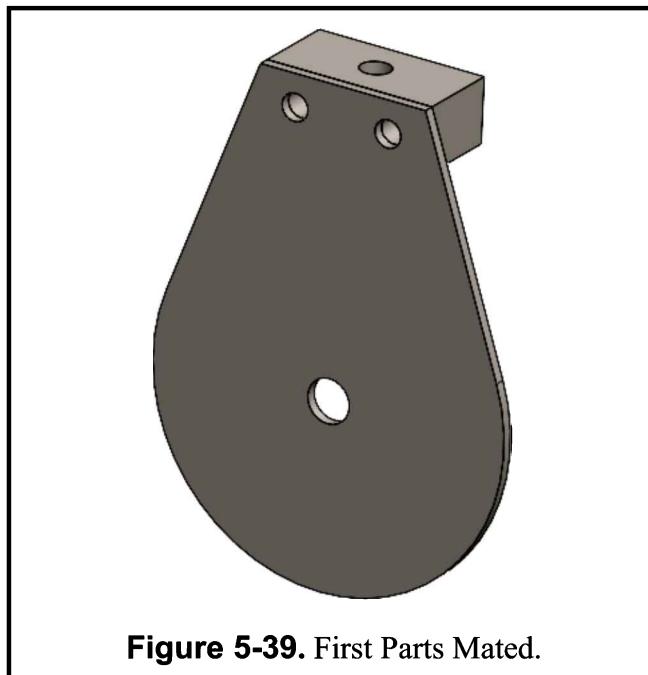
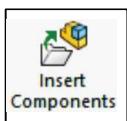


Figure 5-39. First Parts Mated.

If needed, click and drag the **Base Plate** away from the **Spacer**, and select the inside face of the **Base Plate** and the outside face of the **Spacer**. Add a **Coincident** mate, click **OK** and close the **Mate** command. Your assembly will look like **Figure 5-39**. The **Base Plate** is now fully defined in reference to the **Spacer**.



Use the **Insert Components** command to add a second copy of the **Base Plate**, click to add it to the assembly and mate it to the other side of the **Spacer** using the same mates as before. **Zoom** and **Rotate** the assembly as needed to select the faces to be mated.

Close the **Mate** command to continue. Your assembly should look like **Figure 5-40**.

Now you need to add the Pulley to the assembly. Use the **Insert Components** command and locate it in the assembly.

Select the **Mate** command and add a **Concentric** mate between the face at the center hole of the **Base Plate** and the cylindrical face at the center of the **Pulley**.

After this mate is added, the pulley can move along the hole's axis and is not centered about the **Base Plates**.

There are different ways to center the **Pulley**:

You can add a **Distance Mate** between model faces, but this approach would probably not maintain your design intent to keep the **Pulley** centered if the **Spacer** or the **Pulley** change in size.

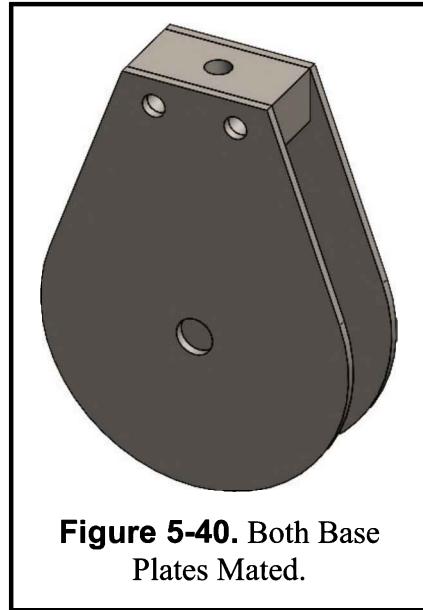


Figure 5-40. Both Base Plates Mated.

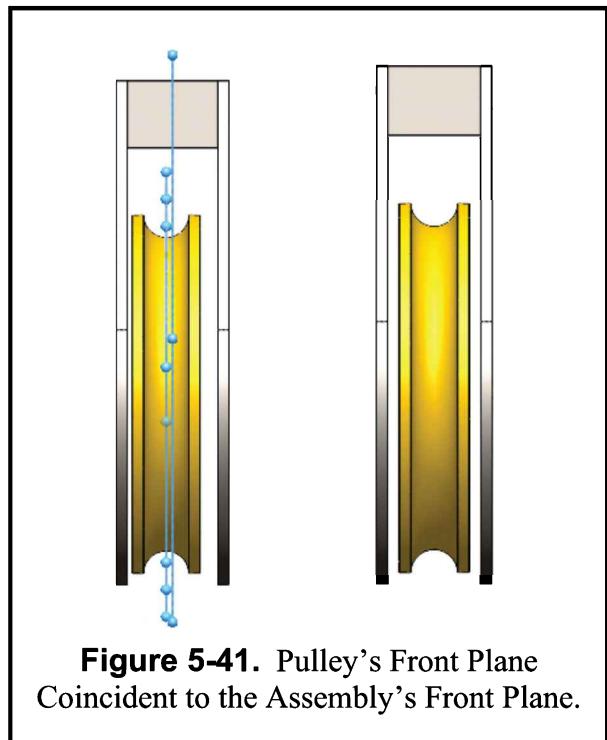


Figure 5-41. Pulley's Front Plane Coincident to the Assembly's Front Plane.

A different approach is to take advantage of the symmetry built into your parts. In this case, you can add a **Coincident Mate** between the **Pulley's Front Plane** and the assembly's **Front Plane**, which is coincident to the **Spacer's Front Plane**.

Change to a **Right** view orientation, expand the FeatureManager, and pre-select the Assembly's **Front Plane**, and the **Pulley's Front Plane**. Click in the **Mate** command and add a **Coincident** Mate. Now your Pulley is centered and will remain centered even if the size of the **Spacer** or the **Pulley** changes. See **Figure 5-41**.

Next you need to add the **Eye Hook** to the assembly. Use the **Insert Component** command and locate it above the **Spacer**. Use the **Mate** command to add a Concentric Mate between the top hole in the **Spacer** and the circular shaft of the **Eye Hook**. Change to a **Right** view and drag the Eye Hook enough for it to pass through the **Spacer**, as shown in **Figure 5-42**.

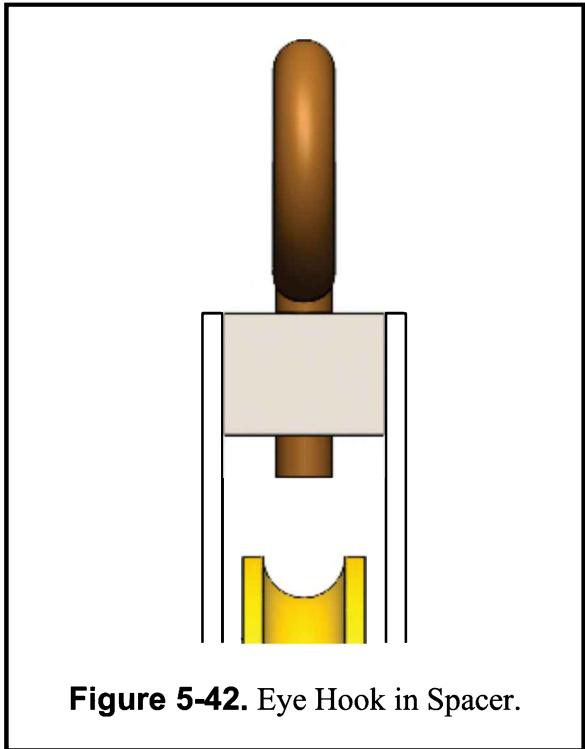


Figure 5-42. Eye Hook in Spacer.

Rotate the view to see the bottom of the **Eye Hook** and the **Spacer**, select the bottom faces of both parts and add a **Distance Mate**. Set the distance to **0.125"**. Preview the mate and make sure the **Eye Hook** passes through the **Spacer**; if it does not, click in the **Flip Dimension** option to fix it.



When finished, close the **Mate** command and expand the **Mates** folder in the FeatureManager. You will see all the mates added so far to your assembly, as shown in **Figure 5-43**.

Now that the major components have been assembled, you can see the **Spacer** is Fixed in place, indicated by the letter (f), the **Base Plates** are fully defined, and the **Pulley** and **Eye Hook** have one un-constrained degree of freedom indicated by the (-) sign next to their name in the FeatureManager, and able to rotate about one axis. See **Figure 5-44**.

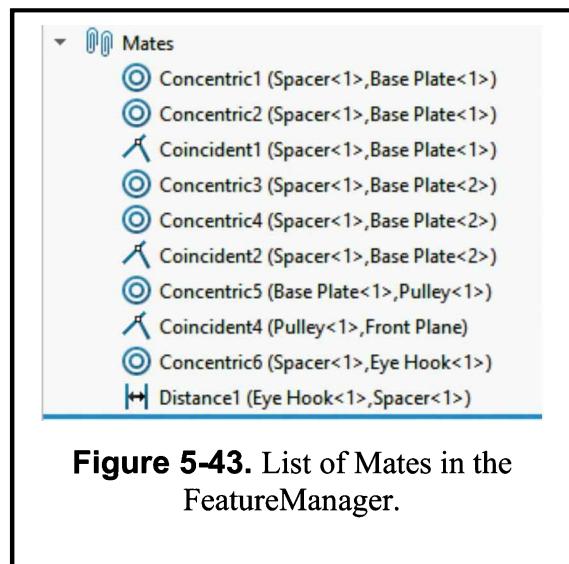


Figure 5-43. List of Mates in the FeatureManager.

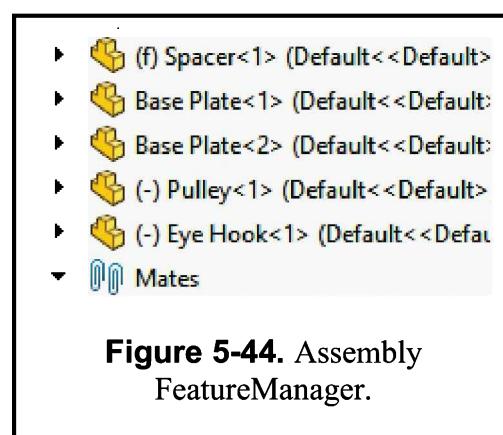


Figure 5-44. Assembly FeatureManager.

To verify their motion, click and drag them to see their rotation. The motion will be easier to visualize in the **Eye Hook**.

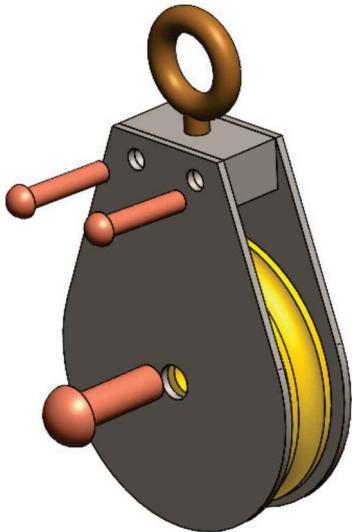


Figure 5-45. Rivets Added to Assembly.

To finish the assembly, now you need to add the rivets. Use the **Insert Components** command to add two **Small Rivets** and one **Big Rivet**. After adding the rivets, you can right-click on them to rotate and then click-and-drag to locate them approximately as shown in **Figure 5-45**.

Now use the **Mate** command to assemble the three rivets. Each rivet will have a **Concentric** mate between the rivet's shaft and their corresponding hole, and a **Coincident** mate between the bottom face of the rivet's head and the **Base Plate**.

After the Rivets have been mated, your assembly should look like **Figure 5-46**. Save your assembly as **Swivel Eye Block.sldasm** to finish.

Manufacturing Note: If you rotate the assembly, you will see the Rivets extending about 0.25" on the other side. The way

these rivets are assembled is with a peen operation. Peening the end of the rivet with a hammer or automatic press causes the flat end of the rivet to deform in a spherical shape, preventing the rivet from coming out of the hole.

Next make a new drawing and add an Isometric view using the Shaded with Edges display style.

In the Drawing sheet, select the menu **Tools**, **Options**, **Document Properties**, and under the **Annotations** section select **Balloons** and change the **Font** to Arial Regular, and a Height of **18 pt**. Click **OK** to accept the changes.

From the Annotations tab select the **Balloon** command and select each component to add their respective **Item Number**. After adding the balloons, select the **Note** command, use the No Leader option and add the part name using an **18 pt** font next to the corresponding balloon.

Now select the assembly view and click in the menu **Insert**, **Tables**, **Bill of Materials**. Use the option Select Parts only, Display all Configurations of the same part as one item and click **OK**. Locate the Bill of Materials in the upper right corner of your drawing

Save your drawing as **PULLEY ASSEMBLY.slddrw** and **Print** out a copy for your lab instructor, as shown in **Figure 5-47**.

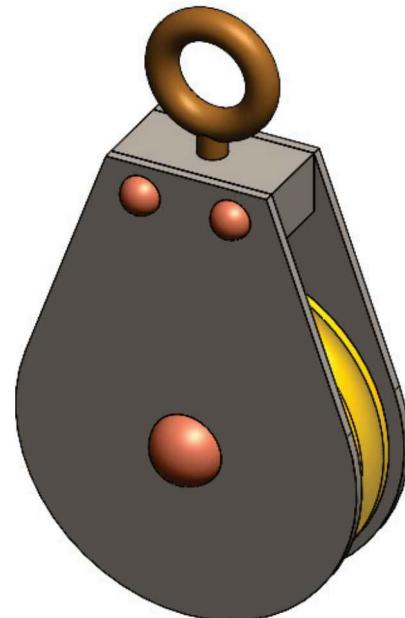
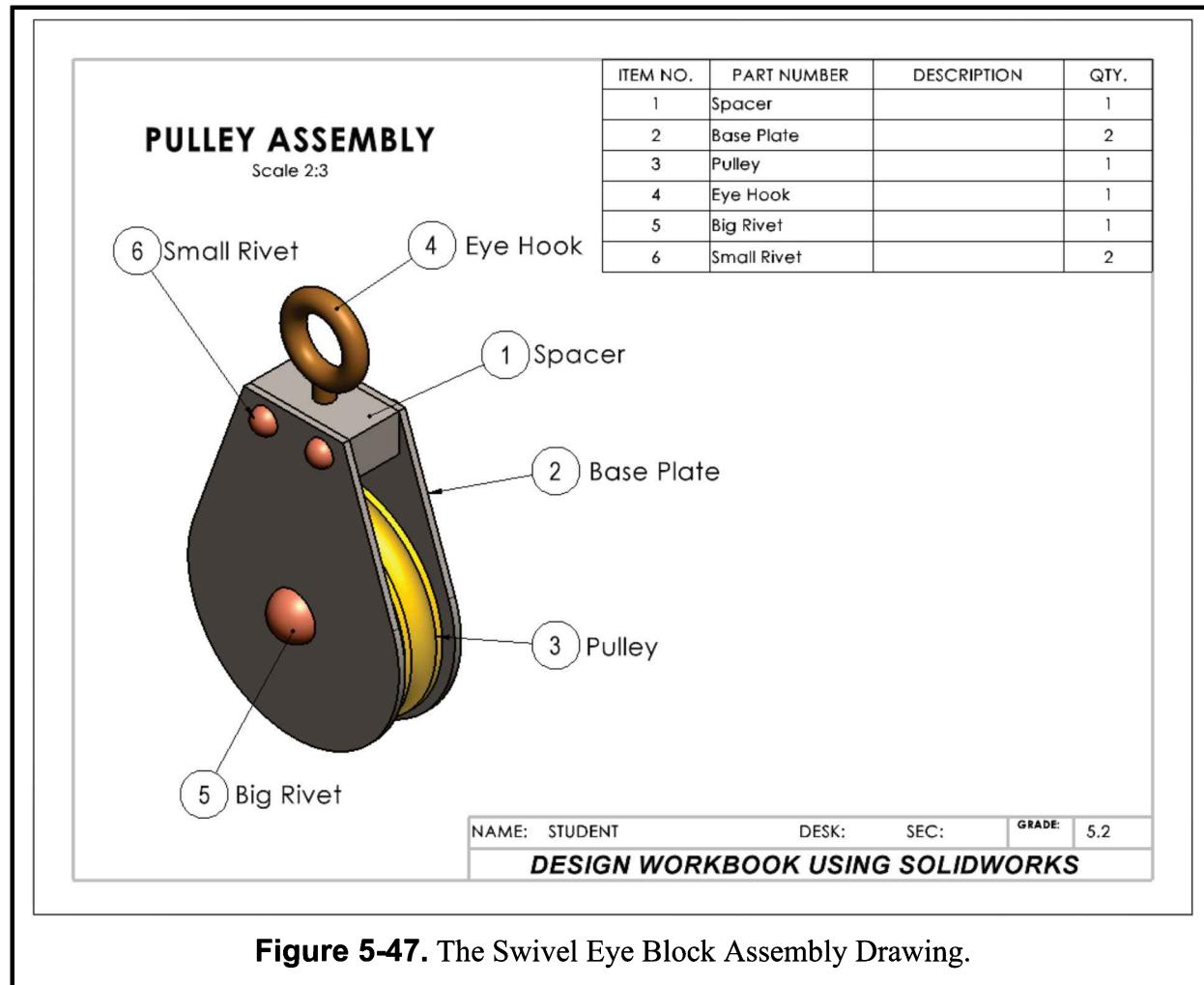


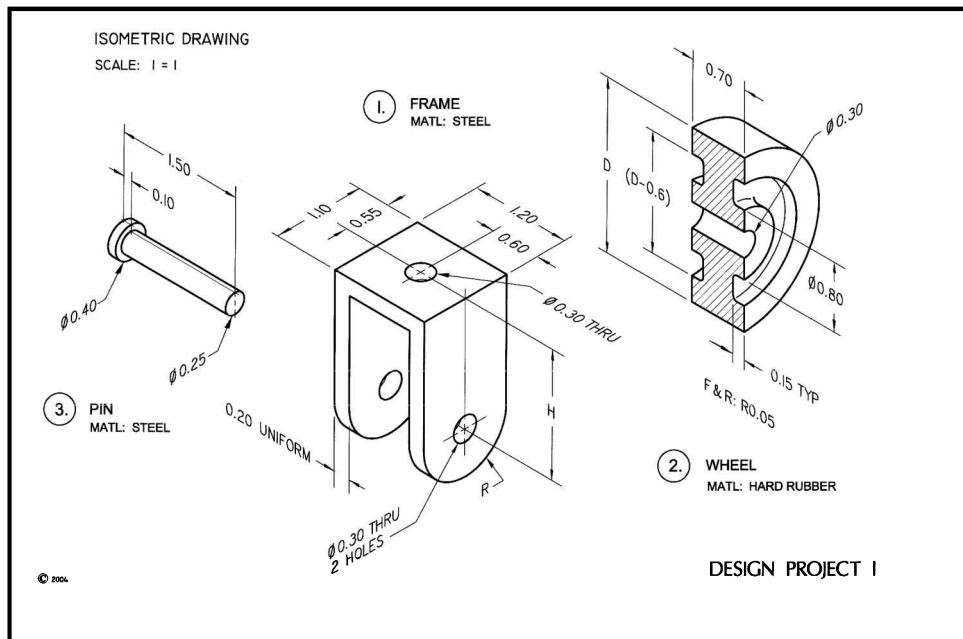
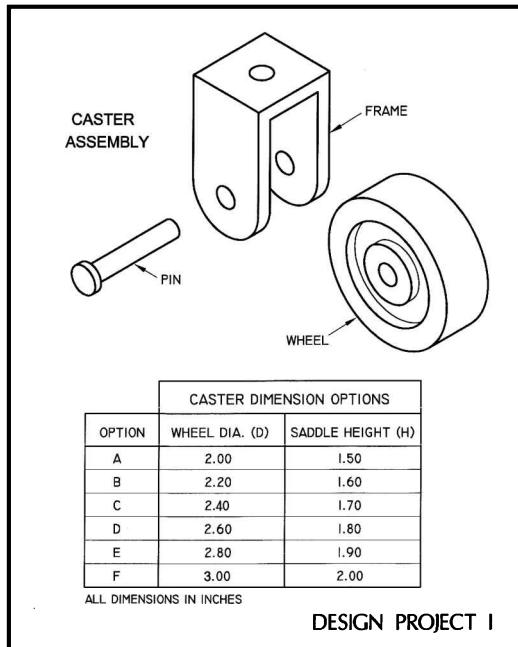
Figure 5-46. Finished Swivel Eye Block Assembly.



Supplementary Exercise 5-1: CASTER ASSEMBLY

Create the parts of the Caster Assembly of the Option specified by your instructor in the Table provided.

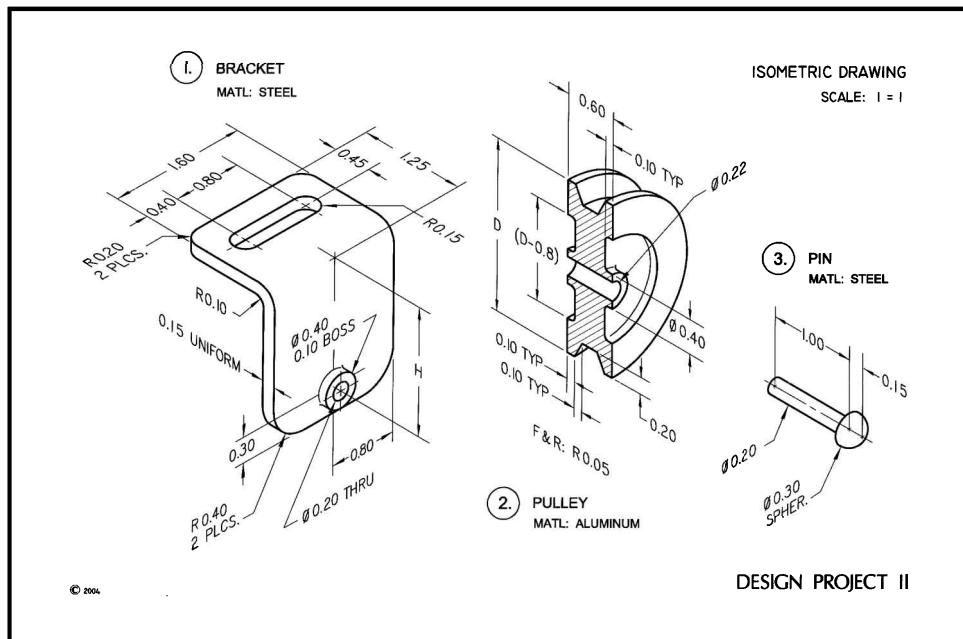
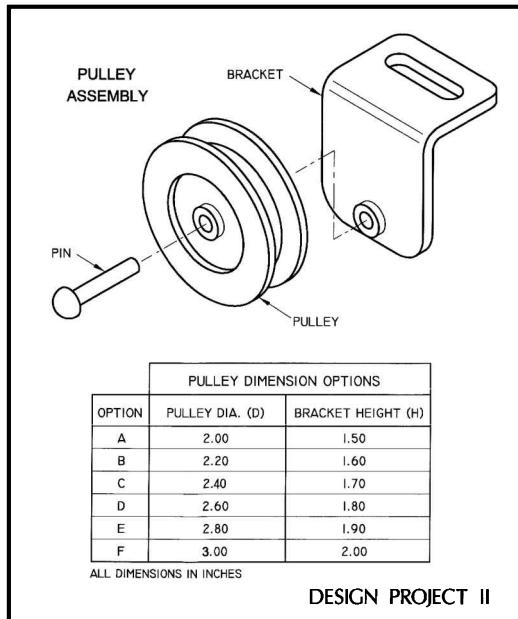
1. Create the Assembly indicated by your instructor following the table and make the corresponding Assembly Drawing with a Bill of Materials.



Supplementary Exercise 5-2: PULLEY ASSEMBLY

Create the parts of the Pulley Assembly of the Option specified by your instructor in the Table provided.

1. Create the Assembly indicated by your instructor following the table and make the corresponding Assembly Drawing with a Bill of Materials.



NOTES:

Design Workbook Lab 6: Part Evaluation and Configurations

In Engineering Graphics Lab 6, you will learn how to use the tools available to calculate a design's **Mass Properties**, and how to create a family of parts using design tables. One of the biggest advantages of a solid model is that you can accurately calculate mass properties without the need to fabricate a physical model, as well as digitally **Measure** every characteristic of your design. Additionally, you will learn how to generate multiple similar but different components using a **Design Table**, where you can control multiple design dimensions using an Excel spreadsheet.

THE MEASURE TOOL

The **Measure** tool can be found in the menu **Tools**, **Evaluate**, **Measure**, or from the Evaluate tab in the FeatureManager (see **Figure 6-1**). When you use the Measure tool, the cursor will change to a ruler; at this time you can select the entity (entities) to measure in the screen.

You can measure 2D sketch lines or 3D solid model geometry, including:

Sketch Line/3D model edge returns the length of the entity.

Sketch Arc/3D model circular edge returns the length of the arc and the diameter. If the arc is a full circle, the length is the circumference (see **Figure 6-2**).

3D Model Face returns the area and the total length of all model edges connected to the face.

Pair of Edges/Faces/Vertices returns the distance/angle/total length/total surface area of the selected items.

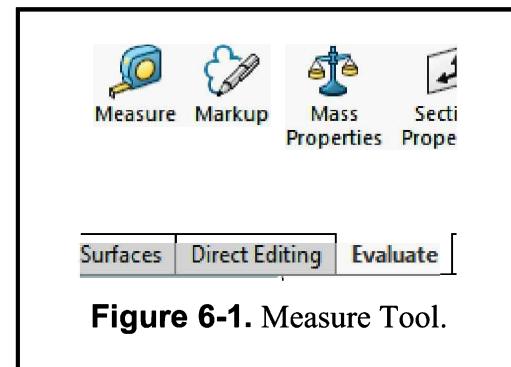


Figure 6-1. Measure Tool.

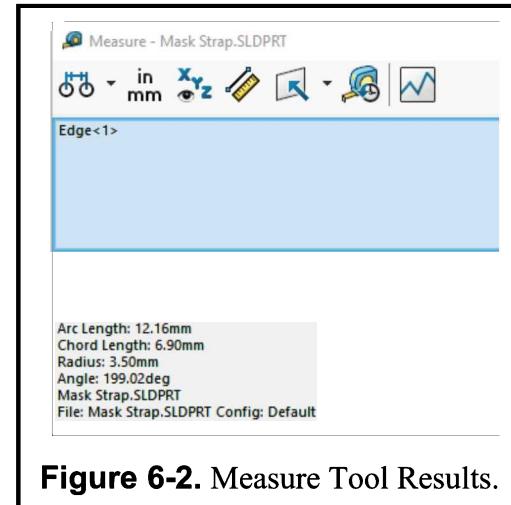


Figure 6-2. Measure Tool Results.

THE MASS PROPERTIES TOOL

Also located in the Evaluate tab or in the menu **Tools**, **Evaluate**, **Mass Properties**, this tool allows you to calculate the mass properties of a part or assembly.

Before you can calculate the mass properties accurately, you must set the part's material, including the density. **Table 6-1** includes the density of some common materials. Once the part's material is set, you can then select the **Mass Properties** command.

Design Workbook Using SOLIDWORKS 2020

The result will be reported in the “Mass Properties” Report, as shown in **Figure 6-3**. The mass properties and their units of measure reported include the following (see **Page 6-11** for definitions):

- **Part Name**
- **Density (lbs/in³)**
- **Mass (lbs)**
- **Volume (in³)**
- **Surface Area (in²)**
- **Center of Mass (in)**
- **Moments of Inertia (lbs*in²)**

After the component’s “Mass Properties” are calculated, you can review the results in this window. The options available include:

Print will open the print dialog to print the “Mass Properties” Report.

Copy will copy the entire report to the computer clipboard. Then you can paste this information in other applications.

Options will allow you to change the units of measure of the mass properties calculated, shown in **Figure 6-4**.

Recalculate will recalculate the mass properties after other options are changed.

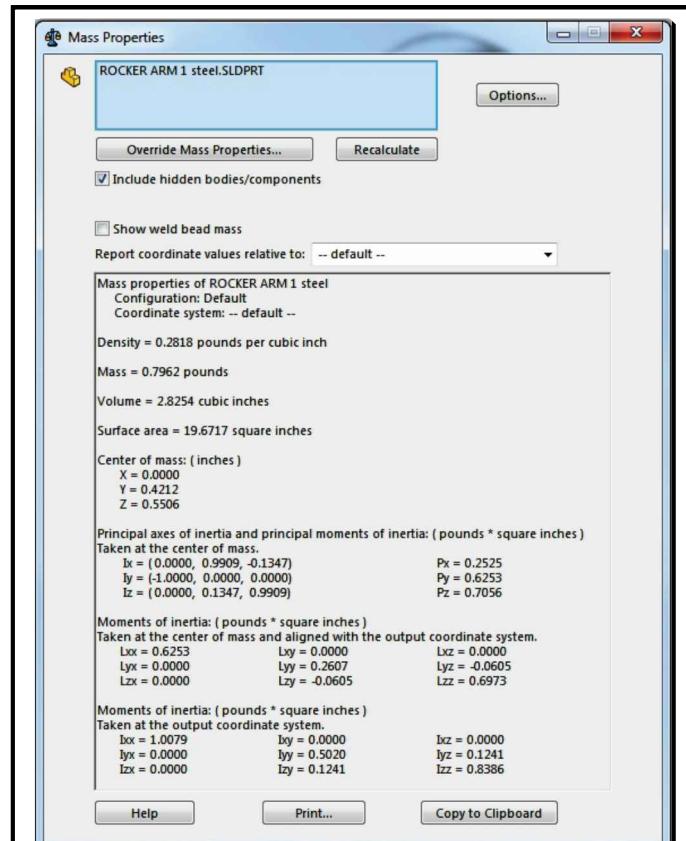


Figure 6-3. Mass Properties Report.

ABOUT THE MASS PROPERTIES UNITS

Some mass properties calculated are based on the model's geometry and are independent of the material's density (assumed to be uniform). Examples of these properties are volume and center of mass (centroid). Other properties, such as mass and moments of inertia, are dependent on the material's density. **Table 6-1** lists the density of some common engineering materials. You can use these density values to set the material properties of your part. Weight and Mass are often confused. Below is an example of the correct way to calculate them for a unit one-inch cube.

Note: The “Density” value used by SOLIDWORKS (see **Figure 6-4**) is actually the value of the “Unit Weight” listed below in **Table 6-1**. This lack of uniformity in terminology contributes to the general confusion on this matter.

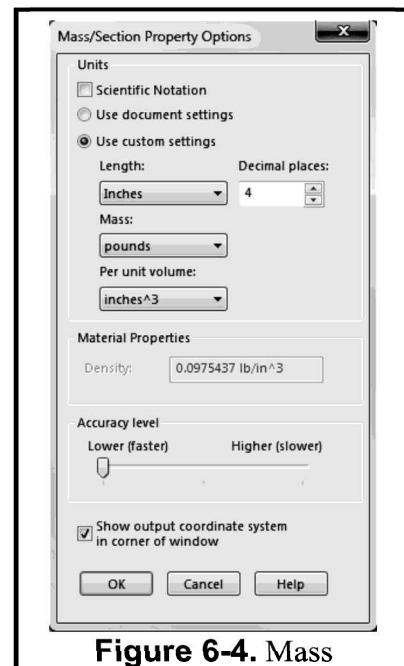


Figure 6-4. Mass Property Options.

Example Calculation: Material is Mild Steel (density is 0.728×10^{-3} lbs-sec 2 /in 4)

$$\text{Weight} = (\text{unit weight}) \times (\text{volume}) = (0.281 \text{ lbs/in}^3) \times (1.00 \text{ in}^3) = 0.281 \text{ lbs.}$$

$$\begin{aligned} \text{Mass} &= (\text{density}) \times (\text{volume}) = (0.728 \times 10^{-3} \text{ lbs-sec}^2/\text{in}^4) \times (1.00 \text{ in}^3) = 0.728 \times 10^{-3} \text{ lbs-sec}^2/\text{in.} \\ &= (\text{weight}) / (\text{gravity}) = (0.281 \text{ lbs}) / (386 \text{ in/sec}^2) = 0.728 \times 10^{-3} \text{ lbs-sec}^2/\text{in.} \end{aligned}$$

Table 6-1. Some Unit Weights and Densities of Common Materials.

Material	Unit Weight (lbs/in 3)	Density (lbs-sec 2 /in 4)
Aluminum	0.097	0.251×10^{-3}
Brass	0.307	0.794×10^{-3}
Chromium	0.259	0.671×10^{-3}
Copper	0.323	0.837×10^{-3}
Magnesium	0.063	0.163×10^{-3}
Plastic	0.036	0.093×10^{-3}
Rubber	0.041	0.106×10^{-3}
Steel	0.281	0.728×10^{-3}
Titanium	0.163	0.422×10^{-3}

Exercise 6.1: ROCKER ARM MASS PROPERTIES

For Exercise 6.1, you will build the rocker arm, which is designed to rotate about a principal axis. You will then copy the first rocker arm data into a second rocker arm file and make a change in its geometry. Mass Properties calculations will be performed on both models and the results will be compared to each other.

ROCKER ARM 1

Start a new part using the **ANSI-INCHES** template, add a sketch in the **Front Plane** and draw a circle with a diameter of **1.75"** centered in the origin. When complete **Extrude** it **1.25"** out from the **Front Plane**.

Next add a new sketch in the **Front Plane** and draw the profile shown in **Figure 6-5**. This will be extruded to create the Rocker Arm. Make sure you add the following relations, and dimension as indicated:

Both **Vertical lines** are **Tangent** to the **Arc**.

The **Circle** is **Concentric** to **Arc**.

The **Horizontal** line has a **Midpoint** relation to the origin.

Now **Extrude** the sketch **0.75"** in the same direction as the first extrusion. Your Rocker Arm model will look as **Figure 6-6**.

The last step to finish this model is to add a cutout for a keyway through the part. Change to a Front view, select the front face of the model, add a new sketch and draw the profile shown in **Figure 6-7**. Draw a circle and a rectangle, and then use the **Trim** tool.

 **Midpoint** Before adding the dimensions, add a vertical centerline from the origin up to the top horizontal line, and add a **Midpoint** relation between the horizontal line and the top endpoint of the centerline; this way the keyway will remain centered about the origin.

Add the necessary dimensions to fully define the sketch as shown in **Figure 6-8**, and make a **Cut Extrude** using the **Through All** end condition. Save your model as **Rocker Arm 1.sldprt** to continue.

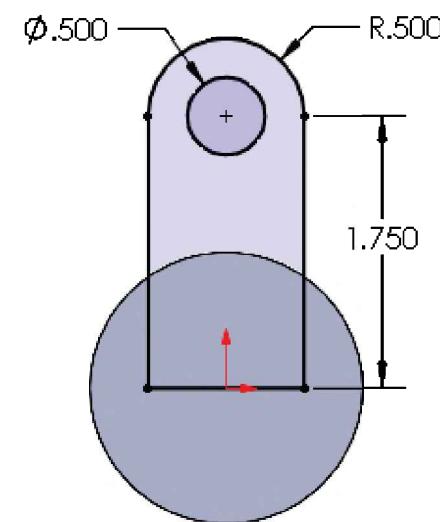


Figure 6-5. Dimensions for the Upright Part of the Rocker Arm.

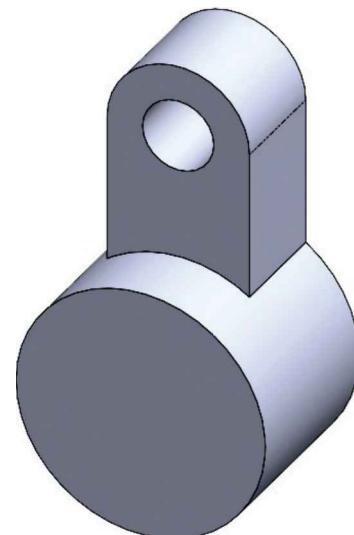


Figure 6-6.

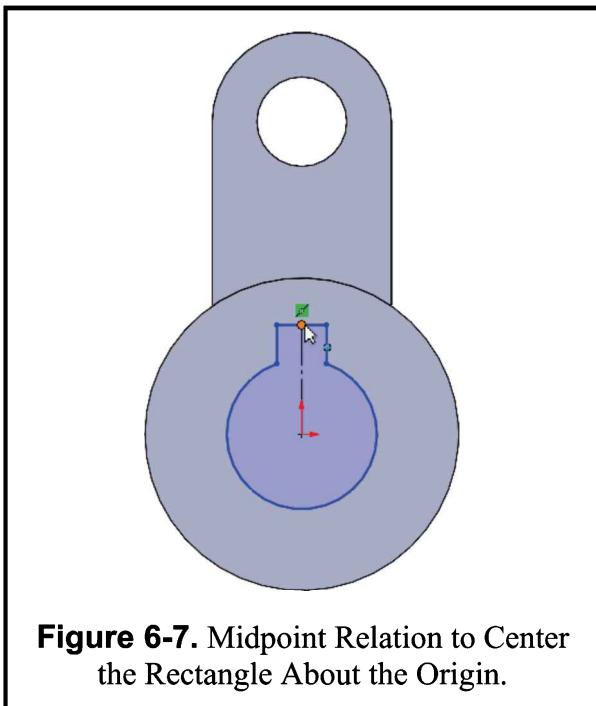


Figure 6-7. Midpoint Relation to Center the Rectangle About the Origin.

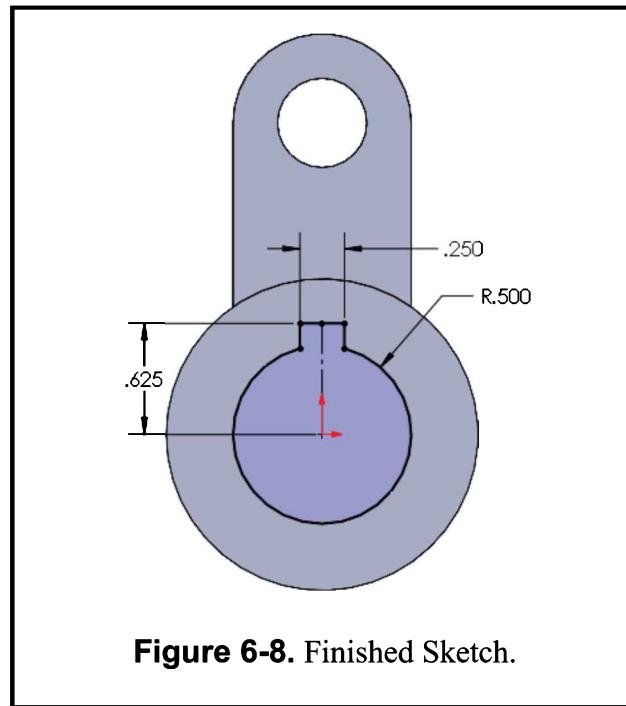
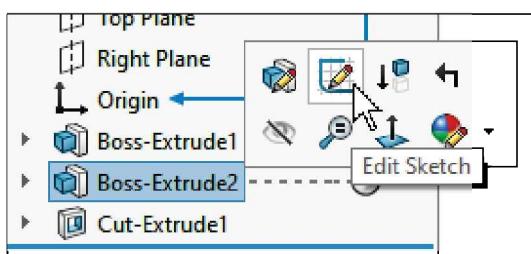


Figure 6-8. Finished Sketch.

ROCKER ARM 2

The **Rocker Arm 2** will be very similar to **Rocker Arm 1**. Use the **Save As** command and save the model as **Rocker Arm 2.sldprt**. After saving it select the **Boss-Extrude2** feature and click on the **Edit Sketch** command from the context toolbar.



While editing the sketch, double click to change the arc's radius to **0.625"** and the height to **1.50"**, as shown in **Figure 6-9**.



After making these dimensional changes, click on the **Rebuild** command to exit the sketch, and update the 3D model with the new dimensions.

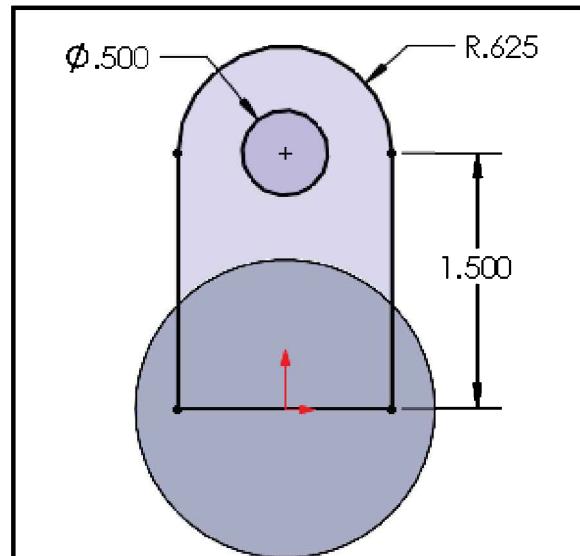


Figure 6-9. Changing the dimensions of the Rocker Arm.

Now you need to change the extrusion depth of the upright feature. Select the **Boss-Extrude2** feature in the FeatureManager tree. Select the **Edit Feature** option from the context toolbar, change the extrusion's depth to **0.500** inches and click **OK** to rebuild the model. The solid model will now be rebuilt with the new extrusion depth.

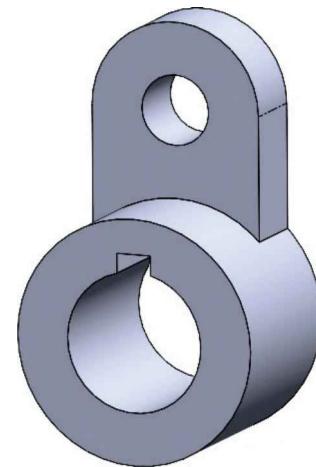
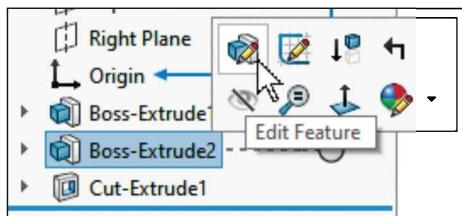


Figure 6-10. Finished Rocker Arm 2.

The completed **Rocker Arm 2** is shown in **Figure 6-10**. Save your model to continue.

MASS PROPERTIES ANALYSIS OF ROCKER ARM 1

To start your mass properties calculation, open the **Rocker Arm 1** part. After opening the file, you need to change the units to four decimal places. Select the menu **Tools, Options, Document Properties, Units**, change the **Mass/Section Properties** Length units to **4 decimal places** and click **OK** to continue.

Now you have to define the material the **Rocker Arm 1** will be made of. As you have done before, set the material to **Cast Carbon Steel** from the **Steel** library.

By assigning a material to your model, you are setting the correct physical properties of **Cast Carbon Steel** to our model, which will help us correctly calculate its mass properties.

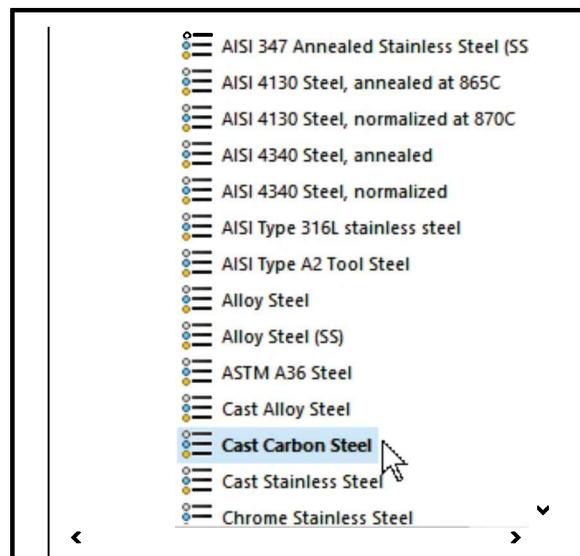
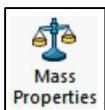


Figure 6-11.



Select the menu **Tools, Evaluate, Mass Properties**, or from the Evaluate tab click on **Mass Properties**. The "Mass Properties" window is now shown with all properties calculated for **Rocker Arm 1**, made of **Cast Carbon Steel**.

The Mass Properties report for **Cast Carbon Steel** will be displayed as shown in **Figure 6-12**.

The Mass Properties report includes the part's:

- Density (assigned by the material selection)
- Mass
- Volume
- Surface Area
- Center of Mass
- Principal axes of inertia
- Principal Moments of Inertia
- Moments of Inertia (taken at center of mass)
- Moments of Inertia (taken at output coordinate system)

Because of the design and functionality of a Rocker Arm, a critical design parameter is the part's resistance to rotation about the axis of rotation (Z-axis), in this case concentric with the large cylindrical hole.

Based on this criterion a part with a lower moment of inertia about the Z-axis would be a more desirable design because it would offer less resistance to rotation. To find the best option, you will evaluate both **Rocker Arm** designs using two different materials in each, to help you determine the optimum design.

From the Mass Properties report copy the full report using the **Copy to Clipboard** button and paste the information to a word processor. Personalize the report and add a title including the part's Name and Material, your Name, Seat number or assigned class ID. As an example:

**“Mass Properties of Rocker Arm 1.
Material: Cast Carbon Steel.
Student Name, Seat Number, ID.”**

After you have personalized the report, save it as **Rocker Arm 1-Cast Carbon Steel**.

Return to SOLIDWORKS, close the Mass Properties report, and change the part's material to **1060 Alloy** from the **Aluminum** material section.

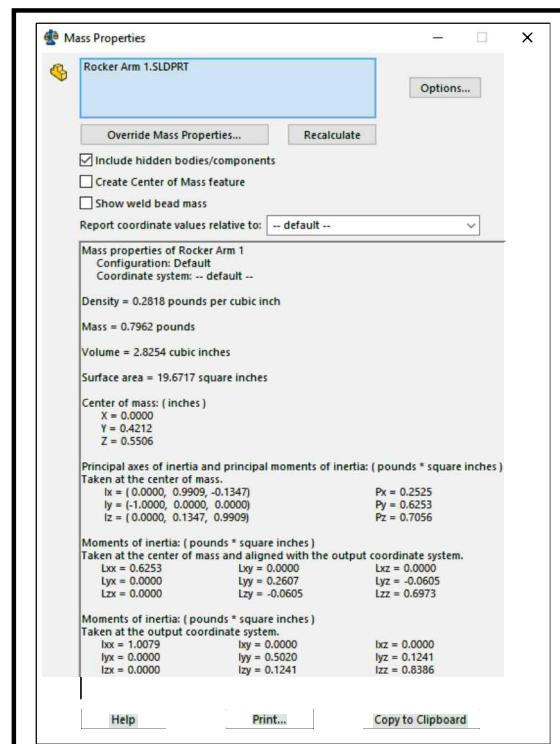


Figure 6-12. Mass Properties for Cast Carbon Steel Rocker Arm 1.

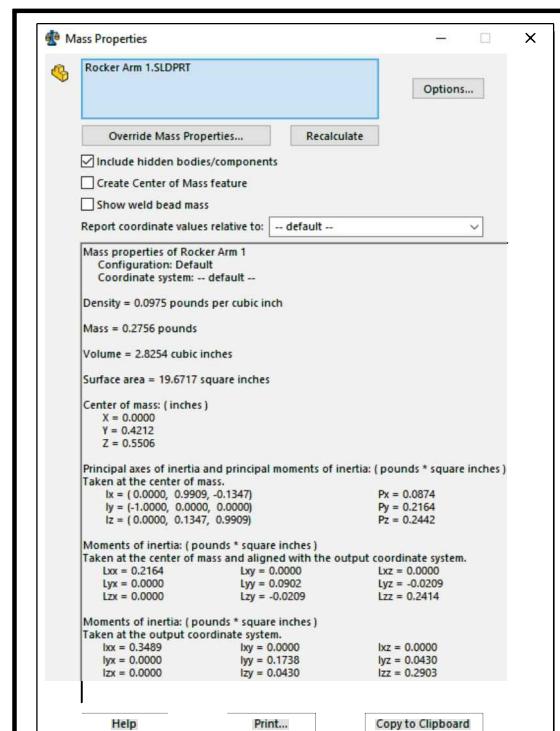


Figure 6-13. Mass Properties for Aluminum Rocker Arm 1.

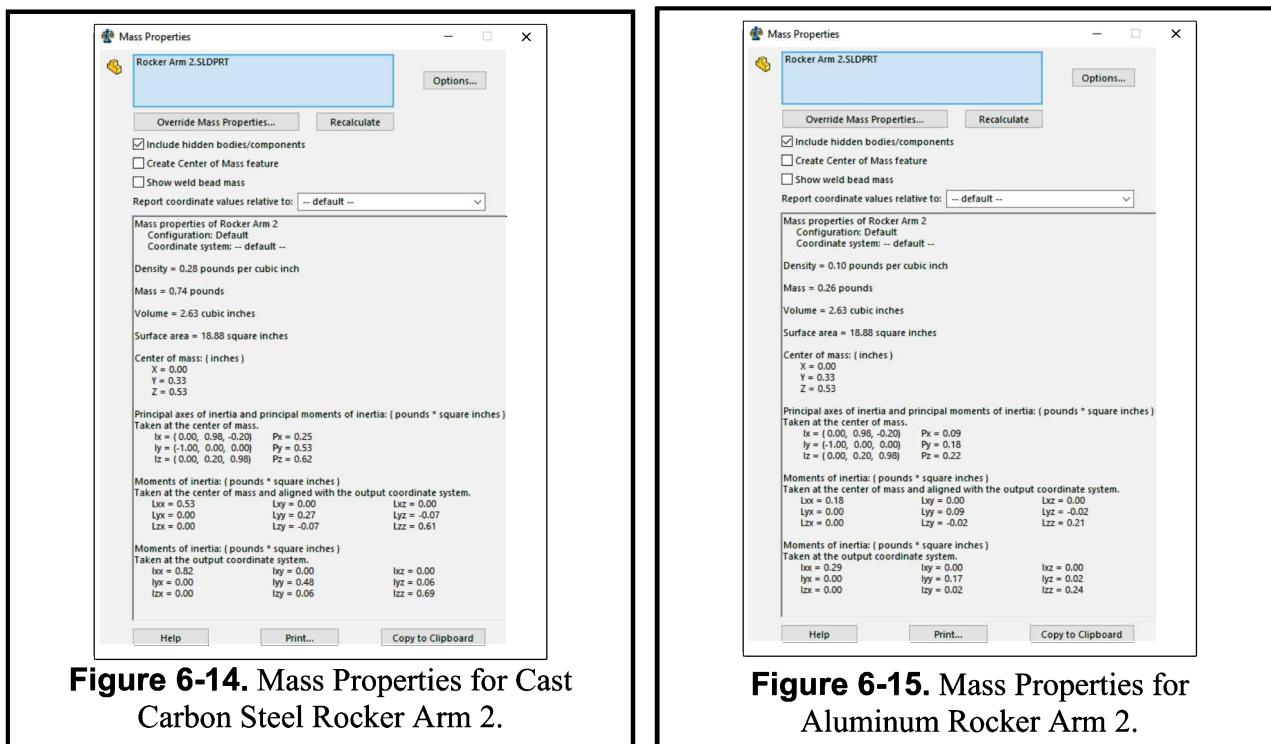
Design Workbook Using SOLIDWORKS 2020

Once you have assigned the Aluminum material properties, select the Evaluate tab again, and re-calculate the part's **Mass Properties**. The report for the **Aluminum 1060 Alloy** will be displayed as shown in **Figure 6-14**.

As you previously did for the **Cast Carbon Steel**, copy the **Aluminum** mass properties to a word processor, personalize it and save it as **Rocker Arm 1-Aluminum**.

Now you need to repeat the same process with the second design. Return to SOLIDWORKS and open the part **Rocker Arm 2**. Calculate the mass properties, copy the properties to your word processor, personalize your report and save it as **Rocker Arm 2-Cast Carbon Steel**.

Finally, change the **Rocker Arm 2** material to **Aluminum, 1060 Alloy**, calculate the mass properties and prepare the corresponding report. Save the last report as **Rocker Arm 2 - Aluminum** to finish. The mass properties for the **Rocker Arm 2** part with the **Cast Carbon Steel** and **Aluminum** are shown in **Figure 6-14** and **Figure 6-15**.



COMPARISON OF MASS PROPERTIES FOR ROCKER ARMS 1 and 2

After you have completed the mass properties analysis for both Rocker Arms 1 and 2 with different materials, you can evaluate your results and determine which is the best design and material.

TASK: Identify the report with the best combination of Geometry and Material that is easiest to rotate (requires the least torque) about the central Z-axis and turn it in to your lab instructor.

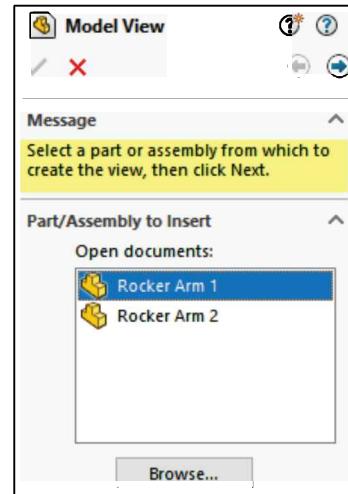
CREATE A DRAWING WITH BOTH ROCKER ARMS

Now you will make a new drawing including both **Rocker Arm** parts. One way is to make an assembly with both parts, and then make an assembly drawing as in the previous lesson. In order to learn additional functionality, the option you will use in this lab will cover adding an individual drawing view of each **Rocker Arm** to the drawing sheet.



Start by creating a new drawing using the **ANSI-Inches** template and click in the **Model View** command from the Drawing tab, or the menu **Insert, Drawing View, Model**.

- If the **Rocker Arm** parts were open in SOLIDWORKS at the time of making the new drawing, they will be listed under the **Open Documents:** section. Select the **Rocker Arm 1** part and click on the blue **NEXT** arrow at the top right of the command's options.
- If the parts were not open, click on the **Browse** button, locate the **Rocker Arm 1** part, click **Open** and then **NEXT** as in the previous option.



On the next page you can select the desired view orientation and options. For this exercise select the **Dimetric** view orientation, change the **Display Style** to **Shaded with Edges**, change the Scale to **3:2**, and if desired, turn On the **Preview** option. Click on the left side of the drawing sheet to locate the **Rocker Arm 1** view.

Repeat the previous steps to add the **Rocker Arm 2** drawing view on the right side of the drawing sheet, as seen in **Figure 6-16**.

From the Annotations tab select the **Note** command and add a note above each view with the part's name and scale. For the part's name use a **Bold Font, 20pt** high. For the scale's note use a **Regular Font, 15pt** high.

Arrange the views and notes in your drawing sheet, save the finished drawing as **Rocker Arms** and print a copy to submit to your lab instructor, along with the four mass properties reports.

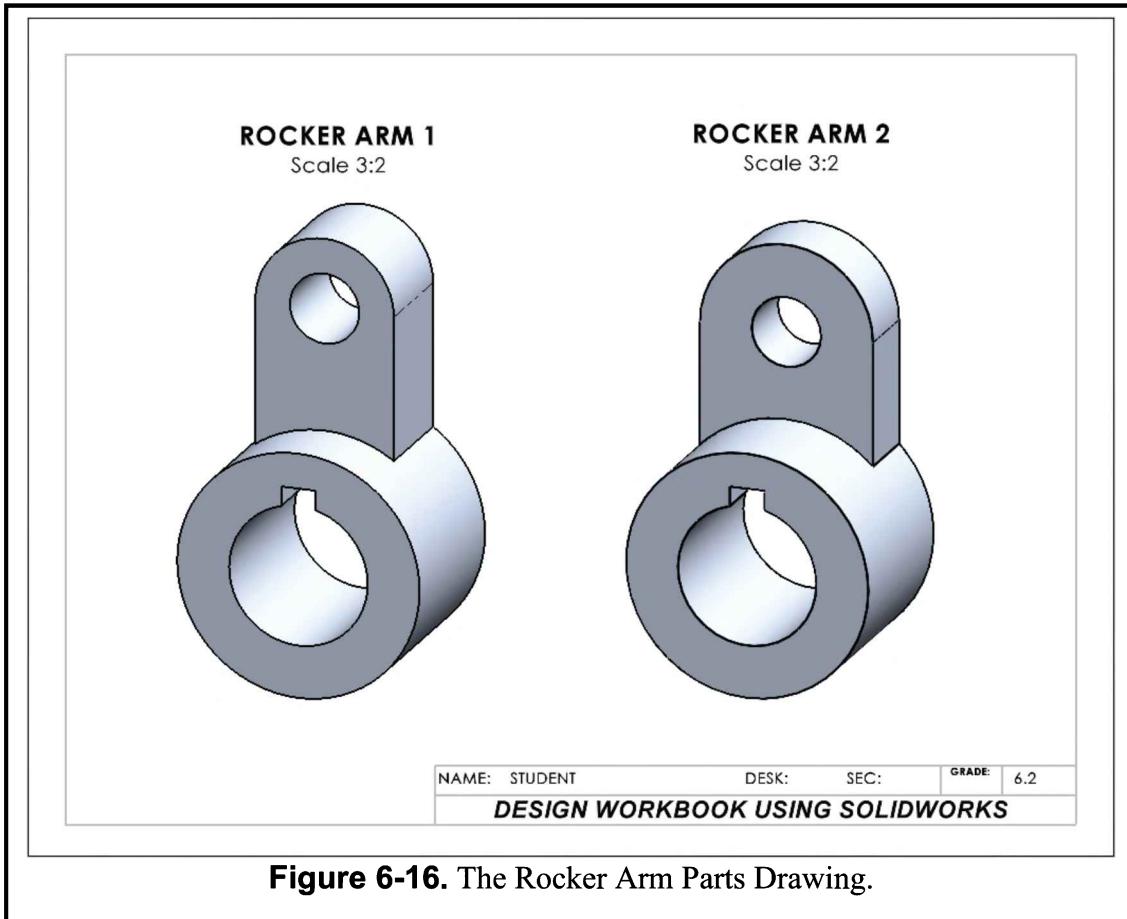


Figure 6-16. The Rocker Arm Parts Drawing.

INFORMATION PROVIDED IN SOLIDWORKS MASS PROPERTIES REPORTS

1. **DENSITY** is the mass or the weight per unit volume of the material the part is made from.
2. **MASS**: The mass of a body is the measure of its property to resist change in its steady motion. The mass depends on the volume of the body and the density of the material of which the body is made. In this case with SOLIDWORKS, mass is equivalent to weight.
3. **VOLUME**: The volume of a body is the total volume of space enclosed by its boundary surfaces.
4. **SURFACE AREA**: The surface area is the total area of the boundary surfaces defining the solid model.
5. **CENTER OF MASS**: Center of Mass (or Centroid) of a volume is the origin of coordinate axes for which first moments of the volume are zero. It is considered the center of a volume. For a homogeneous body in a parallel gravity field, mass center and center of gravity coincide with the centroid.
6. **PRINCIPAL AXES OF INERTIA AND PRINCIPAL MOMENTS OF INERTIA**: Principal moments of inertia are extreme (maximal, minimal) moments of inertia for a body. They are associated with principal axes of inertia which have their origin at the centroid and the direction of each usually given by the three unit-vector components. For these axes, the products of inertia are zero.
7. **MOMENTS OF INERTIA**: A moment of inertia is the second moment of mass of a body relative to an axis, usually X, Y, or Z. It is a measure of the body's property to resist change in its steady rotation about that axis. It depends on the body's mass and its distribution around the axis of interest.

Exercise 6.2: SOCKET DESIGN TABLE

In Exercise 6.2, you will make a family of parts for a Socket, using a Design Table. A Design Table allows you to create multiple configurations (or variations) of similar parts or assemblies by specifying parameters in an embedded Microsoft Excel worksheet. These parameters include model dimensions and features that will vary from one configuration to the next. Once a design table is added, you can select to the ConfigurationManager tab to review the different configurations, or variations. Also, these configurations can be used in assemblies and drawings.

Make a new part using the **ANSI-INCHES** template and save it as **Socket.sldprt**. Add a new sketch in the **Top Plane** and draw the profile in **Figure 6-17**. When you add dimensions, SOLIDWORKS automatically assigns them an internal name with the format:

DimensionName @ FeatureName

The **DimensionName** is the name of a dimension; the automatically assigned names have the format D1, D2, etc. for each sketch or feature. **FeatureName** is the name of the Feature or Sketch the dimension belongs to. For example, the **0.770"** dimension was the first dimension added to *Sketch1*; therefore, it will be named **"D1@Sketch1."**

A Design Table uses SOLIDWORKS' dimension names to assign a different dimension value for each configuration.

When you select a dimension, its name and other properties are displayed in the PropertyManager. Select the **0.770"** dimension to see its properties, as are shown in **Figure 6-18**. In the "Primary Value" section the dimension's name is listed as **D1@Sketch1** and shows the value of that dimension (**0.770 in**).

To make it easier to identify dimensions used in a Design Table, you have to rename them. With the **0.770 in** dimension selected, type **"Socket_Diameter@Sketch1"** in the name text box. The resulting dimension name is shown in **Figure 6-18**.

NOTE: You can enter only the part of the name before the "@" symbol; SOLIDWORKS will automatically fill in the rest of the name.

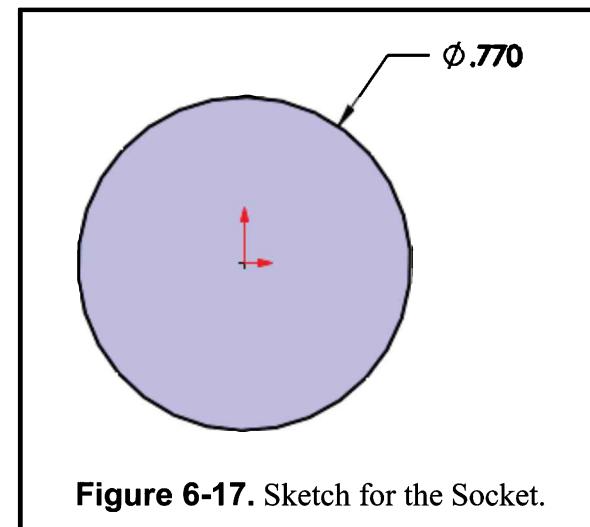


Figure 6-17. Sketch for the Socket.

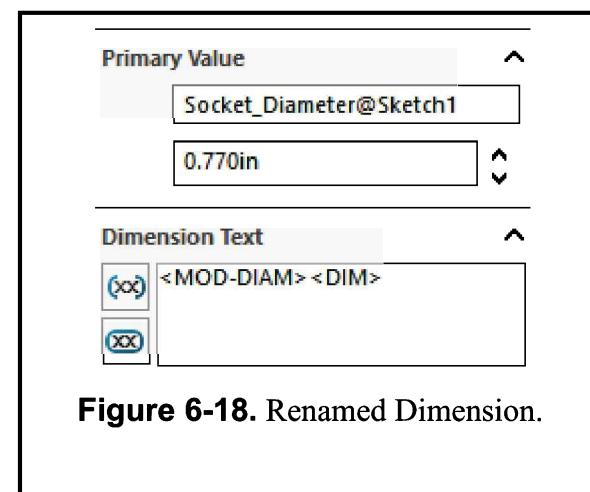


Figure 6-18. Renamed Dimension.

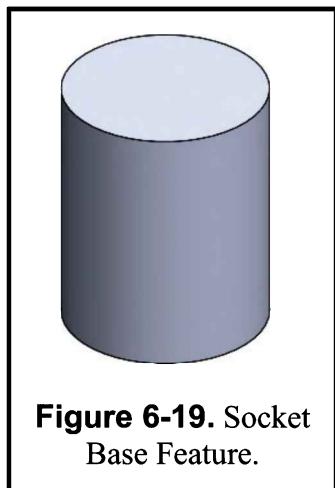
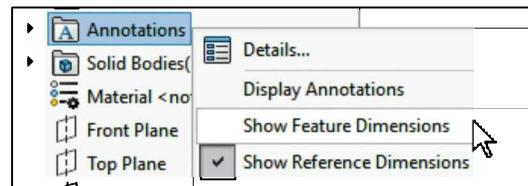


Figure 6-19. Socket Base Feature.

After renaming the dimension, extrude the sketch up **0.960"** to create the base feature as shown in **Figure 6-19**.

When the first extrusion is created, its depth is also assigned a dimension. To make the model dimensions visible you can right-click in the **Annotations** folder of the FeatureManager and turn on the **Show Feature Dimensions** option, making all model dimensions visible until the option is turned off.



Select the **0.960"** dimension and change its name to **"Socket_Height@Boss-Extrude1"** in the dimension's properties. Remember that **"Boss-Extrude1"** is the name of the first feature.

The result is shown in **Figure 6-20**. Right click in the **Annotations** folder and turn off the **Show Feature Dimensions** option.

Select the **Fillet** command and add a **0.03"** fillet to the top edge of the first feature as shown in **Figure 6-21**.

Now you will add the square hole used to attach the Socket to the ratchet. Change to a **Bottom** view orientation, add a new sketch in the bottom face and draw the profile shown in **Figure 6-22** using the **Center Rectangle** option. Add an **Equal** relation between a vertical and a horizontal line to fully define the sketch with a single dimension. When finished make an **Extruded Cut** of **0.450"** deep.

Change to a **Top** view, select the top face, and add a new sketch. Draw a **Hexagon** centered in the origin using the **Inscribed Circle** option and dimension the diameter **0.566"**. Select a hexagon line and add a **Horizontal** relation to fully define your sketch. Just as you did with the other dimensions, rename the diameter dimension as **Socket_Size@Sketch3**.

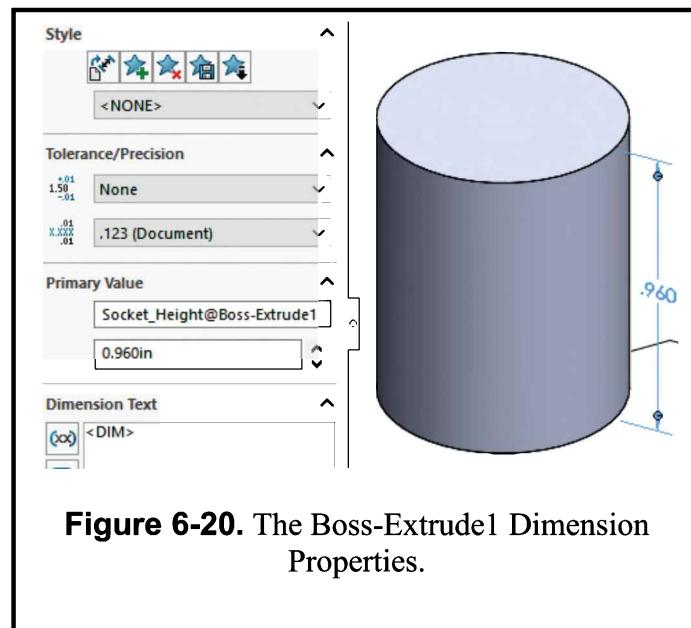


Figure 6-20. The Boss-Extrude1 Dimension Properties.

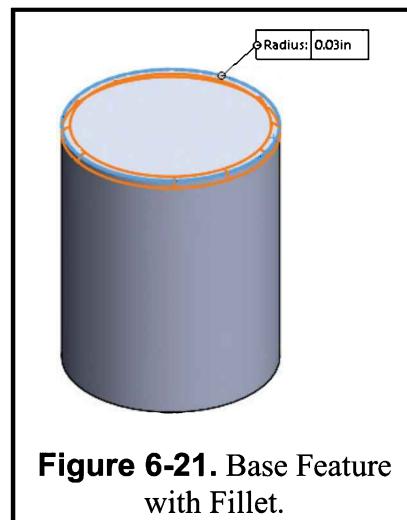


Figure 6-21. Base Feature with Fillet.

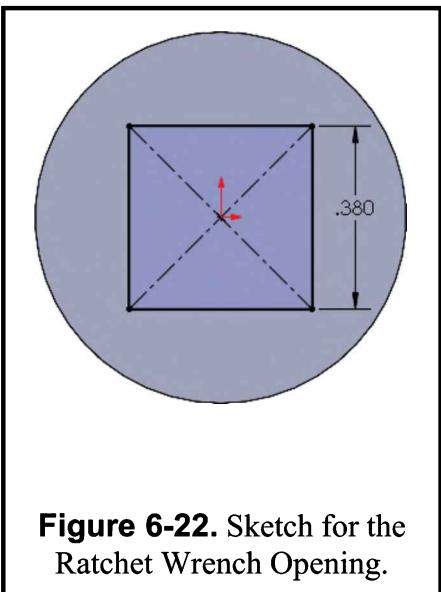


Figure 6-22. Sketch for the Ratchet Wrench Opening.

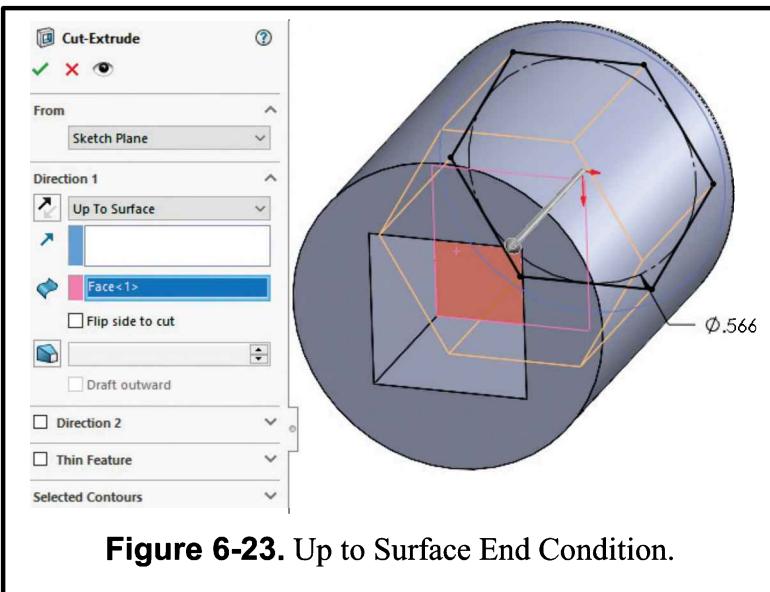
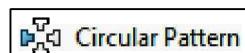


Figure 6-23. Up to Surface End Condition.

With the sketch finished make an **Extruded Cut** using the **Up To Surface** end condition and select the top face of the square hole on the bottom of the socket, as shown in **Figure 6-24**.



Figure 6-24. Finished Socket Opening.



To complete the socket opening, select the **Circular Pattern** command from the Features tab, select a circular edge of the first extrusion as the direction, and add the hexagonal cut in the **Features to Pattern** selection box.

Use the **Instance Spacing** option, enter **30°** and **2** copies. You will have to use the **Geometry Pattern** option at the bottom. This option will make an exact geometric copy of the hexagonal copy, without calculating the **Up to Surface** end condition of the original hexagonal cut. The socket with the **Circular Pattern** will look like **Figure 6-24**.

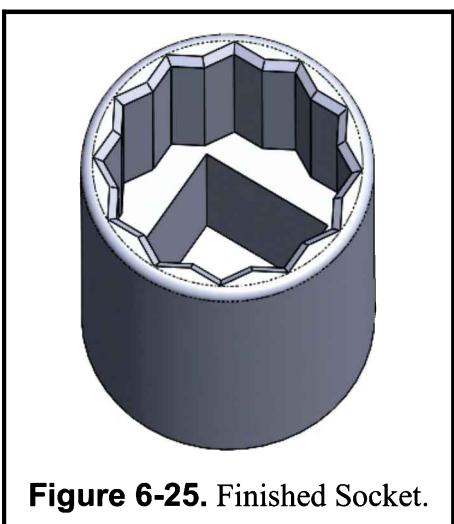
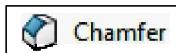


Figure 6-25. Finished Socket.



The last step in the construction of the socket is to add a **Chamfer** to all the short edges of the polygon on the top face of the Socket. Set the value of the **Chamfer** to **0.023"**. The finished socket should look like **Figure 6-25**. Save your model as **SOCKET.sldprt**.

With your part complete you are now ready to add a **Design Table** to create additional variations of the socket, called **Configurations**. Select the menu **Insert, Tables, Design Table**, use the **Auto-Create** option, and click **OK** to continue. In the **Dimensions** window list, you will see all the dimensions available in the model. This is why renaming the dimensions is a good idea.

Hold down the **Ctrl** key, select the following dimensions to add them to the **Design Table** and click **OK** to continue:

- **Socket_Diameter@Sketch1**
- **Socket_Height@Boss-Extrude1**
- **Socket_Size@Sketch3**

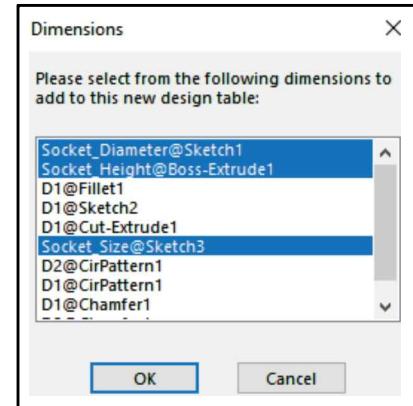
NOTE: If you attempted to add a sketch or create a feature multiple times, the **Sketch** or **Feature** name counters *may* have a different number from the list above. If this is the case, there should be no problem; the dimension names will match your **Dimensions** list.

Immediately an Excel spreadsheet is added to the SOLIDWORKS part, as seen in **Figure 6-26**. The pre-selected dimensions have been added to the **Design Table** on Row 2, starting at Column B.

To add more configurations to the design table, you must enter the configuration names on Column A, starting at Row 3 going down, and fill the table with the corresponding dimension values for each configuration. Fill the Design Table as shown in **Figure 6-26**.

NOTE: Do not click outside of the spreadsheet until you are finished entering information, otherwise, the Design Table is considered complete and you will exit Excel and return to SOLIDWORKS.

NOTE: If you need to modify your design table after completing it (or if you accidentally exited before finishing it), select the **Configuration Manager** tab at the top of the FeatureManager (seen in **Figure 6-27**), expand the Tables folder, right click on **Design Table** and select **Edit Table**. When you finish entering the table information, click on the



The screenshot shows an Excel spreadsheet titled '1 Design Table for: Socket'. The table has columns labeled 'A', 'B', 'C', 'D', 'E', and 'F'. Row 1 contains the header labels: 'A' (empty), 'B' (empty), 'C' (empty), 'D' (empty), 'E' (empty), and 'F' (empty). Row 2 contains the first data row with values: 'B' (empty), 'C' (empty), 'D' (empty), 'E' (empty), 'F' (empty). Rows 3 through 8 contain data for different socket configurations:

	Socket_Diameter@Sketch1	Socket_Height@Boss-Extrude1	Socket_Size@Sketch3
3	0.770	0.960	0.566
4	0.864	1.005	0.629
5	0.948	1.018	0.692
6	1.004	1.132	0.755
7	1.084	1.120	0.818
8	1.168	1.255	0.880
9			
10			

The cell containing the value '0.880' in the 'F' column of Row 8 is highlighted with a green border. The bottom of the spreadsheet shows the 'Sheet1' tab and standard Excel ribbon buttons.

Figure 6-26. Socket Design Table.

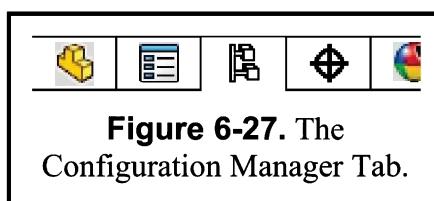


Figure 6-27. The Configuration Manager Tab.

SOLIDWORKS graphics area to complete the Design Table and create the different configurations. The new part configurations are generated and listed as shown in **Figure 6-28**.

To review the configurations created select the Configuration Manager shown in **Figure 6-29**. The configuration name with a green check mark is the currently active configuration.

To activate a different configuration right click in a configuration's name and select Show Configuration from the context toolbar or double-click on it. Activate the different configurations to see their differences.

Notice the configurations driven by the **Excel** table are preceded by an **Excel** icon, and the **Default** configuration is not. Also, the **Default** configuration is the same as "**9-16 Socket**" because the values entered in row 3 are the same as your original model dimensions.

Since the "**9-16 Socket**" and the **Default** configuration have the same dimensions, right click on the **Default** configuration, and delete it. You cannot delete the active configuration, activate a different one and then delete the **Default** configuration.

To document the different Socket configurations, you can make a new assembly and add the Socket part multiple times using a different configuration for each instance.

Select the **New** command and create a new **Assembly**. When the **Begin Assembly** command is displayed, click in the **Keep Visible** pin next to the **Cancel** button.

Select the **Socket** part from the **Open Documents**: list, select a configuration from the **Configuration**: drop-down list and click in the assembly to add it. Select a different configuration, add it to the assembly and continue adding all configurations approximately as shown in **Figure 6-30**. Finish the **Begin Assembly** command and save the assembly as **Socket Configurations.sldasm**.

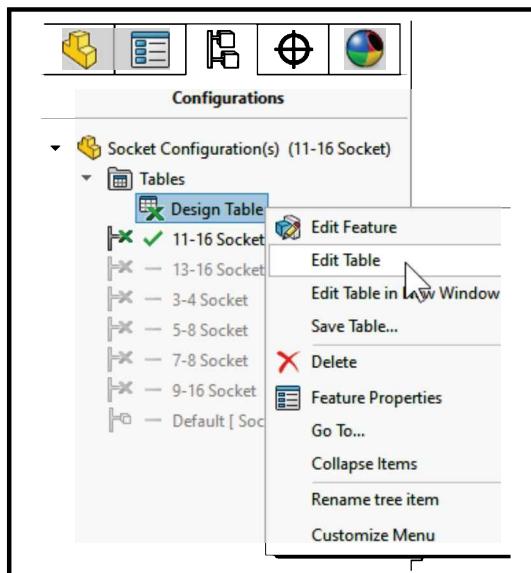


Figure 6-28. Edit Design Table.

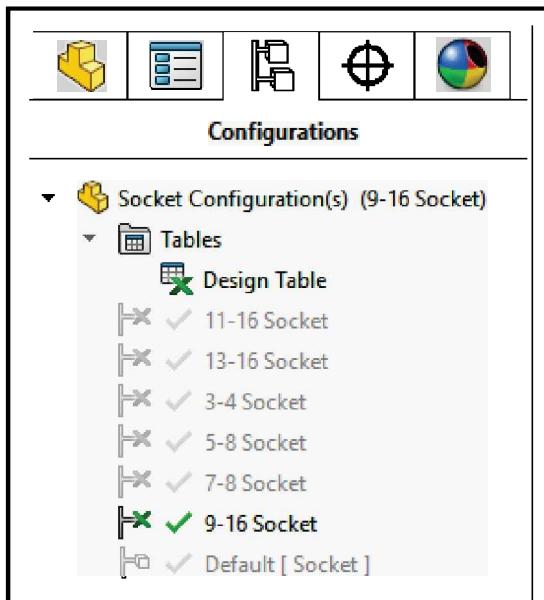
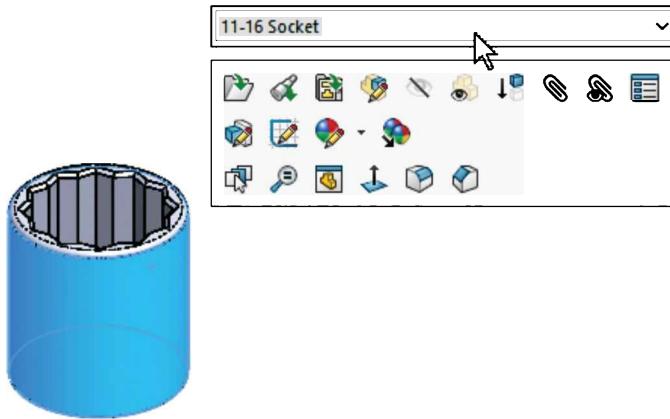


Figure 6-29. Different Socket Configurations.

NOTE: In an assembly, if you select a part with configurations, the context toolbar will include a drop-down list from which you can select a different configuration.



Make a new drawing using the **Inches** template and add a shaded **Trimetric** view. Add a note next to each configuration indicating their name as shown in **Figure 6-30**. Save the drawing as **Socket Configurations.slddrw** and print a copy for your lab instructor to finish.

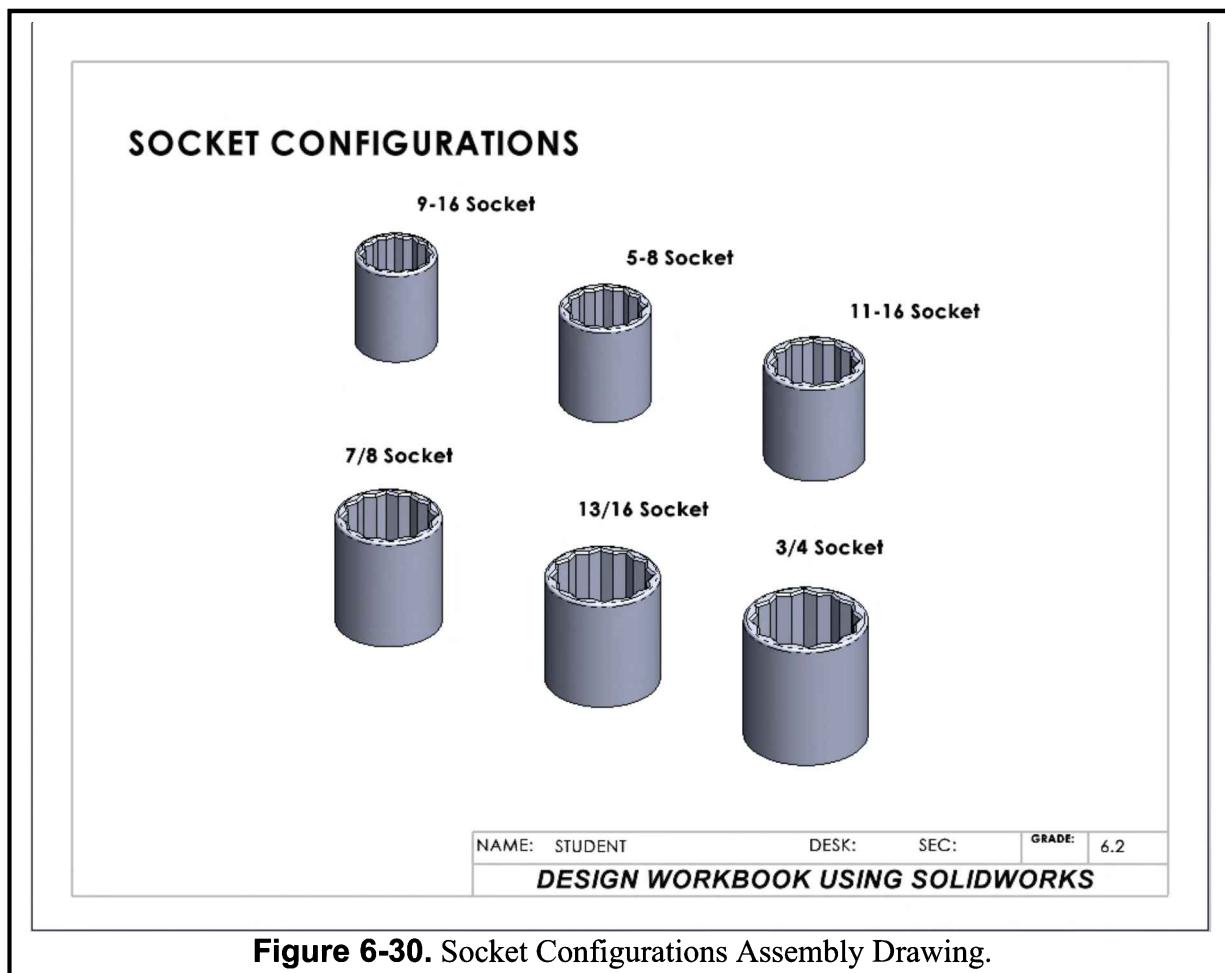
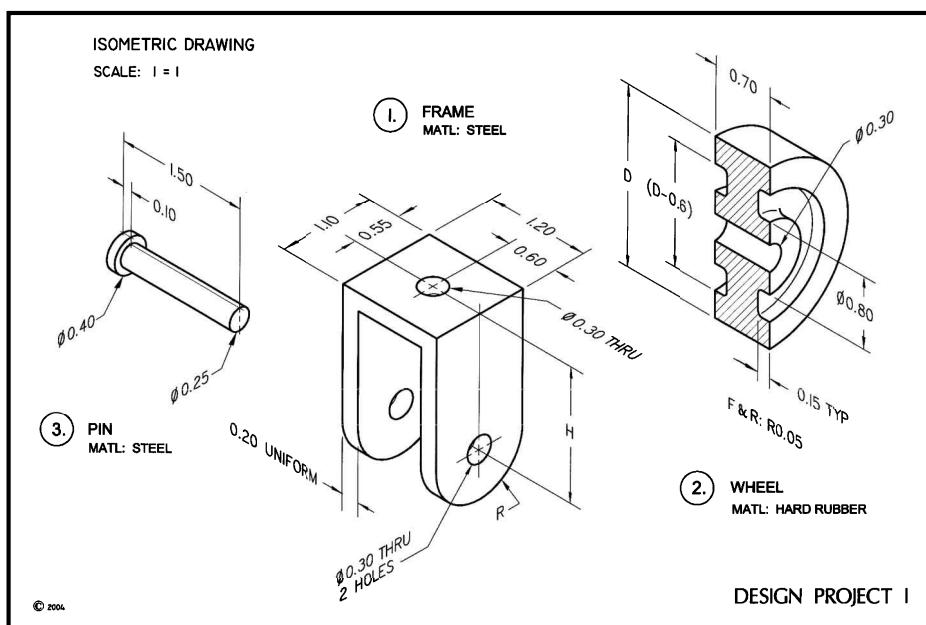
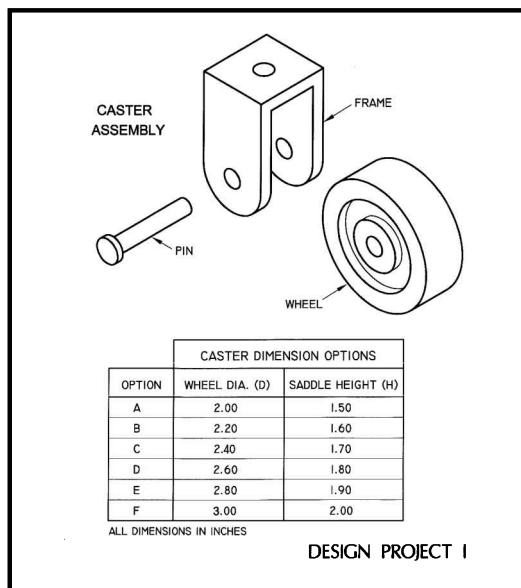


Figure 6-30. Socket Configurations Assembly Drawing.

Supplementary Exercise 6-1: CASTER AND CASTER FRAME CONFIGURATIONS

Create new parts with Design Tables to generate the different configurations needed for the Caster Frames and Caster Wheels specified in the Table provided. See your instructor for specific instructions.

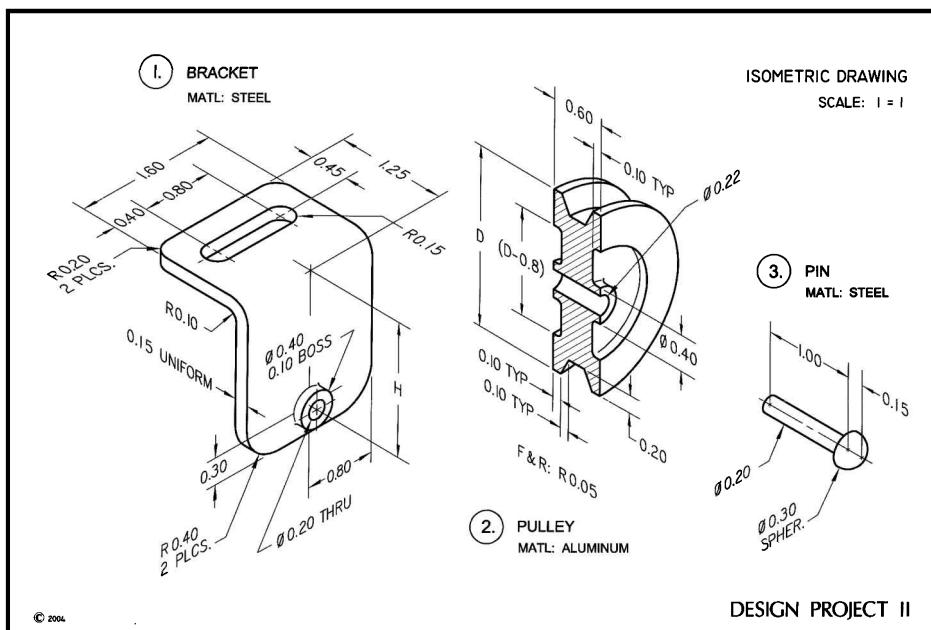
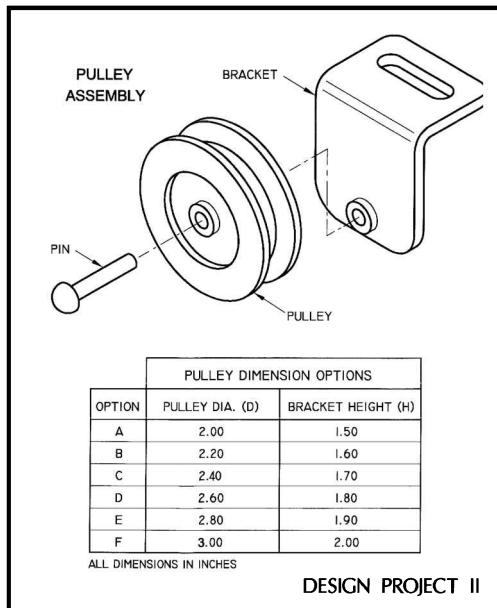
1. Generate an assembly drawing of the six **Caster Frames** generated with the Design Table. Use **Figure 6-30** as an example.
2. Generate an assembly drawing of the six **Caster Wheels** generated with the Design Table Use **Figure 6-30** as an example.



Supplementary Exercise 6-2: PULLEY AND PULLEY BRACKET CONFIGURATIONS

Create new parts with Design Tables to generate the configurations needed for the Caster Frames and Caster Wheels specified in the Table provided. See your instructor for specific instructions.

1. Generate an assembly drawing of the six **Pulley Brackets** generated with the Design Table. Use **Figure 6-30** as an example.
2. Generate an assembly drawing of the six **Pulleys** generated with the Design Table. Use **Figure 6-30** as an example.



NOTES:

Design Workbook Lab 7: Static Stress and Thermal Analysis

INTRODUCTION

SOLIDWORKS allows the designer to build and modify 3D solid models easily by editing its dimensions and constraints. In this lab exercise, you will build a concept for a pillow block. This model will then be analyzed using the **SOLIDWORKS Simulation** Finite Element Analysis (FEA) add-in software to find the stress distribution in the part under normal use conditions, supporting a rotating shaft.

Based on the stress analysis results, you will then make changes to the original design's geometry and/or material to improve the part's performance. After the changes are made, you will make a second stress analysis to evaluate if the changes improved the part's performance.

Finite Element Analysis using SOLIDWORKS Simulation

The purpose of performing a Finite Element Analysis early in the design process is to evaluate if your design will be capable of performing the task it was designed to do. A simulation will virtually test your design when exposed to the conditions anticipated under normal use, including external forces, pressure, temperature, vibrations, etc. Once you have the simulation results, you can make an informed decision about how to modify your design, re-run the analysis, and modify it again, until the results are satisfactory.

A huge advantage of FEA analysis is that you can significantly reduce prototype and physical testing costs.

The object of this exercise is to build a Pillow Block and run a simulation using the **SOLIDWORKS Simulation** under the anticipated loads expected.

Before you begin this exercise, make sure the simulation software is installed as an "Add In" to SOLIDWORKS. To load the "Add-In" select the menu **Tools, Add-Ins**, and activate the **SOLIDWORKS Simulation** package, as shown in **Figure 7-1**.

After the Add-In is loaded a new Simulation tab is added to the CommandManager, and a new toolbar is available in the menu **View, Toolbars**.

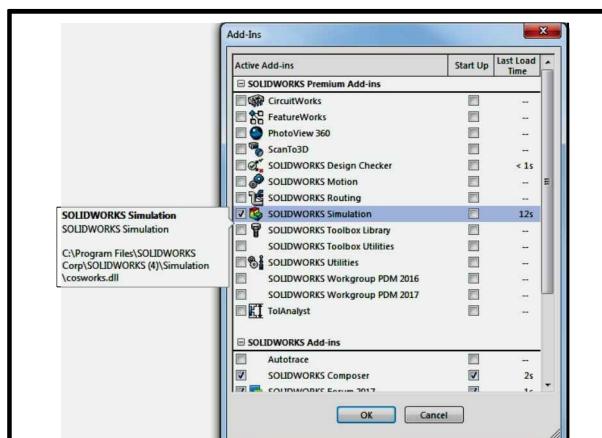


Figure 7-1. SOLIDWORKS Simulation Add-In.

The Simulation toolbar shown in **Figure 7-2** includes the list of external loads you can add to your design, to approximate the actual conditions your design will be exposed to during normal operation. A description of each of load commands is given in **Table 7-1**.

Table 1. Definitions

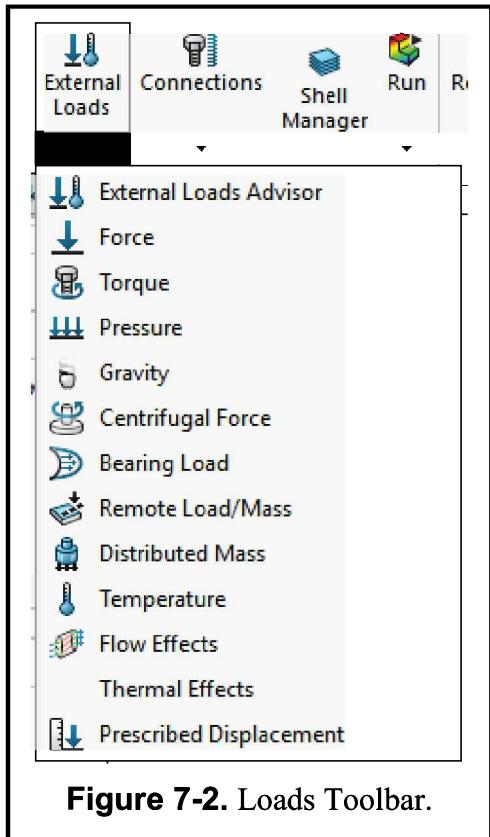


Figure 7-2. Loads Toolbar.

Restraints	Applies restraints to the selected entities for the current structural study to prevent the model from moving under the loads (static, frequency, or buckling).
Pressure	Applies a pressure to the selected faces for the current structural study (static, frequency, or buckling).
Force	Applies a force, torque, or moment to the selected entities in the current structural study (static, frequency, or buckling).
Gravity	Defines gravity loading for the current structural study (static, frequency, or buckling).
Centripetal Force	Applies centripetal forces for the current structural study (static, frequency, or buckling).
Remote Load	Applies a remote load, used for structural studies.
Rigid Connection	Simulates a rigid connection between selected faces in structural studies.
Bearing Load	Applies bearing loads on selected faces of different components.
Temperature	Applies temperatures on the selected entities for the current thermal or structural (static, frequency, or buckling) studies.

Exercise 7.1: FINITE ELEMENT ANALYSIS OF PILLOW BLOCK

DESIGNING THE FIRST VERSION OF THE PILLOW BLOCK

The first step is to build the model of the Pillow Block. Start a new part using the **ANSI-INCHES** template, add a new sketch in the **Front Plane** and draw the profile shown in **Figure 7-3**.

Make a **1.40"** extrusion using the **Mid Plane** end condition and click OK to continue.

Now you will make the mounting holes for the pillow block.

Change to a **Top** view, add a sketch on either flat face as shown in **Figure 7-4** and make an **Extruded Cut** using the **Through All** end condition.

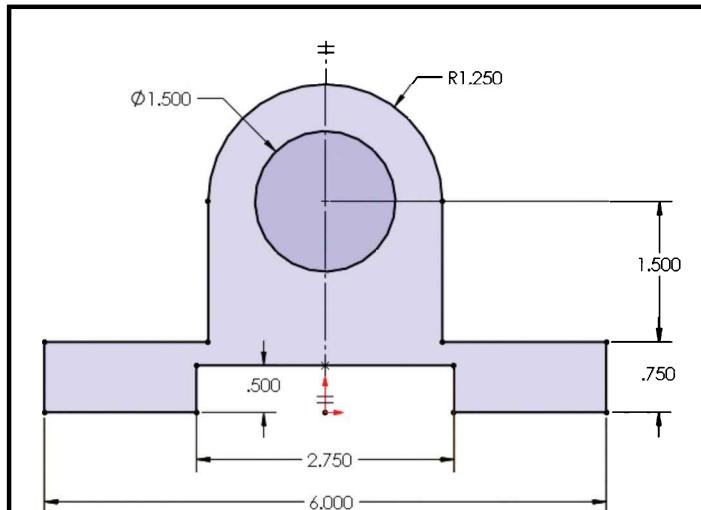
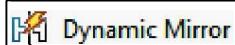


Figure 7-3. Initial Sketch for Pillow Block.



You can use the **Dynamic Mirror** about the vertical **Centerline** to make the sketch symmetric and make the center of either hole **Horizontal** with the origin to fully define your sketch.

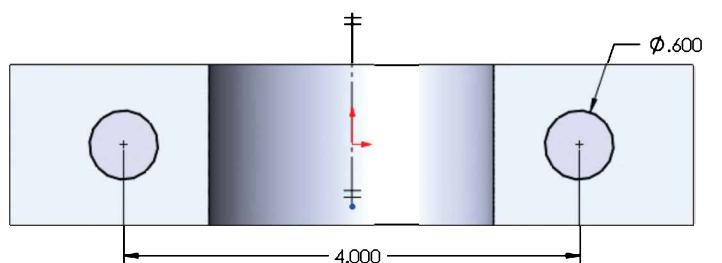


Figure 7-4. Sketch for Mounting Holes.

When finished, your model should look like **Figure 7-5**. Save your part as **PILLOW1.sldprt**.

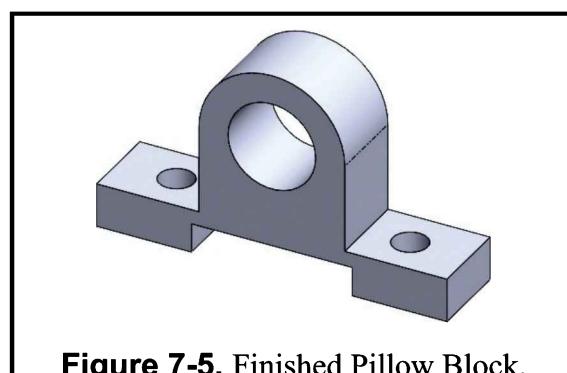
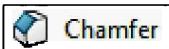


Figure 7-5. Finished Pillow Block.

Now you will build a shaft that fits into the large hole of the Pillow Block. Make a new part using the **ANSI-INCHES** template.

Add a new sketch in the **Front Plane** and draw the profile shown in **Figure 7-6** and make a **7.00"** extrusion using the **Mid Plane** end condition.



When the first extrusion is complete, add a **Chamfer** to both ends using the **Angle-Distance** option. Set the Chamfer parameters to **0.125" x 45°**, as shown in **Figure 7-7**.

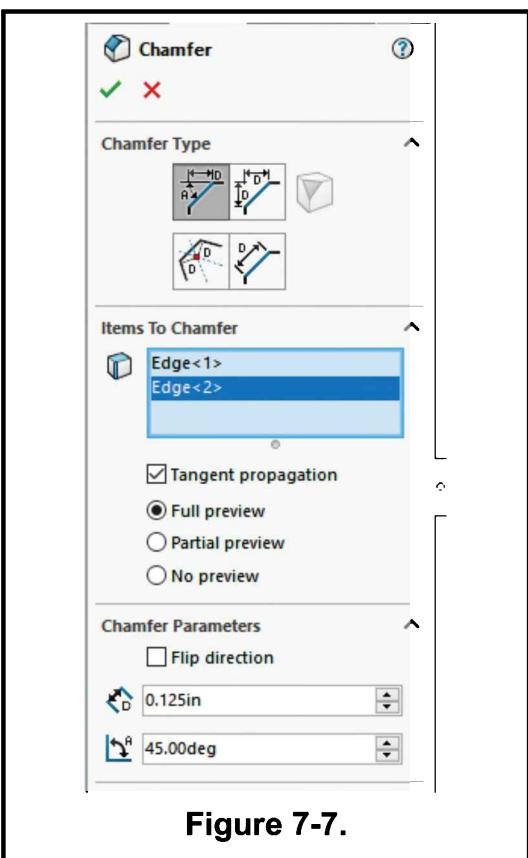


Figure 7-7.

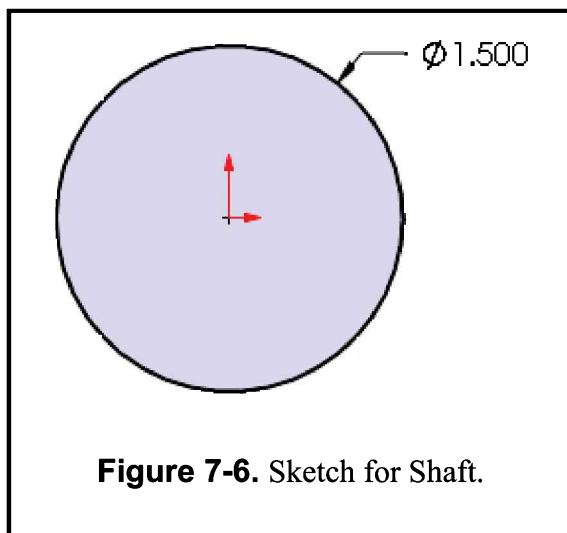


Figure 7-6. Sketch for Shaft.

The completed shaft should look like Figure 7-8. To finish save your part as **SHAFT.sldprt**.

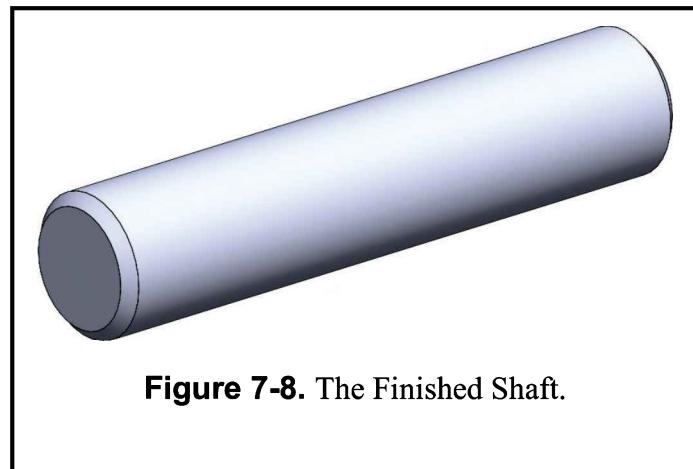


Figure 7-8. The Finished Shaft.

MAKE THE PILLOW BLOCK ASSEMBLY

Before you run the stress analysis, you need to make an assembly using the **Pillow1** and the **Shaft**.

Start a new assembly and add the **Pillow1** part first. Remember to just click **OK** in the **Begin Assembly** command to align it with the assembly's default planes.

Next add the **Shaft** and locate it next to **Pillow1** as shown in **Figure 7-9**. Using the **Mate** command, add a **Concentric** mate between the shaft and the inside face of the **Pillow1**.

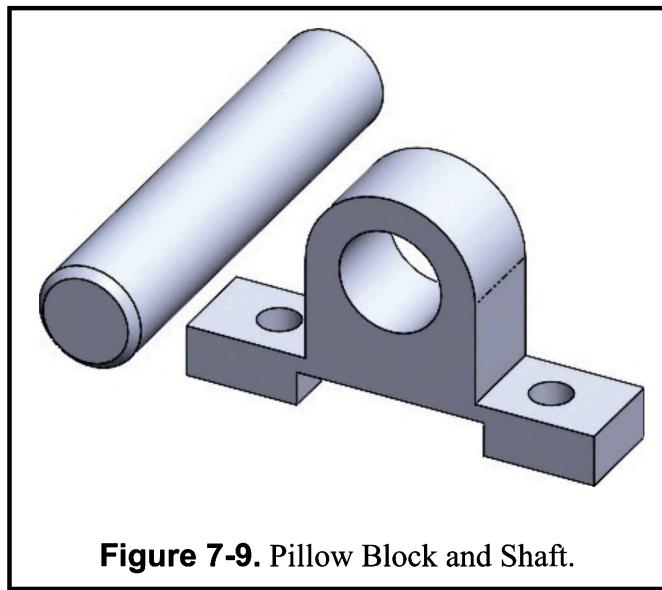


Figure 7-9. Pillow Block and Shaft.

TIP: If you hold down the **Ctrl** key and select the two faces to mate, after releasing the **Ctrl** key you will see the option to add the **Concentric** mate in the context toolbar.

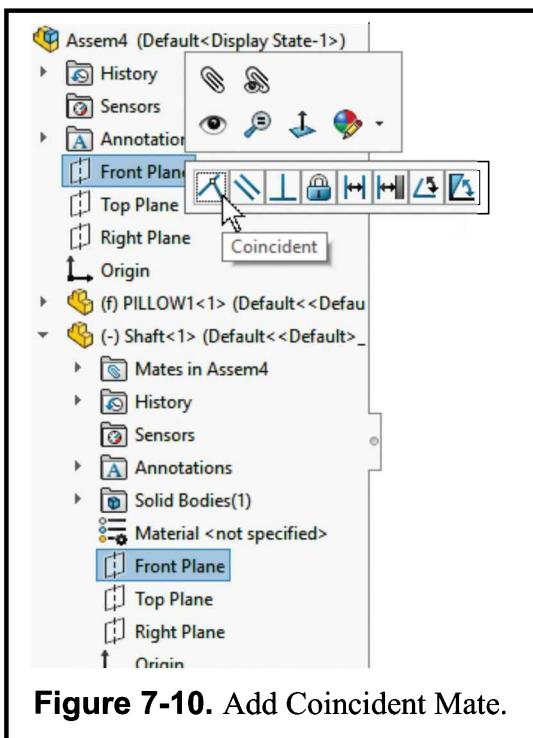
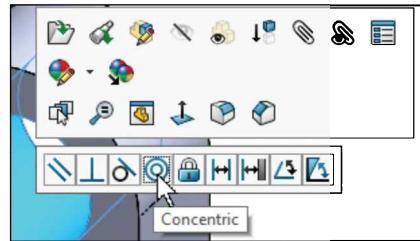


Figure 7-10. Add Coincident Mate.



In the FeatureManager expand the **Shaft**, hold down the **Ctrl** key and select the **Front Plane** and the assembly's **Front Plane**.

Release the **Ctrl** key and click in the **Coincident** mate from the context toolbar, as shown in **Figure 7-10**. This way, the **Shaft** will be centered in the **Pillow1**.

After the parts are mated, save your assembly as **Pillow Assembly-1.sldasm**.

FINITE ELEMENT ANALYSIS USING SOLIDWORKS SIMULATION

A Finite Element Analysis is a complex process that *approximates* a physical phenomenon, in this case a shaft rotating in a pillow block, using a mathematical process to determine if the design will meet your design specifications or not, in other words, if **the design fails or not**.

An analysis can take in consideration many different physical conditions like force, pressure, temperature, heat, vibration, etc. In your Pillow Block design, you will only make a static analysis with a vertical force on the shaft.

Word of advice: It should be noted that just because an analysis *reports* the design is safe, does not necessarily mean it will *work* as intended. An analysis only considers the external conditions entered, and nothing else. In other words, if the simulation parameters are incorrect or do not represent the actual physical conditions accurately, the results will likely be incorrect. This could be caused by an incorrect or missing force, pressure, temperature, vibration, material, speed, etc. This is the reason why an analysis needs to be performed by a qualified person who understands the conditions to be simulated and is trained to interpret the results obtained.

If the Simulation Tab does not appear in the CommandManager, select the menu **Tools, Add-ins**, select **SOLIDWORKS Simulation** and click **OK**. This will add a Simulation menu and tab.

In order to make an analysis you need to define a new study. The minimum information required to run a static stress analysis is to define how the part is supported (**Fixtures**), what external forces and/or pressures are acting on the part/assembly (**External Forces**), what the component(s) are made of (**Material**), and in the case of an assembly, how the parts are connected to each other (**Connections**).

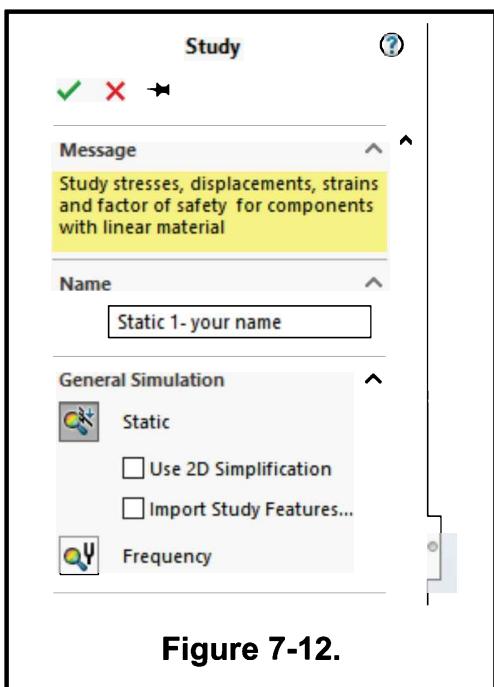


Figure 7-12.

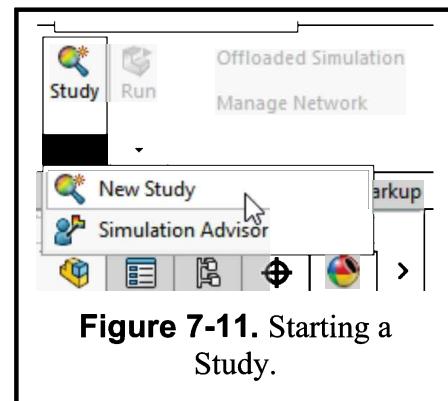


Figure 7-11. Starting a Study.

To start the analysis, select the **Simulation** tab, click on **New Study** and select the **New Study** command, as shown in **Figure 7-11**.

- In the Study name type “*Static 1- your-name*”.
- In the type of study select “*Static*”.
- And click **OK** as shown in **Figure 7-12**.

Now you can see the Study Manager Tree tab. See **Figure 7-13**.

Assigning Material to the Parts

If you had assigned a material to a part in SOLIDWORKS, this information will be used in the analysis. Also, you can use the **SOLIDWORKS Simulation** materials library, which includes a wide variety of materials, including their physical properties needed for analysis.

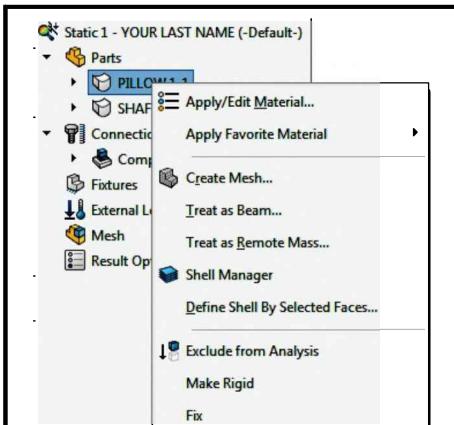


Figure 7-14. Assigning Materials.

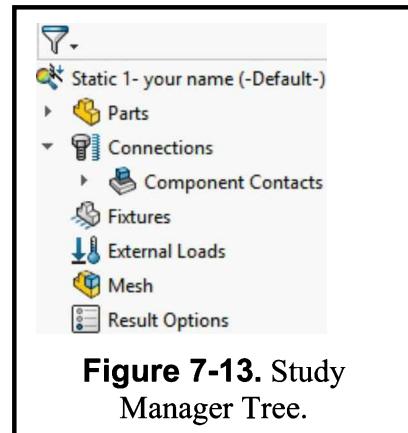


Figure 7-13. Study Manager Tree.

Since you have not defined a material for either part, expand the **Parts** section of the study, right click on **Pillow1** and select **Apply/Edit Material**. From the library select **Cast Alloy Steel** and click **OK**. Repeat the same process to define the **Shaft's** material as **Allow Steel**.

A green check mark will be added to a part after the material has been defined.

Applying Restraints

The restraints will define how the model is constrained. It is important to make sure the model cannot move, otherwise the **Static analysis** will return an error.

For this example, you will assume the bottom faces of the **Pillow Block** are fixed and cannot be moved. In the Study Manager right click in **Fixtures** and select **Fixed Geometry**. Select the two faces indicated in **Figure 7-16** and click **OK** to continue.

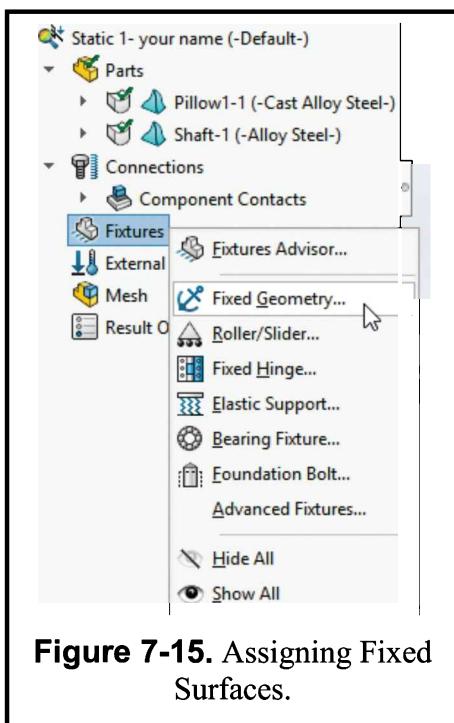


Figure 7-15. Assigning Fixed Surfaces.

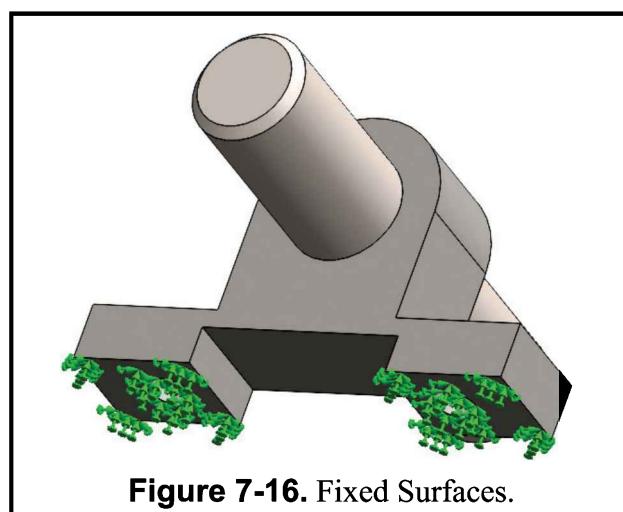


Figure 7-16. Fixed Surfaces.

Applying the Force on the Shaft

In your **Static** analysis you will apply a force to the **Shaft** to determine the stress in the **Pillow Block**. In the Simulation Manager right click on **External Loads** and select the **Force** option, as shown in **Figure 7-17**.

Activate the **Force** option and select the cylindrical face of the **Shaft**. By default, the direction of a force is perpendicular to the face where the force is applied. For a cylindrical face, the direction is radial.

For your analysis, the direction of the force must be downward. Activate the option **Selected direction** and select the **Top Plane** from the fly-out FeatureManager. Set the units to the **SI** system.

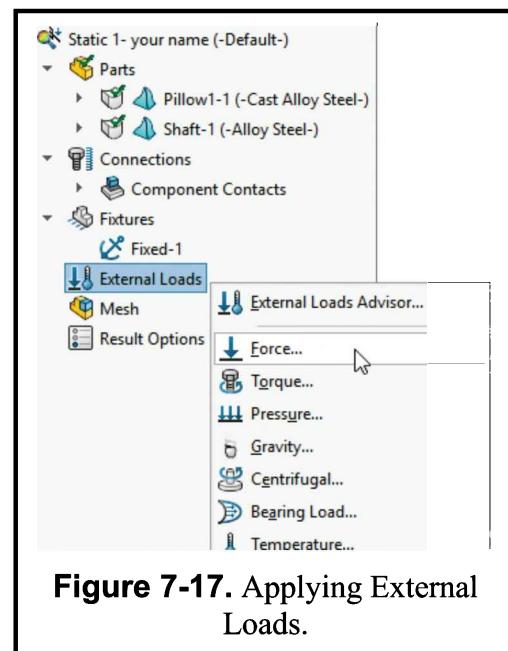


Figure 7-17. Applying External Loads.

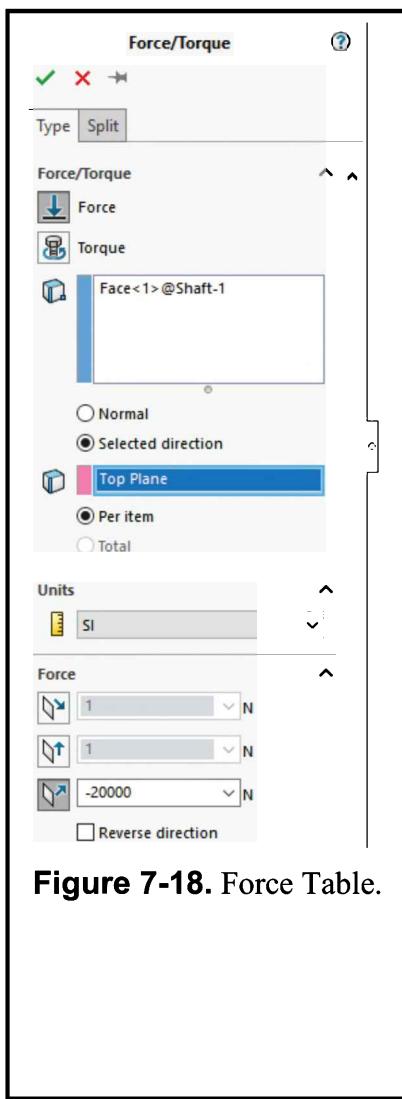


Figure 7-18. Force Table.

In **Force** use the option **Normal to Plane**,  enter a value of **-20,000 N** and click **OK** to add the force, as shown in **Figure 7-18**. Make sure the direction vectors are pointing down, and if needed, click in the **Reverse Direction** checkbox.

After adding the force to the shaft, you have added all the information needed to run your analysis. Your analysis study should now look like **Figure 7-19**.

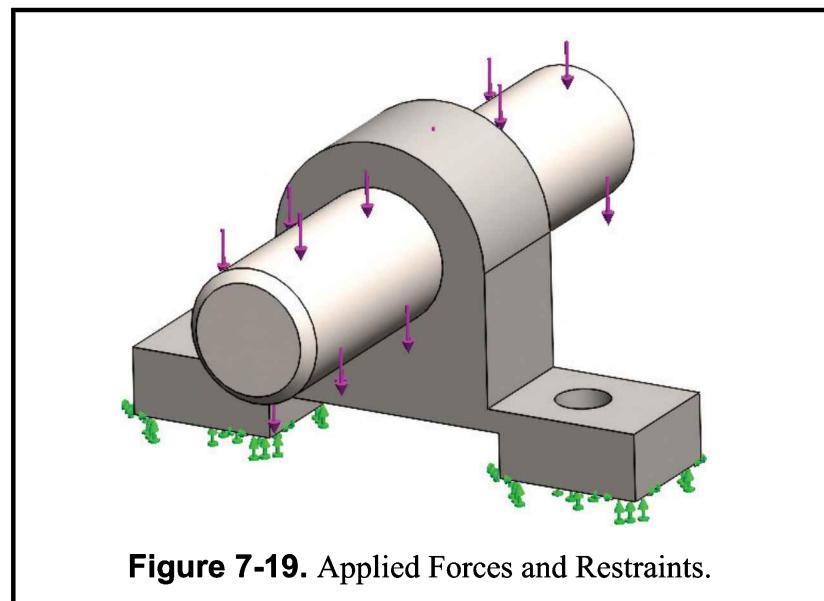


Figure 7-19. Applied Forces and Restraints.

Creating the Mesh

The next step in your analysis is to generate the Mesh for the analysis. In simple terms, the model is automatically divided in thousands of small pieces or elements. An equation is generated for each element, and after the equations for all elements are solved simultaneously the stresses are calculated.

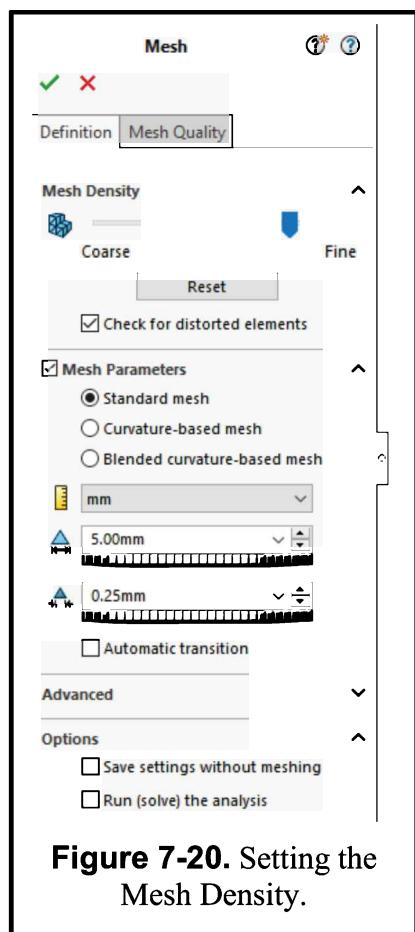


Figure 7-20. Setting the Mesh Density.

To generate the mesh, in the Simulation Manager right click on **Mesh** and select **Create Mesh**.

In the **Mesh** options set the units to **mm**, change the **Global Size** to **5 mm** and the **tolerance** to **0.25 mm**, as seen in **Figure 7-20**. Keep in mind that a smaller mesh will typically yield a more accurate solution, but also will take longer to calculate. Click **OK** to generate the mesh. When the mesh is complete a green checkmark is added in the Simulation Manager, and the mesh is displayed as seen in **Figure 7-21**.

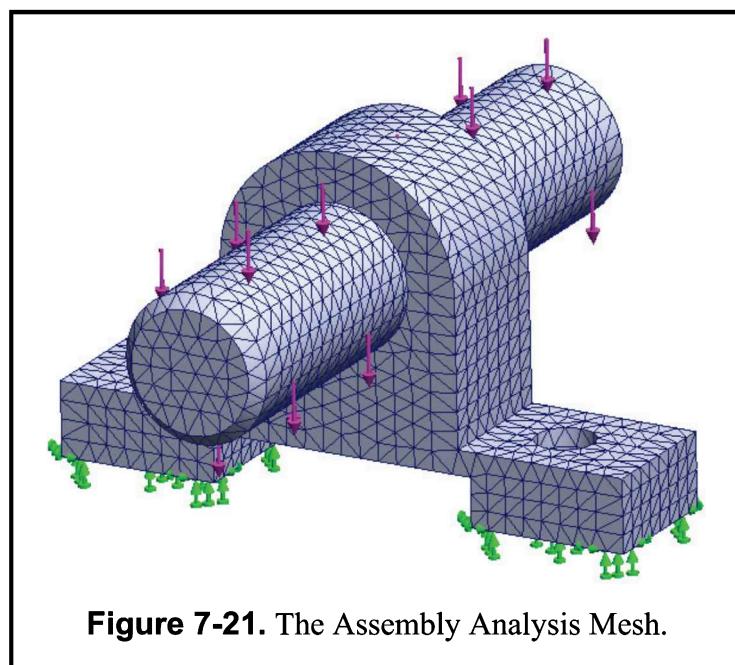
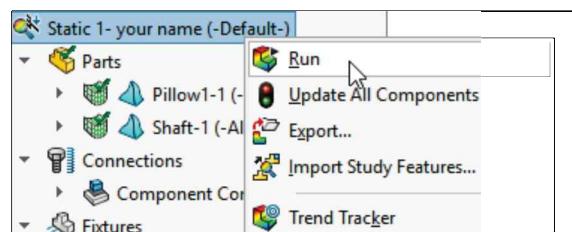


Figure 7-21. The Assembly Analysis Mesh.

Static Analysis

After the mesh is generated and the necessary information has been defined (forces, materials and restraints) you are ready to run the analysis. In the Simulation Manager, right click on the name of the study (**Static 1 -your name**) and select **Run** to begin the solution process. Depending on the number of components and the complexity of the analysis, the analysis may take longer to run. In this case, the analysis is quite simple and would only take a few seconds to solve.



Post-processing

After the analysis is complete a **Results** folder is added to the Simulation Manager, including Stress, Displacement and Strain as shown in **Figure 7-22**. To display the results, you can right click on any of them and select **Show**. You can also change the parameters to display the results or delete them. When the Stress results are displayed, they look like **Figure 7-23**.

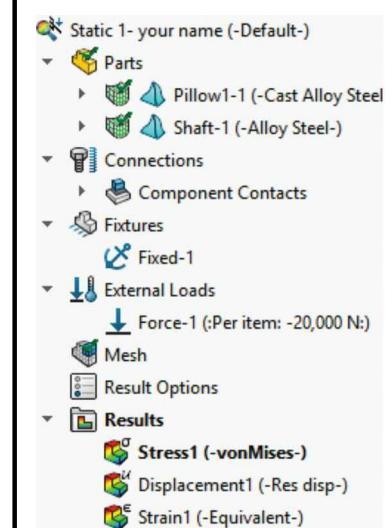
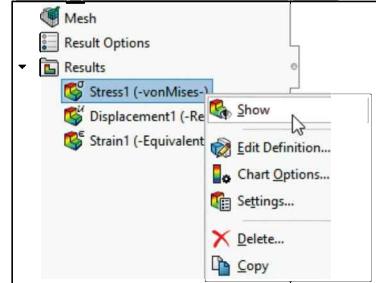


Figure 7-22. Analysis Results.

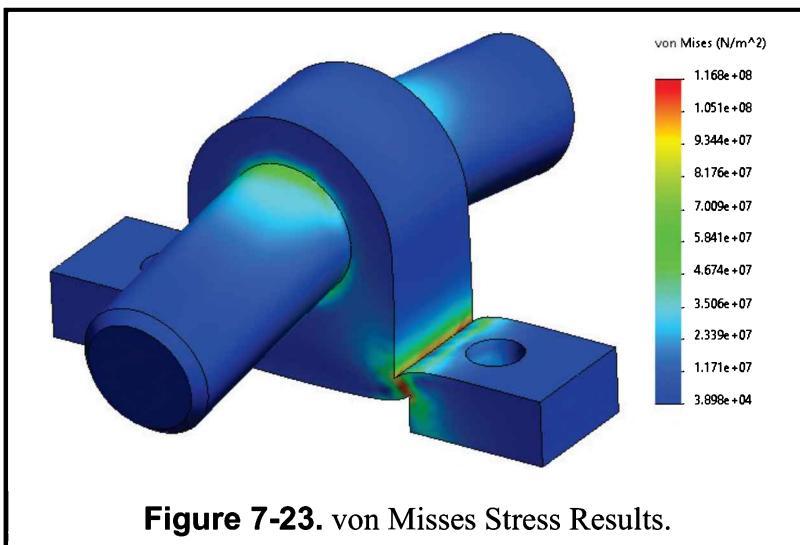


Figure 7-23. von Misses Stress Results.

The stress results show the model with an exaggerated deformation. To change how the results are shown, right click on **Stress1** and select the option **Edit Definition**. Here you can change the **Deformed shape**'s scale, the type of results to display as well as the units of measure, as shown in **Figure 7-24**. Feel free to explore the different options available to show the results.

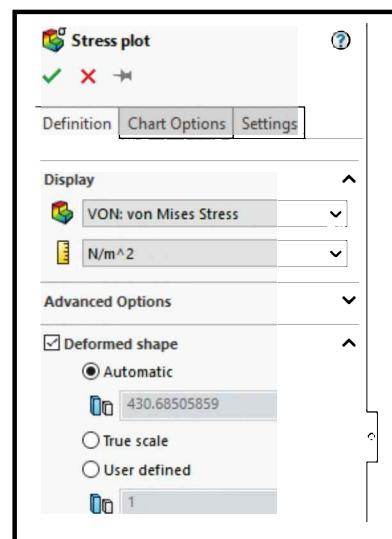


Figure 7-24. Stress Plot Options.

If you right click in the **Stress1** results and select the **Animate** option, the model will deform from the original shape to the final deformation using the scale used in the **Deformed shape** option, in this case in the range of 430 times the actual scale. Using the **Section Clipping** option will allow you to show areas above or below a certain stress value.

Right click on **Stress1** and select **Print** to make a printout of the results. Change the print format to Landscape and click **OK**. Make a new drawing of the **Pillow Assembly -1** with an **Isometric** view and print a hard copy to submit to your lab instructor along with the **Stress1-Von Mises** results plot.

ANALYZING THE PILLOW BLOCK RESULTS

The results of the analysis will help you understand if your design will perform as intended. In a static stress analysis, an important parameter to consider is the material's **Yield Strength**. This is the stress at which the material starts to permanently deform. In simple terms, it means that a region with a stress **below** this level will "spring back" into the original shape, and a region with a stress **above** this level will be permanently deformed or break and would be considered to have failed.

Another factor to consider in a design are the unexpected "over-loads" that could occur due to unforeseen circumstances. A **Factor of Safety (FOS)** can be defined as the relation between the Failure Stress and the maximum Allowable Stress. For most practical purposes when using **Static Stress Analysis**, the **Yield Strength** can be considered to be the **Failure Stress**.

$$FOS = \frac{\text{Failure Stress}}{\text{Allowable Stress}}$$

Depending on the type of design, the application, environment, etc., different factors of safety values are typically used by the industry. For this lab we will assume an arbitrary **FOS = 5**. Reviewing the material properties for **Cast Alloy Steel** you will see the **Yield Strength** is approximately **241 x 10⁶ N/m²**.

Material Type	Property	Value	Units
Alloy Steel	Elastic Modulus	1.9e+11	N/m ²
Alloy Steel (SS)	Poisson's Ratio	0.26	N/A
ASTM A36 Steel	Shear Modulus	7.8e+10	N/m ²
Cast Alloy Steel	Mass Density	7300	kg/m ³
Cast Carbon Steel	Tensile Strength	448082500	N/m ²
Cast Stainless Steel	Compressive Strength		N/m ²
Chrome Stainless Steel	Yield Strength	241275200	N/m ²
Galvanized Steel	Thermal Expansion Coefficient	1.5e-05	/K
Plain Carbon Steel	Thermal Conductivity	38	W/(m·K)

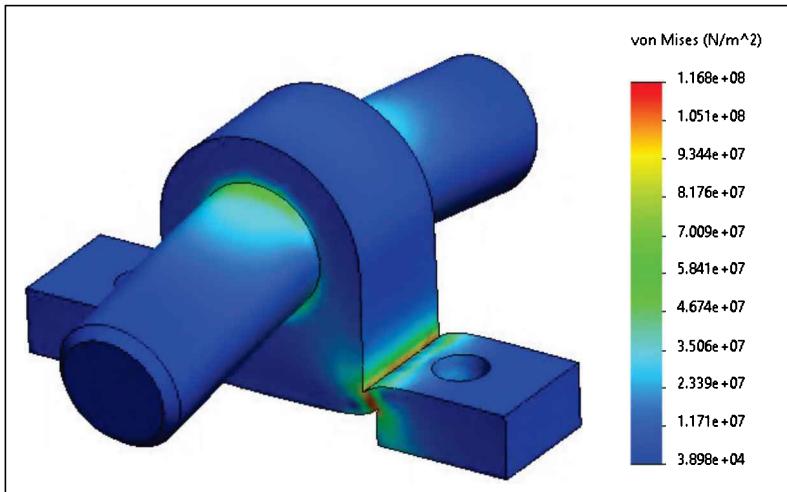
Considering this, the maximum stress allowed in the part should be:

$$\text{Maximum Stress} = \frac{\text{Yield Strength}}{\text{FOS}}$$

$$\text{Maximum Stress} = (241 \times 10^6 \text{ N/m}^2) / 3 = \underline{80.3 \times 10^6 \text{ N/m}^2}$$

Reviewing the analysis Stress Plot you can see the maximum stress calculated in the part are the red colored regions around the Pillow Block's base, and the corresponding stress in the vertical scale with a value of approximately **116 x 10⁶ N/m²**, which is higher than the calculated maximum allowable stress calculated with the Factor of Safety.

NOTE: Because of the nature of how FEA analysis calculates results, the stress values you obtain may be slightly different but generally close to these values.



In order to reduce the stress in the part, the designer has different options, including:

- Change the geometry
- Reduce the external loads
- Change the material

Depending on the application, design restrictions, budget, etc., you could change one or more parameters. For this lab we will only change the part's geometry.

PILLOW BLOCK DESIGN CHANGES

To make the changes open the **Pillow1** part, select the Boss-Extrude1 feature and select the **Edit Sketch** command from the context toolbar. Change to a **Front** view and change the sketch dimensions as indicated in **Figure 7-25**. After the changes are made, Rebuild the model to continue.

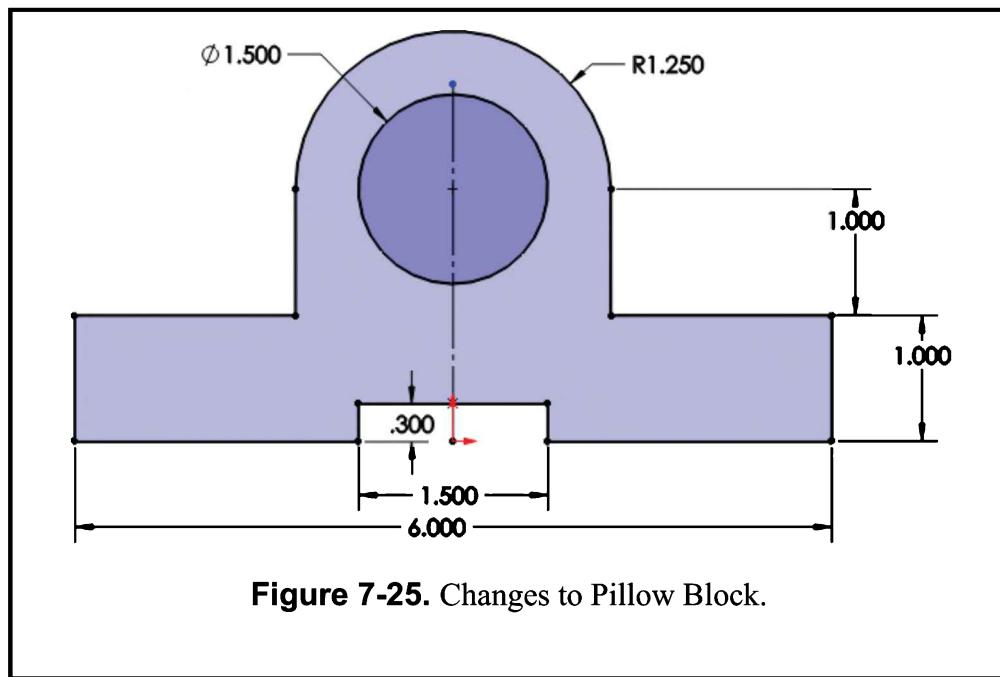
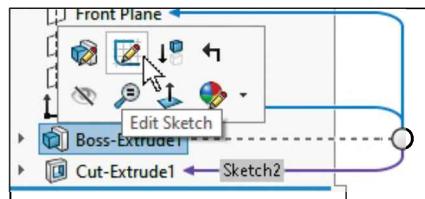


Figure 7-25. Changes to Pillow Block.

Next change to a **Top** view and **Edit** the sketch for the Cut-Extrude1 mounting holes. Change the spacing between holes as shown in **Figure 7-26** and **Rebuild** the part to continue.

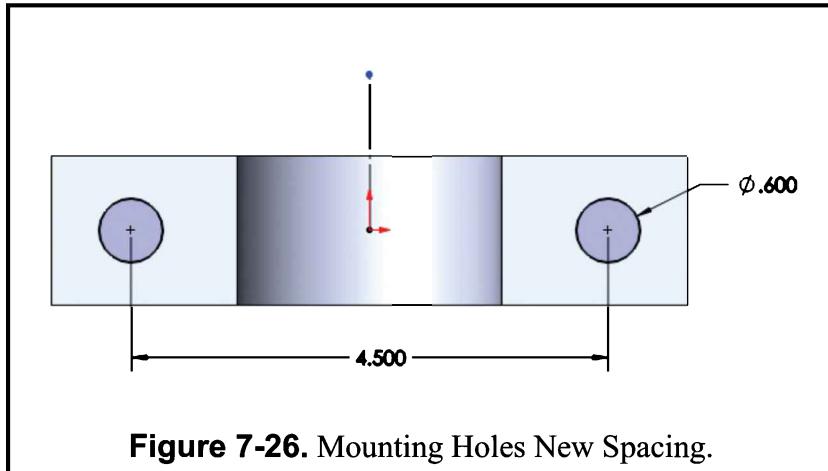


Figure 7-26. Mounting Holes New Spacing.

A common practice to reduce stress concentrations in a part is to round the corners. Add the fillets indicated in **Figure 7-27** to your model and save the changes when finished.

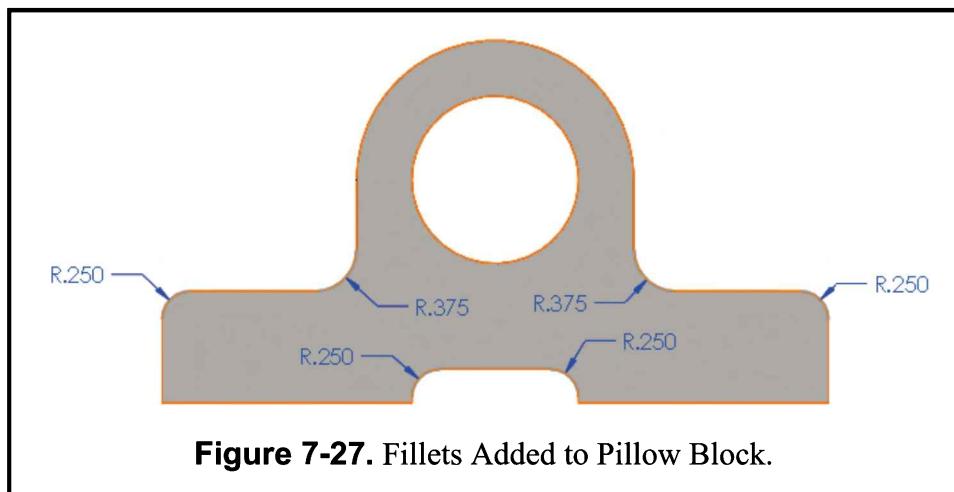
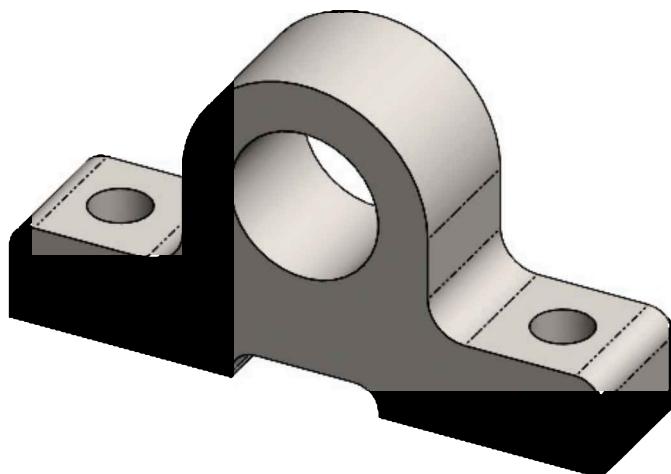
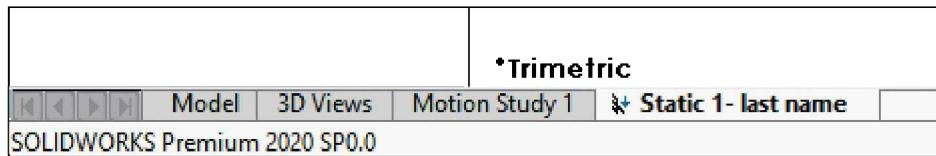


Figure 7-27. Fillets Added to Pillow Block.

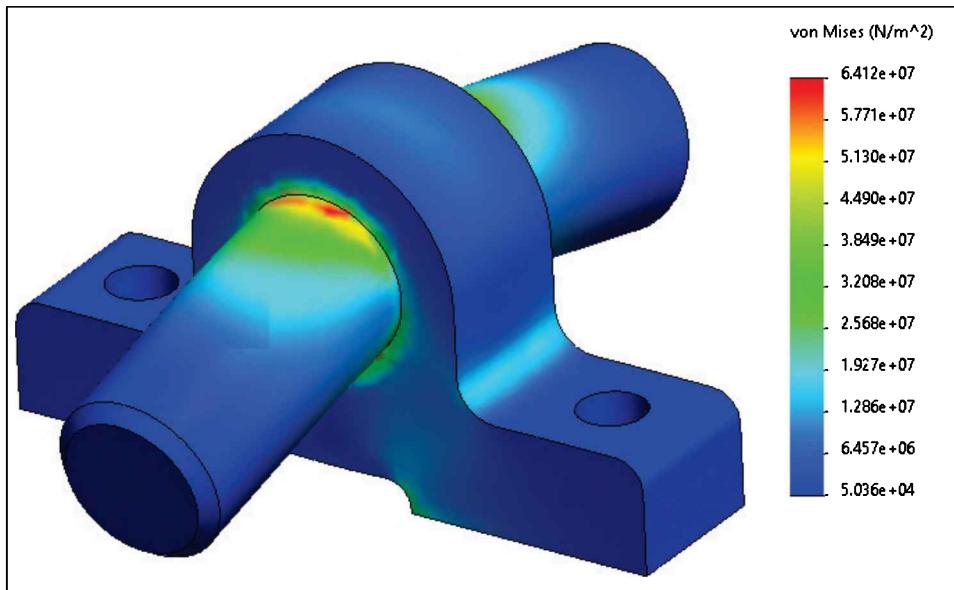


ANALYSIS OF UPDATED PILLOW BLOCK

After the **Pillow Block** has been modified, return to the assembly, and select the **Simulation** tab at the bottom to access the FEA analysis. SOLIDWORKS Simulation will recognize the geometry has been modified and alert you the results are no longer valid. To update the study, right click in the **Static 1- your name study** and select the **Run** option. Using this option will automatically re-mesh the part using the same settings, and re-run the analysis immediately after, presenting you the updated results.



Using the analysis results for the updated design, calculate the lowest **Factor of Safety** using the maximum **Von Mises stress** calculated. Print the results of the updated **Stress1-Von Mises** and turn it in along with the first results to your lab instructor.



Exercise 7.2: Thermal Analysis of a Computer Chip

A computer chip is one of the many components in a computer, yet it contributes significantly to the very thing that may damage its performance. Computer designers must consider temperature, fatigue, and chemical and material properties among the many factors that may affect the performance. Heat has to be a major consideration when designing a computer system. As heat builds up in a component, the function and integrity of that component and even the entire system may be affected.

In this exercise you will design a 3D model of a computer chip and a heat sink to see how one affects the other. Then you will perform a Thermal Finite Element Analysis to visualize how efficiently the heat sink is dissipating heat from the chip.

CONSTRUCTING THE HEAT SINK

Start by making a new part using the **ANSI-INCHES** template and change the Units to **4 decimal places**. Start a new sketch in the **Top Plane** and draw a **Center Rectangle** starting at the origin. Dimension both sides **2.00"** and make an Extruded Boss going down **0.125"**, as shown in **Figure 7-28**.

Now change to a **Front** view and add a new sketch in the front-most face. Draw the Sketch in **Figure 7-29** and make an extrusion using the **Through All** end condition towards the back of the part. Make sure the vertical **Centerline** has a **Midpoint** relation to both the top and bottom horizontal lines to fully define your sketch.

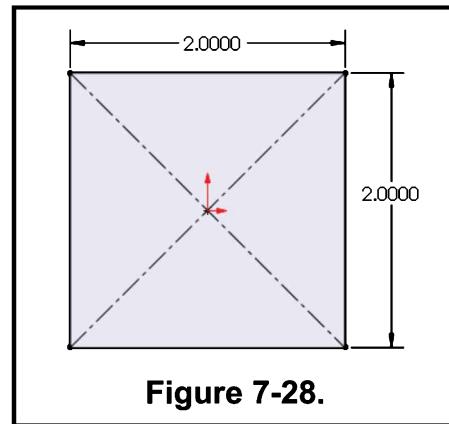


Figure 7-28.

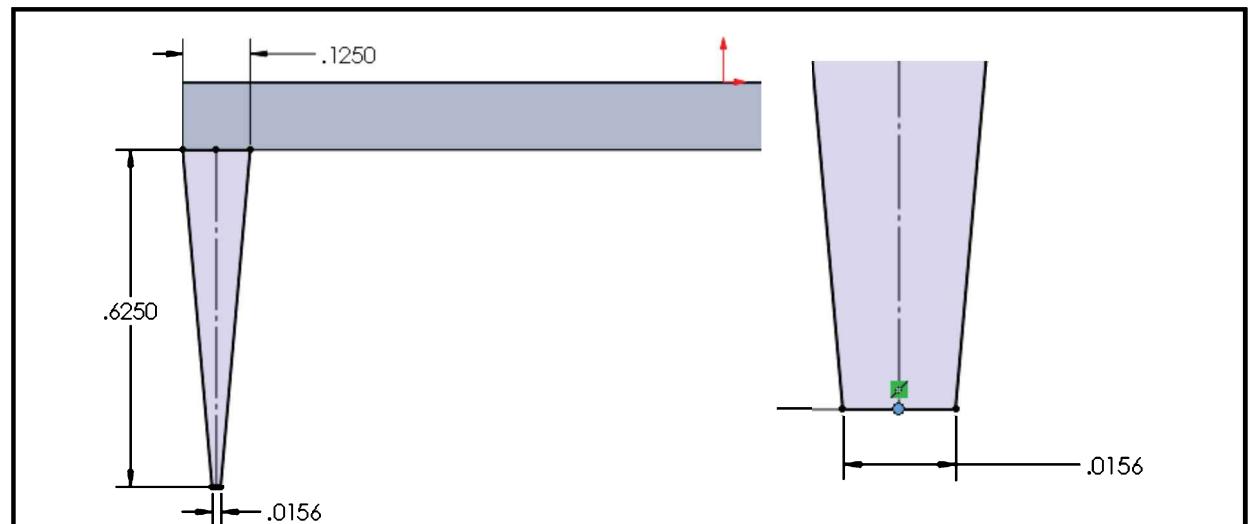


Figure 7-29. Detail of the Small End of the Fin Profile.

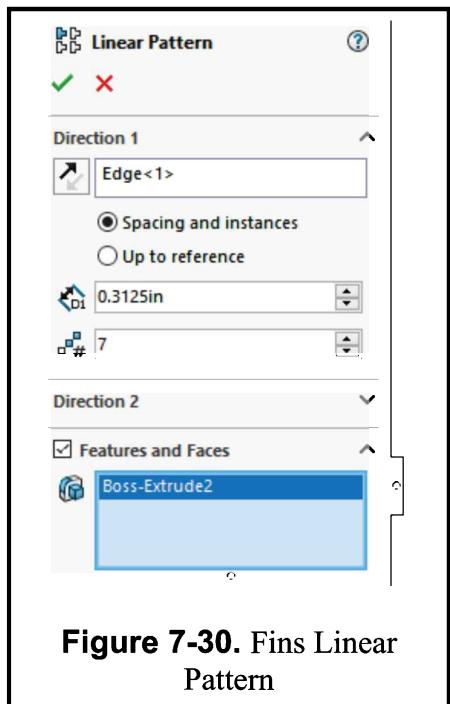


Figure 7-30. Fins Linear Pattern

Now you need to make a **Linear Patter** of Fins. Select the **Linear Pattern** command, select the top edge of the part for the Direction, set the spacing to **0.3125"** with **7** copies, add the Fin to the **Features** to copy and click **OK**.

Set the part's material to **1060 Alloy** from the Aluminum Alloys library and save your part as **Heat Sink.sldprt**. Your finished Heat Sink will look like **Figure 7-31**.

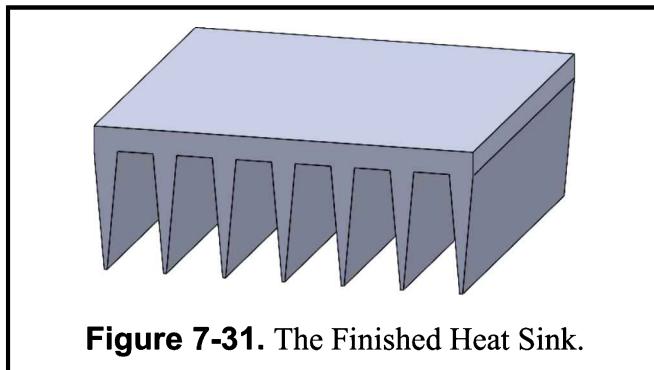


Figure 7-31. The Finished Heat Sink.

DESIGN THE MICROCHIP

Start a new part using the **ANSI-INCHES** template, and just like with the **Heat Sink**, add a new sketch on the **Top Plane**. Draw the profile shown in **Figure 7-32** using a **Center Rectangle** and dimension as indicated. Extrude it **0.125"** up and change the part's material to **Ceramic Porcelain** from the **Other Non-Metals** library. The finished Micro Processor is shown in **Figure 7-33**. Save your part as **Microchip.sldprt**.

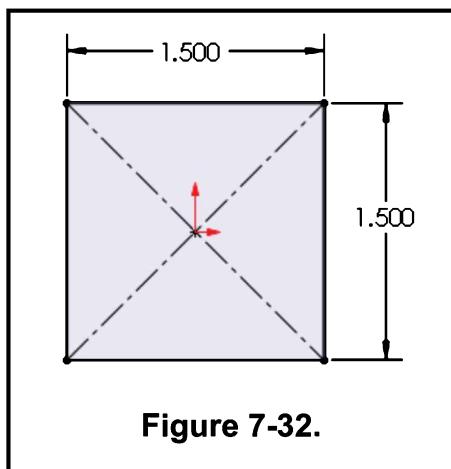


Figure 7-32.

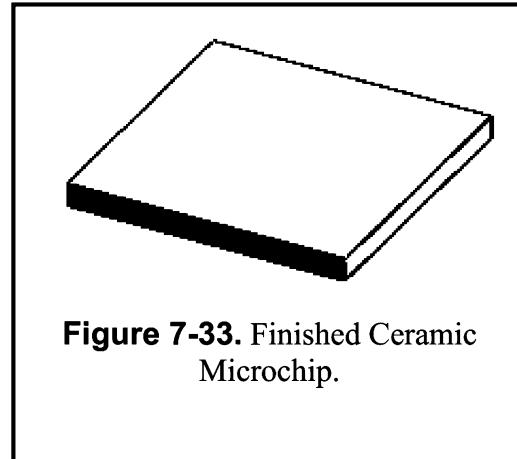


Figure 7-33. Finished Ceramic Microchip.

MICROCHIP AND HEAT SINK ASSEMBLY

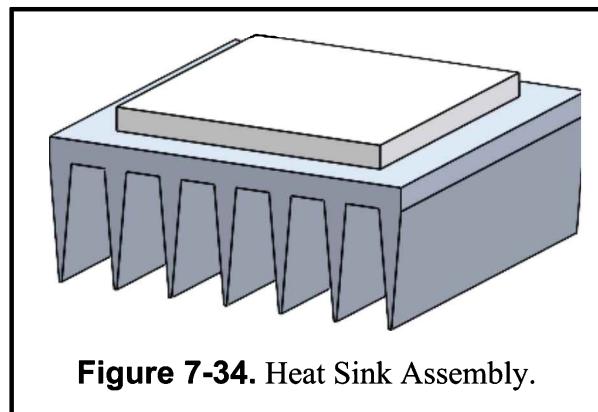


Figure 7-34. Heat Sink Assembly.

Start a new assembly, in the **Begin Assembly** command, select the **Microchip** and click **OK** to add it at the origin. Next add the **Heat Sink** using the same process to also align it with the assembly's origin. Your assembly should look like **Figure 7-34**. Save your assembly as Thermal Study.sldasm.

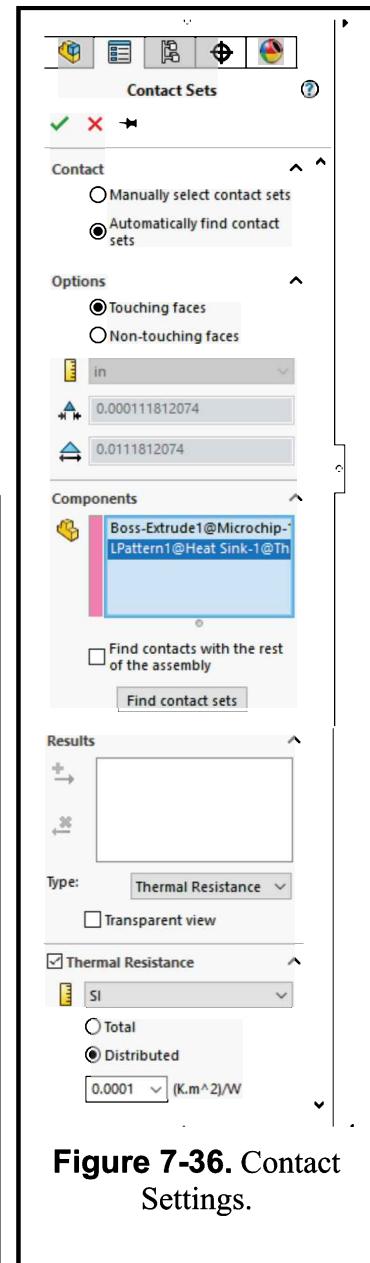


Figure 7-36. Contact Settings.

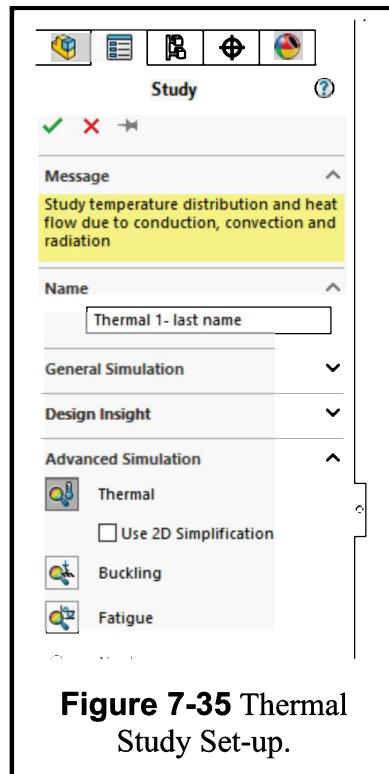


Figure 7-35 Thermal Study Set-up.

THERMAL ANALYSIS

Starting the Study

If not already loaded, go to the menu **Tools**, **Add-Ins** and load the **SOLIDWORKS Simulation** software. Select the **Simulation** tab and click on the **Study** command and click on **New Study**. In the name field type **Thermal Study 1 - your last name**. Select the Thermal option from the Advanced Simulation and click **OK**. See **Figure 7-35**.

In the **Thermal Study** manager right click in **Connections** and select **Contact** Set. Since there are only two components you can use the option **Automatically find contact sets**. In the options select the **Touching faces**. Under **Components** select the two parts of the assembly.

Under **Results** select the **Thermal Resistance** type. Under Thermal Resistance use the **SI** units, select the **Distributed** option and set the value to **0.0001 K-m²/W**. See **Figure 7-36** and click **OK** to continue.

Identifying Heat Source and Convection Settings

Depending on the type of **Microchip**, the heat generated will vary greatly. For this exercise you will assume **15 Watts** of power will be dissipated. In this step you will identify the heat source.

Right click on **Thermal Loads** and select **Heat Power**. In the graphics area right click on the **Heat Sink** and **Hide** it and rotate the assembly to select the bottom face of the **Microchip** that will be in contact and will transfer heat to the **Heat Sink**.

In the **Heat Power** section select the **SI** units and set the power to **15 Watts**, as seen in **Figure 7-37**. Click **OK** to continue and make the **Heat Sink** visible.

The last step before creating the mesh is to identify the surfaces that will disperse the heat from the microchip. Right click on **Thermal Loads** and select **Convection**. Select all the surfaces of the **Heat Sink** except for the top surface in contact with the **Microchip**.

Now set the **Convection Coefficient** to **15 W/(m²-K)** and set the **Bulk Ambient Temperature** to **300° K**, as seen in **Figure 7-38**.

Create the Mesh

Just as with the Static Stress analysis, you also need to create a **Mesh** for a Thermal study.

Right click on **Mesh** and select **Create Mesh**. Under the **Mesh Parameters** change the units to **mm**, set the **Global size** to **2.0 mm** and the **Tolerance** to **0.10 mm**, as shown in **Figure 7-39**.

Click **OK** to mesh the assembly. The finished mesh for the analysis will look as in **Figure 7-40**.

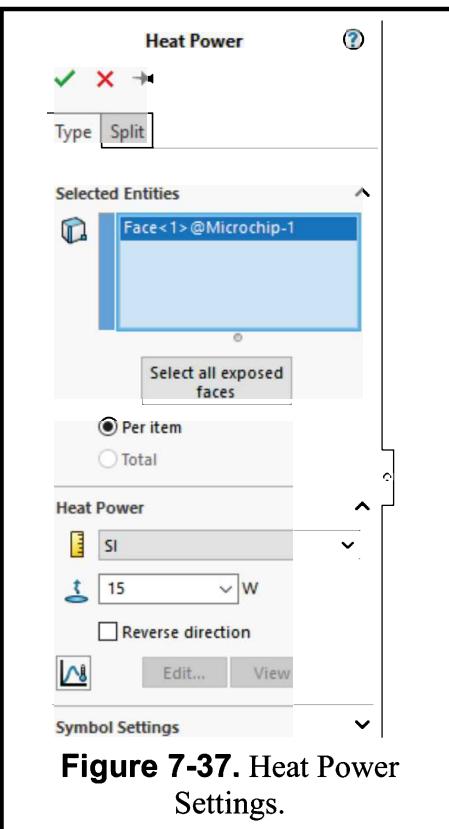
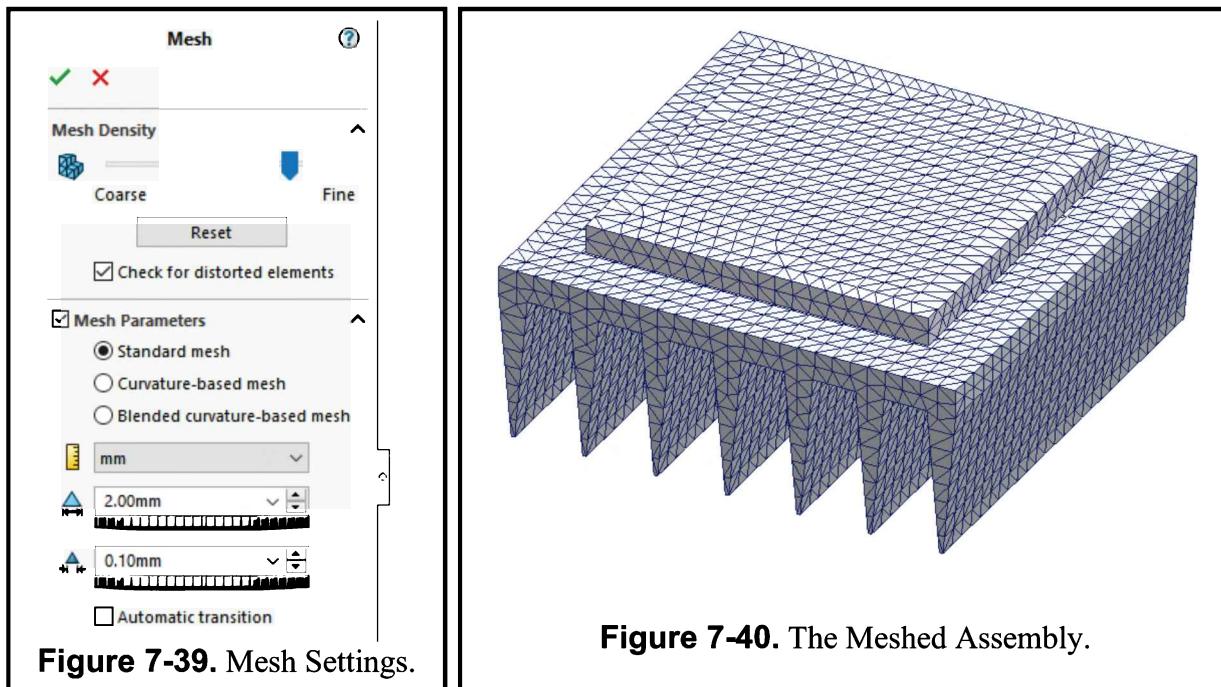


Figure 7-37. Heat Power Settings.



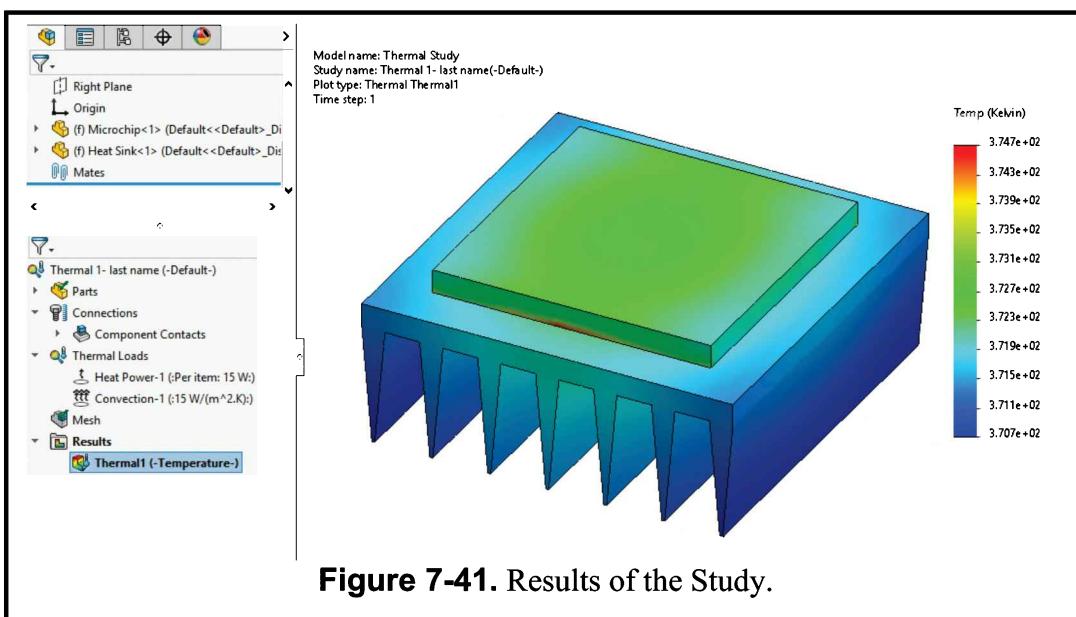
Figure 7-38. Convection Settings.



Running the Analysis

Now that you have entered the necessary information for the analysis, right click on the study name and select **Run**. The simulation processor will run the analysis and should only take a short moment (about a minute).

When the analysis is completed a folder named **Thermal1 (Temperature)** with results will be added to the Study Manager, and the results will be automatically displayed as seen in **Figure 7-41**.



Right click on the **Thermal1 (-Temperature-)** results and select **Print** to get a hard copy of your thermal study results.

Now right click on the **Thermal1** results and select **Edit Definition**. Under **Display** select the **HFLUXN: Resultant Heat Flux**, as seen in **Figure 7-42**.

Under **Advanced Options** check the box for **Show as vector plot** and click **OK**. To modify the results, plot right click on the **Thermal1** results and select **Vector Plot Options**. Change the vector size to **500** and the density of vectors to **25**, as shown in **Figure 7-43**. The heat vectors will be displayed in a wireframe model of the assembly, showing the heat flow (**Figure 7-44**). Finally **print** a copy of the **Heat Flow vectors** to deliver to your lab instructor.

Make a new drawing of the **Thermal Study** assembly and print a hard copy. Submit the assembly drawing along with the two thermal study results to your instructor to finish.

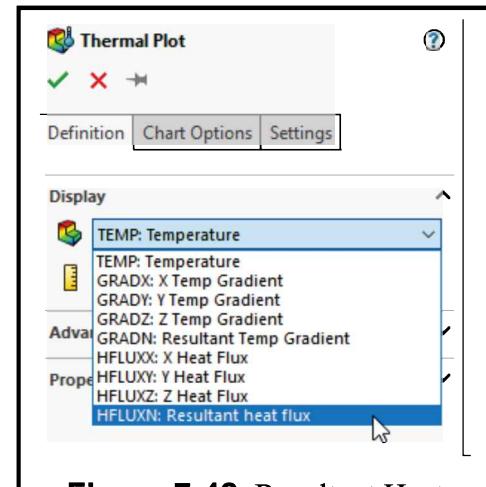


Figure 7-42. Resultant Heat Flux.

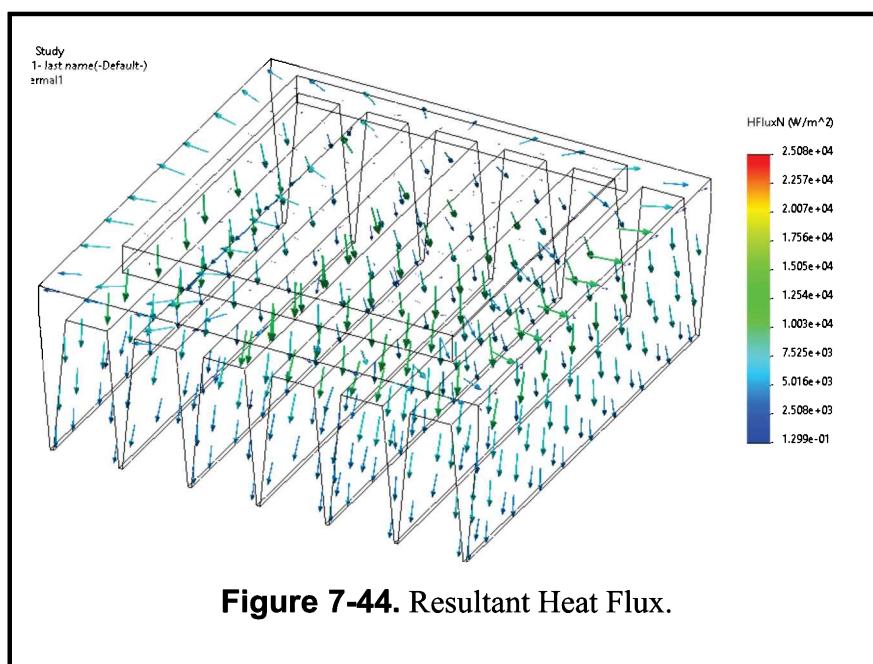


Figure 7-44. Resultant Heat Flux.

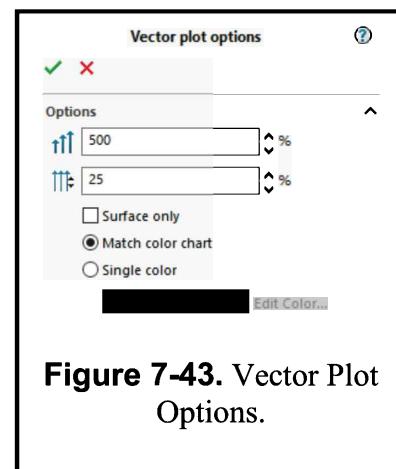


Figure 7-43. Vector Plot Options.

Design Workbook Lab 8:

Animation, Detailing and Rapid Prototyping

In this Lab you will learn how to create an Exploded view of an assembly, used mainly for illustration, assembly instructions or documentation, and how to create a video animation of it. You will also learn how to add different drawing views of a part to a detail drawing, used to document how a part will be manufactured, and finally, you will cover the basic principles of Rapid Prototyping, additive manufacturing, and how to make an .STL file to create a physical prototype of your designs.

ASSEMBLY EXPLODED VIEW



In a previous lab you learned how to assemble individual components, and now you will learn how to create an **Exploded View**. With an assembly open, select the **Exploded View** command from the Assembly tab, or from the menu **Insert**, **Exploded View**.

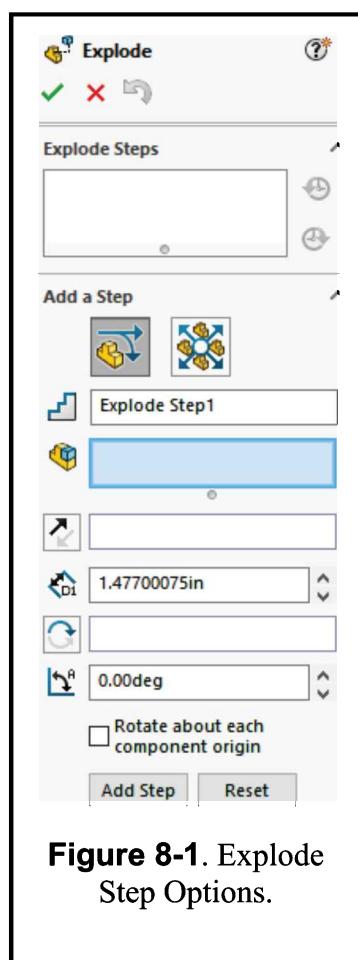


Figure 8-1. Explode Step Options.

When the **Exploded View** command is loaded, as seen in **Figure 8-1**, you can start selecting the components to explode, and where you can define the options for each of the explosion steps of the assembly.



To add a translation or rotation step, activate the **Standard Step** option and select the part to explode. Immediately after selecting the part, you will see three axes to define the exploded step direction. You can:

- Click and drag the direction arrow in the screen, or
- Select the direction arrow, enter the exploded step distance, and click on **Add Step**.

While dragging a direction arrow you will see a ruler to help you approximate the exploded step distance. After the direction arrow is released a new **Explode Step** is added, as shown in **Figure 8-2**. Now you click on **Done** to finish the explode step and continue with the next Explode step.

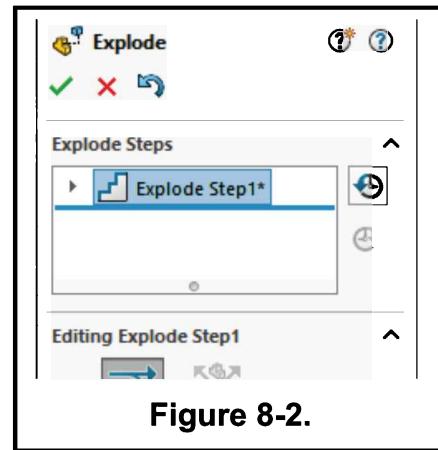


Figure 8-2.

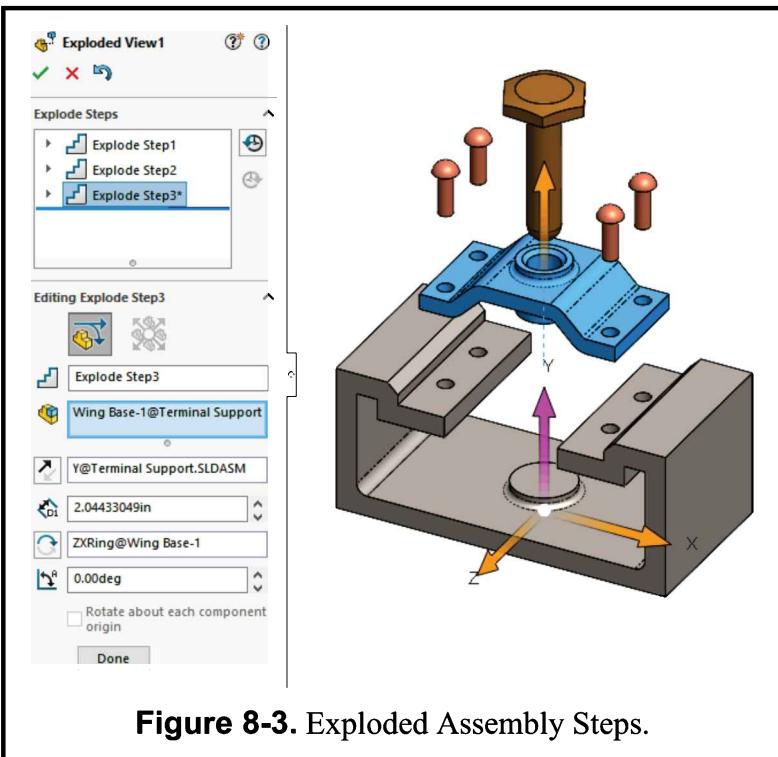


Figure 8-3. Exploded Assembly Steps.

An exploded assembly should look like **Figure 8-3**.

After the assembly's exploded view is finished, select the **Configuration Manager** tab, expand the **Default** (or the currently active configuration) and expand the **Exploded View1** section.

When you select an exploded step, you can see the explode view arrow, where you can click and drag it to adjust the step distance if needed. If you want to edit the entire exploded view, right click on the **Exploded View1** and select **Edit Feature** from the context menu.

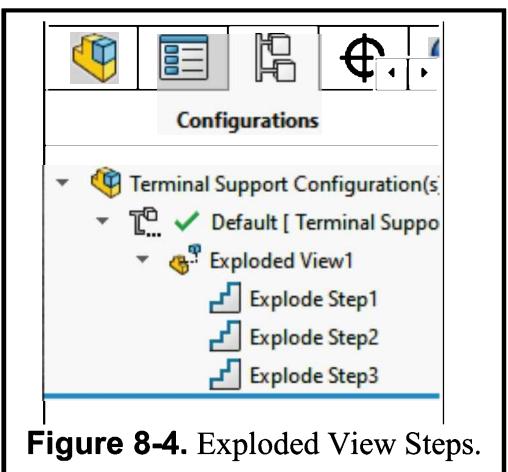
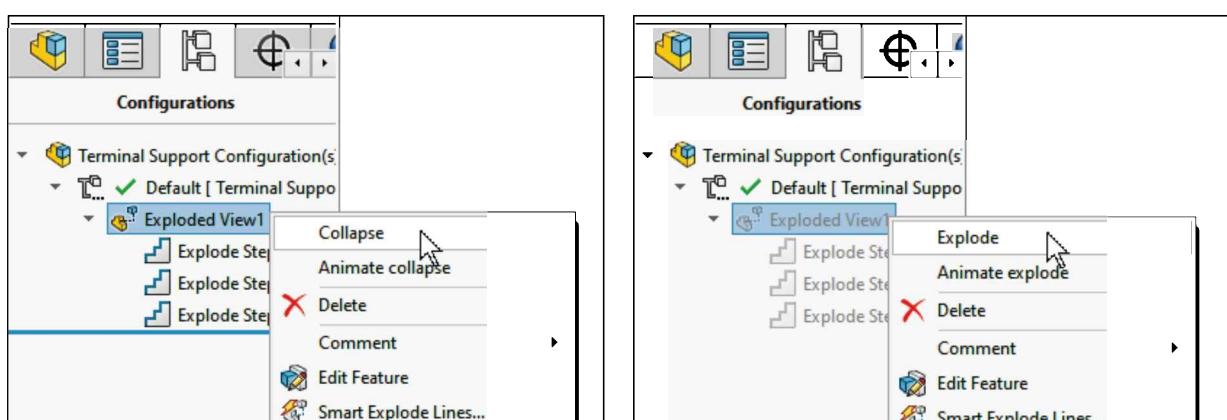


Figure 8-4. Exploded View Steps.

To collapse the exploded view, right click on it and select **Collapse** from the context menu, or double click on it.

To explode it again right click on the collapsed exploded view and select **Explode**, or double click on it.

You can also animate the explode and collapse from this context menu, where you will see an animation controller to play, pause, stop and save the animation.



ANIMATION WIZARD



The **Animation Wizard** is an easy to use tool to create an exploded view animation. You can access it by activating the **Motion Study1** tab on the bottom left side of the screen, or you can add a new study by selecting the **New Motion Study** command from the Assembly tab.



In the Motion Study toolbar select the **Animation Wizard** command. If you have added an Exploded View to your assembly, additionally to **Rotate model**, you will have the option to animate the **Explode** or **Collapse** of the exploded view. The difference between the option in the context menu previously shown and the **Animation Wizard** is that in the latter you can define the length of the animation, and later the timing of the explosion steps, as shown in **Figure 8-5**.

When you select the **Explode** or **Collapse** option and click **Next**, you will be asked to enter the duration of the animation, as shown in **Figure 8-6**.

Selecting the **Rotate Model** option, you will have the option to select the axis to rotate around and the number of rotations.

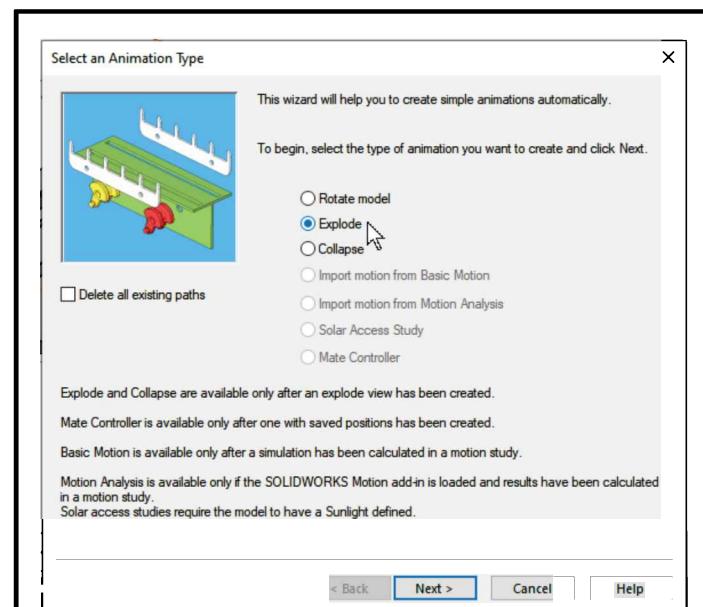
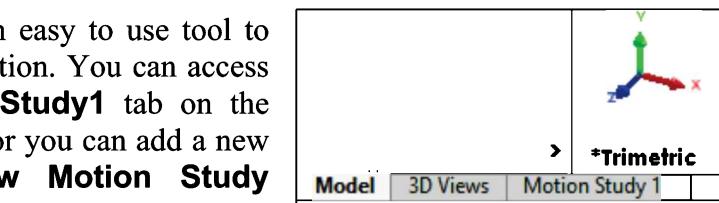


Figure 8-5. Animation Wizard Options.

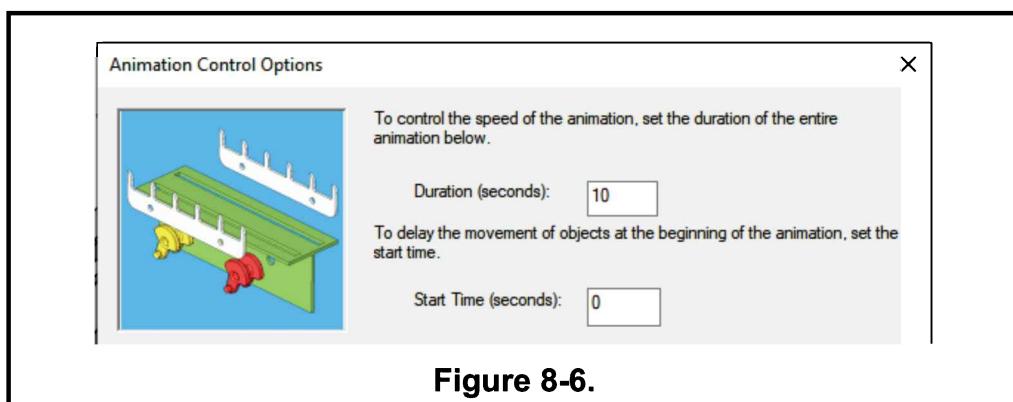


Figure 8-6.

ANIMATION CONTROLLER AND MOTION MANAGER

After creating an animation with the Animation Wizard, the MotionManager is automatically populated with the steps and their timing, as seen in **Figure 8-7**.

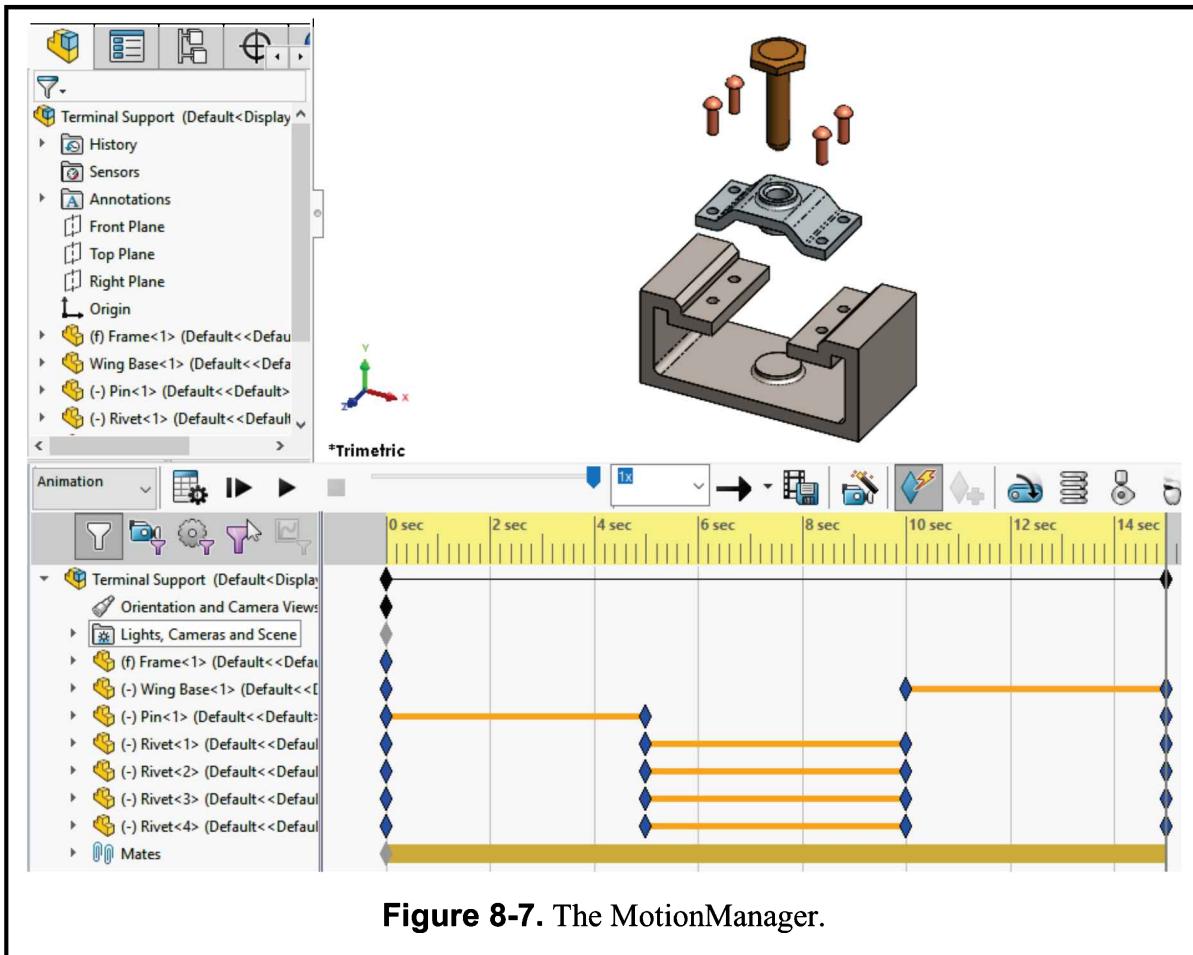


Figure 8-7. The MotionManager.

In the Animation Controller you can find the following options.



Calculate the animation from the start.



Play from Start stops the animation and starts to play from the beginning.

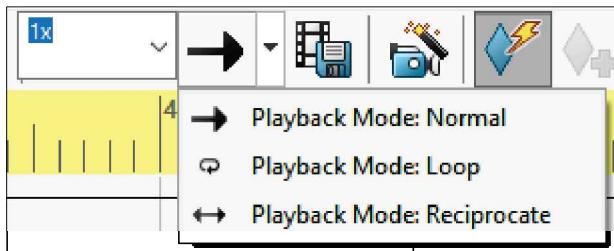


Play the animation at the specified speed.



Stop button stops the animation and sends it back to the start position.

There are three “**Playback Modes**” on the controller.



Normal Mode button sets the animation to run once from start to end at the specified speed multiplier.

Loop Mode button sets the animation to run in a continuous loop.

Reciprocate Mode button sets the animation to run forward and then backwards in a repeating loop.

Save as AVI File button will open the “Save Animation to File” menu and allows you to save the animation as an .AVI file that can then be played on a media player.

Animation Wizard launches the “Animation Wizard” tool covered previously.

INTRODUCTION TO PHYSICAL SIMULATION

Physical Simulation allows you to simulate the effects of motors, springs, and gravity on your assemblies. Physical Simulation combines simulation elements with SOLIDWORKS tools such as mates and Physical Dynamics to move components in your assembly.

The options available included for simulation are:



Linear Motor or Rotary Motor



Linear or Torsional Spring



Contact between components



Gravity effects along either axis and value

Exercise 8.1: Exploded Animation of the TERMINAL SUPPORT ASSEMBLY

CREATE EXPLODED ASSEMBLY VIEW

Open the previously made **Terminal Support.sldasm** assembly. First you need to add an Exploded View to the assembly. Select the menu Insert, Exploded View, or click on the **Exploded View** command from the Assembly tab.

The first part to explode is the **Pin**. Select it in the assembly, click and drag the **Y direction arrow** up, and using the reference ruler move it approximately a distance of **8.00"**. To complete the explode step click on the **Done** button. *Explode Step1* is added to the Explode Steps list at the top, as shown in **Figure 8-8**.

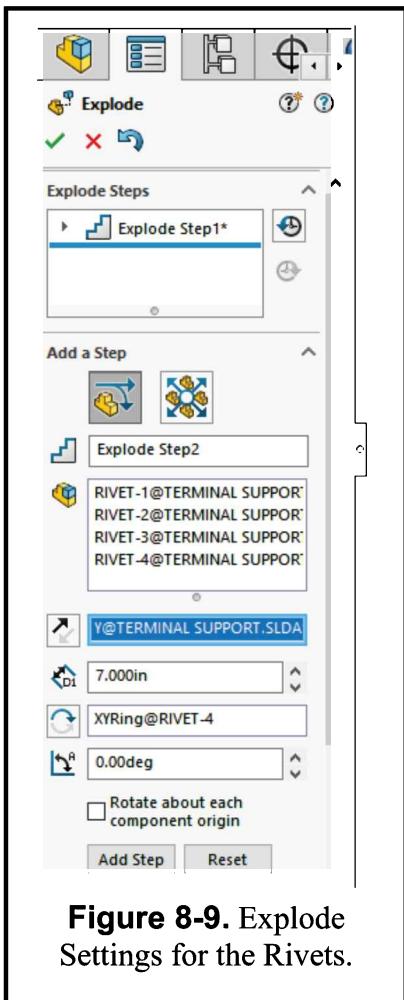


Figure 8-9. Explode Settings for the Rivets.

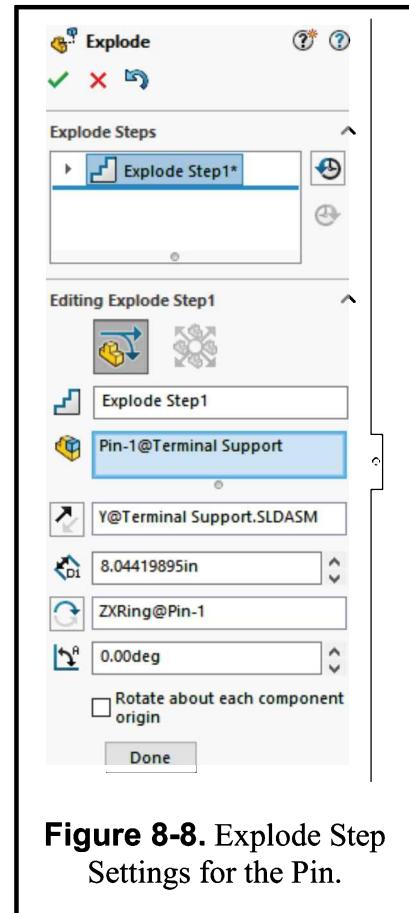


Figure 8-8. Explode Step Settings for the Pin.

In the next step you will explode the four **Rivets** at the same time, but in this case, you will define the exact distance to explode them. Select the four **Rivets** on the screen, and then click in the **Y arrow** to define the explode direction.

Enter **7.0"** in the Explode Distance box and, if needed, toggle the **Reverse Direction** button to make the Rivets go up. To complete the step, click on **Add Step** to continue. Note Explode Step2 is added to the Explode Steps list.

The correct explode settings for the Rivets are shown in **Figure 8-9**.



Figure 8-10. Explode Settings for the Wing Base.

The final step is to explode the **Wing Base**. Select it in the assembly, click on the **Y direction arrow**, enter a distance of **3.00"** and click on **Add Step** to continue. Explode Step 3 is added to the Explode Steps list. The explode settings for the Wing Base are shown in **Figure 8-10**.

The Terminal Support assembly is now exploded as shown in **Figure 8-11**.

After the three exploded steps are added to the assembly, click **OK** to finish the **Explode** command.

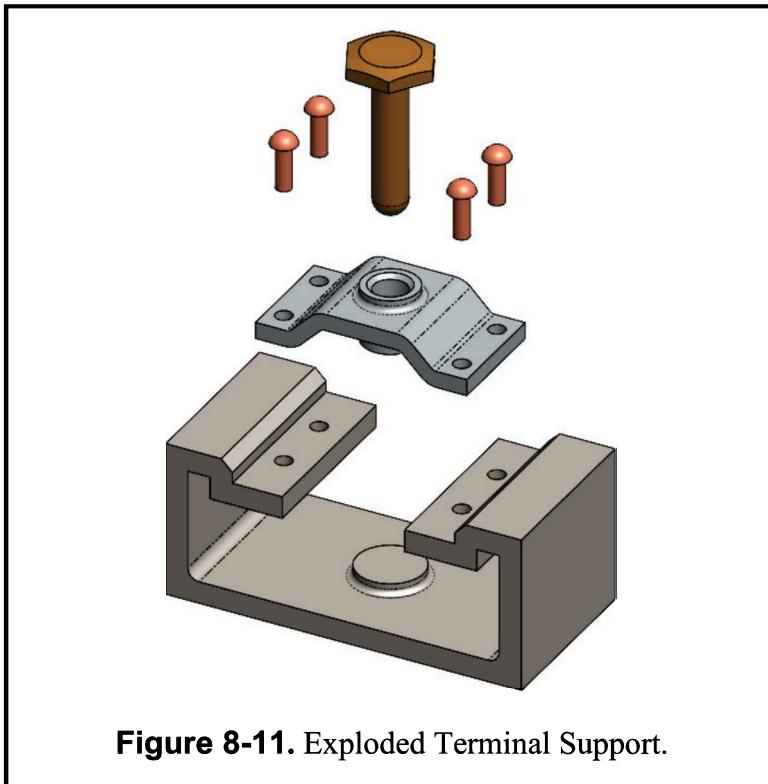


Figure 8-11. Exploded Terminal Support.

ASSEMBLY ANIMATION

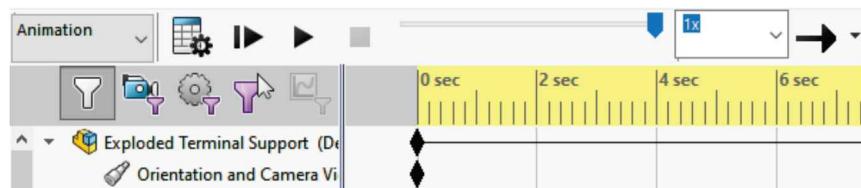


In this step you will create an animation of the assembly's exploded view. Select the **Motion Study1** tab on the lower left side of the screen to access the MotionManager.

Click on the **Animation Wizard** command in the Animation toolbar, select the **Explode** option, and click on Next. Set the **Duration** to **15** seconds, leave the **Start time** at **0** and click on **Finish** to continue.

PLAYING THE ANIMATION

In the Animation Controller toolbar press the **Play** button to see the animation. Change to the different play options including the Normal, Loop and Reciprocate Modes to see the effect on the animation.



SAVING THE ANIMATION



In the next step you will save your animation to a video file.

From the Animation toolbar select the **Save Animation** command.

In the **Save Animation to File** dialog you can select the folder to save the animation, the type of file (default is **.AVI**), the image size (default is the current screen size) and the aspect ratio for the video. Set the animation to **15 Frames per Second** and click on the **Save** button, as shown in **Figure 8-12**.

When asked to select the **Video Compression** codec use the default **Microsoft Video 1** and click **OK**. If asked to recalculate the animation to update the results, click **Yes** to finish. To finish this part of your lab, use the **Save As** command to save your exploded assembly file as **Exploded Terminal Support**.

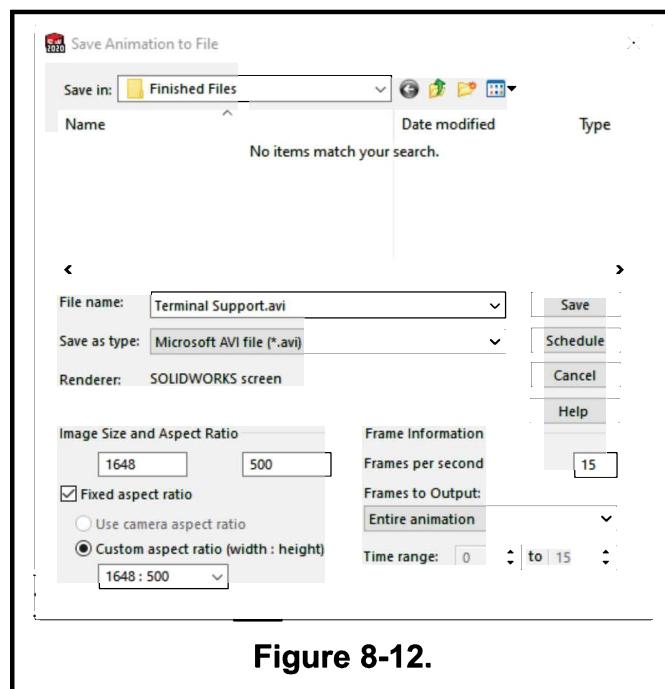


Figure 8-12.

Note: You may have to adjust your model viewport to fit the whole animation on your screen as it is being captured.

ANIMATION MOTION ELEMENTS

In the Motion Manager toolbar, you can optionally add **Linear** and **Rotary motors**, **Linear** or **Torsional Springs**, define **Contact** between different components and **Gravity** effects acting on the assembly. It must be noted that the Motion Elements can only be added to components that have the necessary degrees of freedom. In other words, if the Mates added to the assembly prevent the motion of a component, adding a motion element to it will not make it move.



In this step you will add a **Rotary Motor** to the Pin. Start by adding a new Motion Study to your assembly. Click on the **New Motion Study** command from the assembly tab.



In the animation toolbar click on the **Motor** command and select the **Rotary** option. Select the cylindrical surface of the **Pin** to define the component and direction of the motor, as seen in **Figure 8-13**.

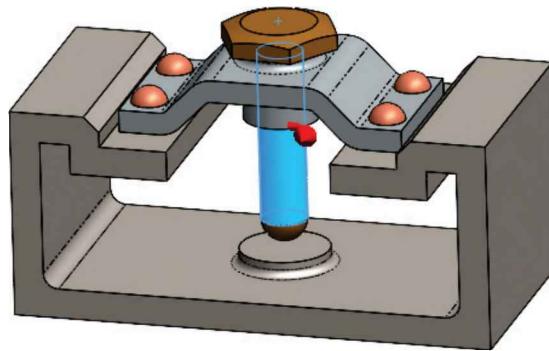


Figure 8-13. Adding a Rotary Motor to the Pin.

In the Motion section set the **Function** to **Constant Speed**, set the speed to **25 RPM** and click **OK**, as shown in **Figure 8-14**.



After adding the **Rotary Motor** select the **Calculate Simulation** button and wait for the simulation to be completed. By default, your animation will last 5 seconds. If instructed by your lab instructor, show your animation by pressing the Play and Stop buttons in the Animation toolbar.

After calculating the animation, you can replay it using the **Normal**, **Loop** or **Reciprocate** options.

The last step to finish your lab is to make an assembly drawing with an exploded view. Add a shaded Isometric and print a copy for your lab instructor, as shown in **Figure 8-15**.

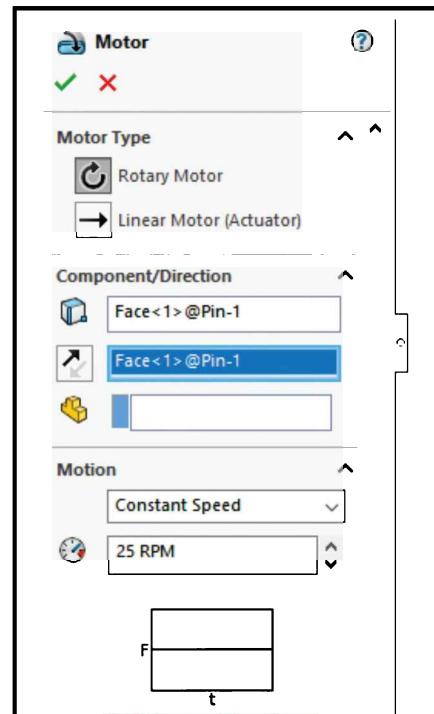


Figure 8-14. Rotary Motor Options.

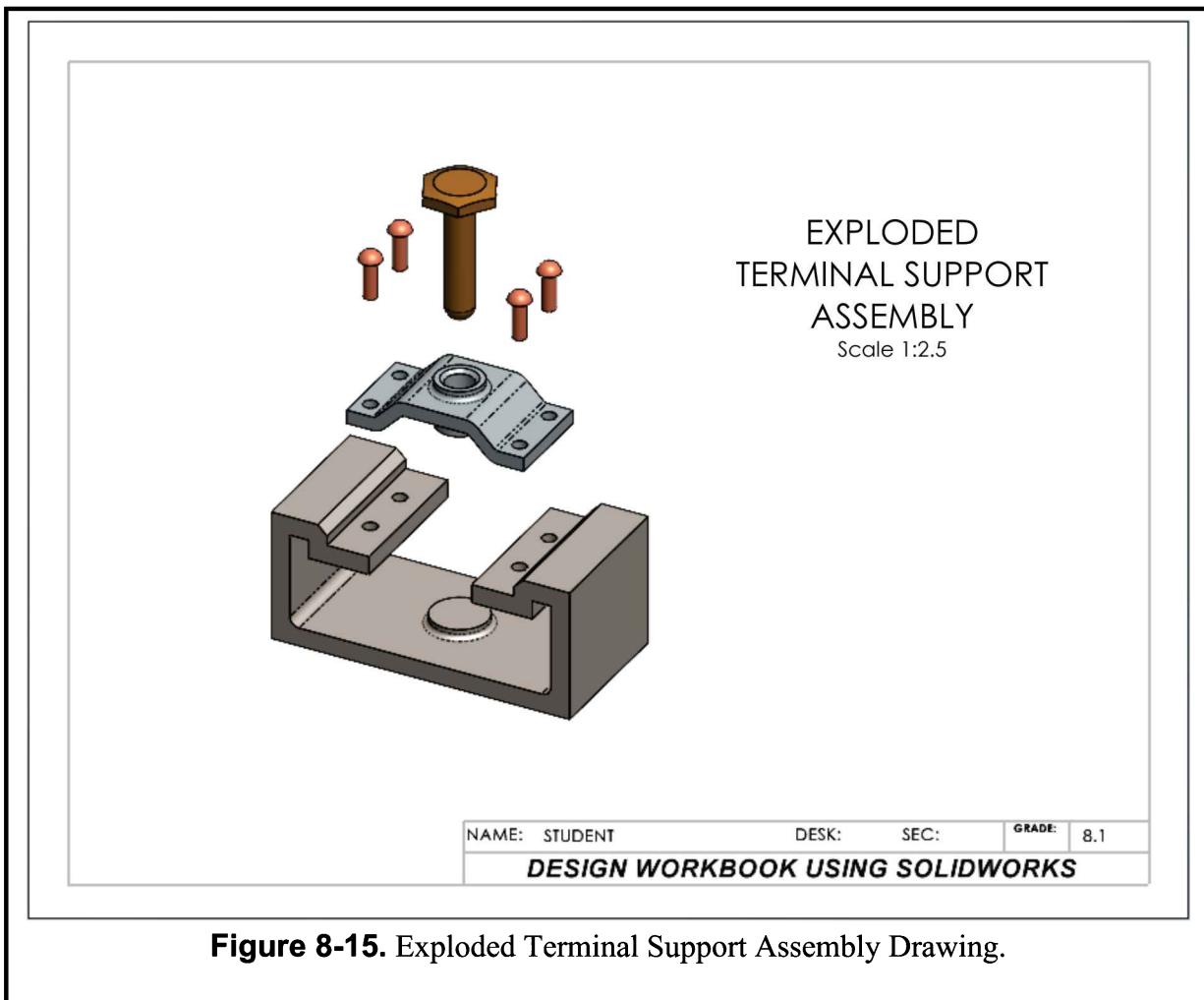


Figure 8-15. Exploded Terminal Support Assembly Drawing.

Exercise 8.2: Exploded Animation of the PULLEY ASSEMBLY

ASSEMBLY EXPLODED VIEW

In this lab you will make an exploded view of the previously made **Swivel Eye Block** assembly, make an animation, and save it as a video.



Open the **Swivel Eye Block** assembly and click on the **Exploded View** command from the Assembly tab.

The first component to explode is the **Big Rivet**. Select it in the screen, set the **Explode Direction** in the **Z** direction and enter a distance of **6.0"**. If needed, change the explode direction to make the **Big Rivet** go to the left, and click **Add Step** to continue to the next step, as shown in **Figure 8-16**.

Using the same steps, explode the **Pulley** in the **Y** direction down **6.00"**, change the direction to go down and click on **Add Step**.

To finish the exploded view, add the rest of the exploded steps as follows:

- Both **Small Rivets, 6.00"** to the left
- Front **Base Plate, 2.00"** to the left
- Back **Base Plate, 3.00"** to the right
- **Pulley, 3.00"** going down.
- **Eye Hook, 3.00"** going up.

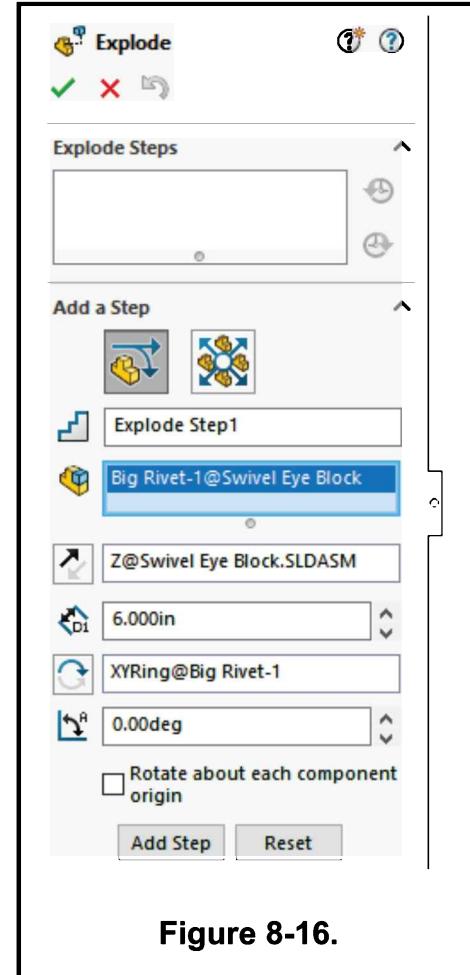


Figure 8-16.

In this exploded view the **Spacer** will remain in place and not move. When the exploded steps are completed click **OK** to finish the **Explode** command. Your exploded assembly should look like **Figure 8-17**.

EXPLODED VIEW ANIMATION



Now you will create an animation using the **Animation Wizard**. Activate the **Motion Study1** tab on the bottom left of the screen, and from the Animation toolbar click on the **Animation Wizard** command.

In the **Animation type** select the **Explode** option, click **Next** and set the **Duration** to **15 seconds**. Finally click **Finish** to create it.

PLAYING THE ANIMATION

In the Animation controller press **Calculate** or **Play** to generate the animation and play it, and evaluate the effect of the **Normal**, **Loop** and **Reciprocate** mode.

Show the animation to your lab instructor for confirmation and use the **Save As** command to save your assembly as **Exploded Pulley Assembly.sldasm**.

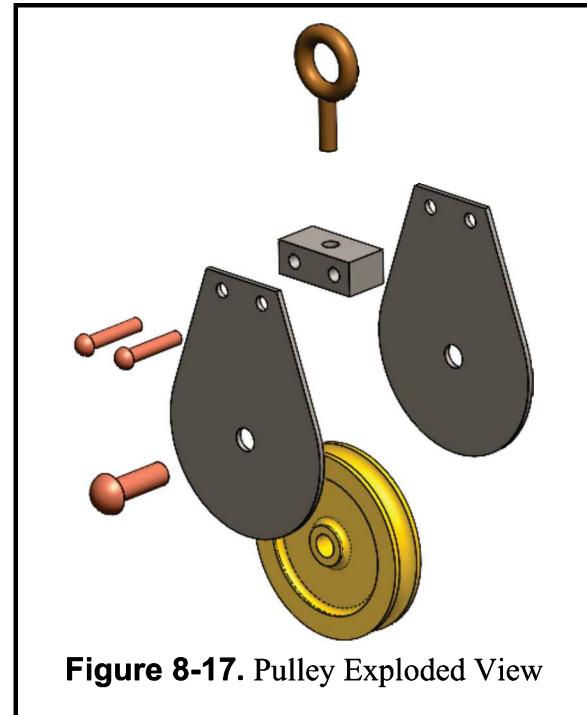


Figure 8-17. Pulley Exploded View

SAVE THE ANIMATION

From the Animation Controller select the **Save Animation** command. When the **Save Animation to File** dialog is presented, use the **.AVI** save type, set the animation to **15 seconds**, and click **Save**, as shown in **Figure 8-18**.

Use the default video compression setting with the default options and click **OK** to save it.

The animation will be played on screen at the same time it is being saved. Remember to adjust your screen size before saving the animation.

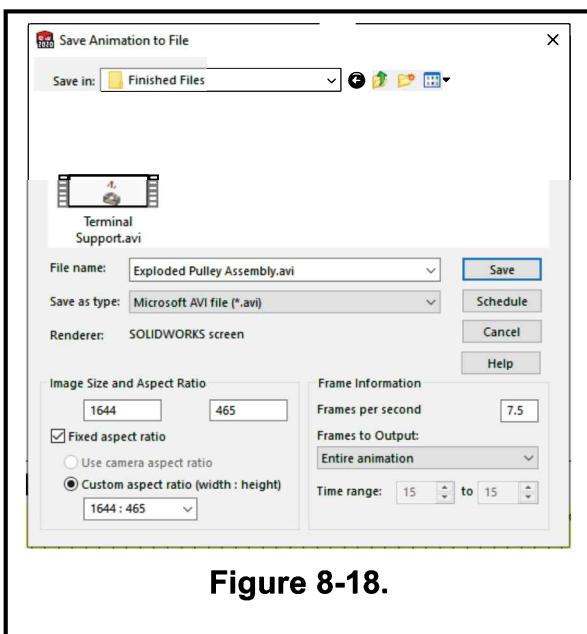


Figure 8-18.

ANIMATION OF PULLEY ASSEMBLY MOTION

In this step you will make an animation of the pulley using a rotating motor. Open the **Swivel Eye Block** assembly, and with an isometric view orientation, select the front Base Plate and from the context toolbar select the command to make **Change Transparency**. The reason to make the part transparent is to be able to see the pulley rotate.

Since the **Pulley** is a revolved feature and it does not have any feature that can make its rotation visible, you will add a texture to better visualize the motion.

In the **Task Pane** select the **Appearances, Scenes and Decals** tab as shown in **Figure 8-19**. Click and drag the **Texture** command and drop it on the **Pulley**. In the context toolbar shown immediately after dropping the texture, select the **Assembly** option.

This way the texture will only be added to the **Pulley** in the assembly, and not in the part file. Think of it as painting the part *after* it is assembled, instead of painting it *before* adding it to the assembly.

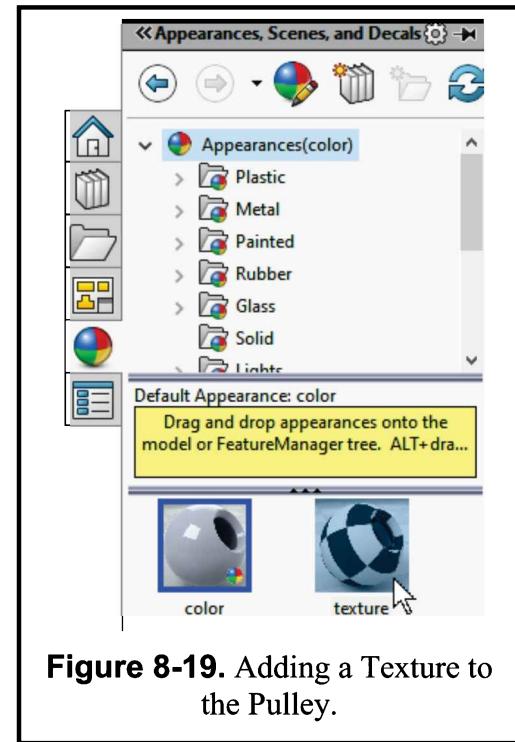
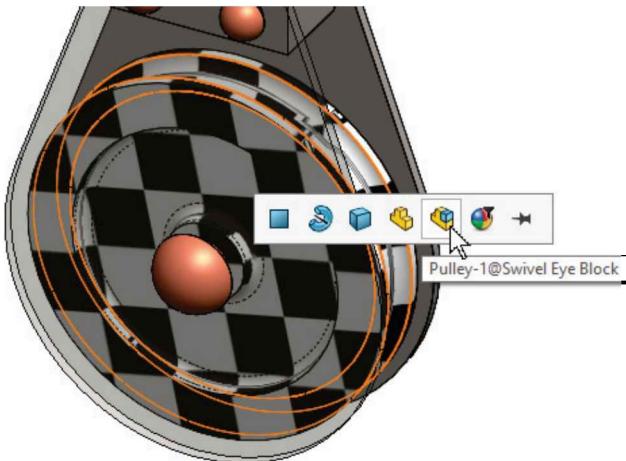


Figure 8-19. Adding a Texture to the Pulley.

Now select the **Motion Study1** tab on the bottom left of the screen and click on the **Motor** command from the Animation toolbar. Select the **Rotary Motor** option and click on an outer circular edge (or cylindrical face) of the **Pulley** to select the outside edge. Using the **Constant Speed** motion set the speed to **20 RPM** and click **OK** to add the motor.

Now click on the **Calculate** or **Play** command from the Animation toolbar to calculate the animation. Play the animation to show it to your lab instructor.

To finish this lab, make a new drawing of the Exploded Pulley Assembly, including a shaded Isometric view and print a copy for your lab instructor, as shown in **Figure 8-20**. Save your drawing as **Exploded Pulley Assembly.slddrw**.

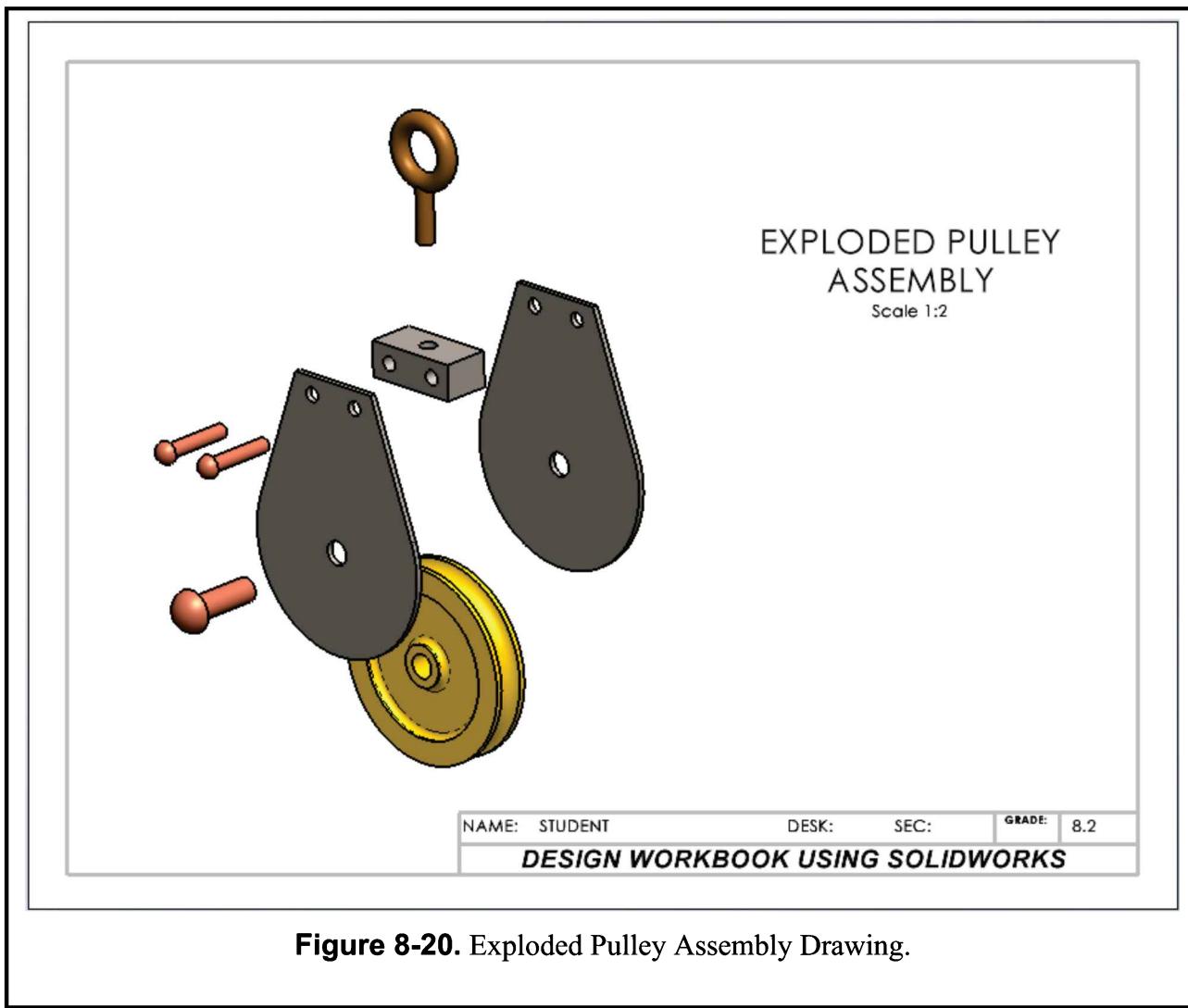
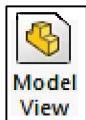


Figure 8-20. Exploded Pulley Assembly Drawing.

Exercise 8.3: Creating Component Drawing Views for Manufacturing

In this lab you will learn how to add multiple model views to a drawing, used to correctly document a part's features and dimensions for manufacturing. In this lab you will be working in the detail drawing environment.



Open the **Toe Clamp** part previously made in **Exercise 3.4**, make a new drawing using the **Inches** drawing template, and from the **View Layout** tab select the **Model View** command. Since the **Toe Clamp** part was already open it is listed in the Open Documents list. Click on the **Next** button to continue.

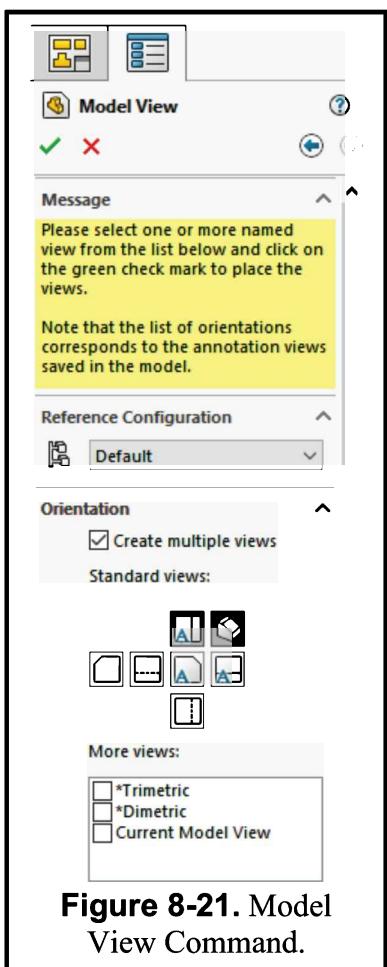


Figure 8-21. Model View Command.

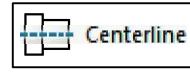
In the next page check the **Create Multiple Views** checkbox, select the **Top** and **Isometric** views, and click **OK** to add the views to the drawing sheet, as shown in **Figure 8-21**.

Now select the Isometric view, and change the **Display Style** to **Shaded with Edges**, set the scale to **User Defined** and enter **1:3**. Drag Locate the isometric view on the upper right side of the drawing.

Select the **Top** view, change the **Display Style** to **Hidden Lines Visible** and move it up in the drawing sheet.



From the **Annotation tab** select the **Center Mark** command. In the **Slot Center Marks** section select the **Slot Ends** option. Click in one of the slot's arcs and click **OK** to continue.



The next step is to add the centerlines. Click on the **Centerline** command, in the **Auto Insert** section check the **Selected View** option and click to select the **Top** view to add the centerline. Click **OK** to finish.

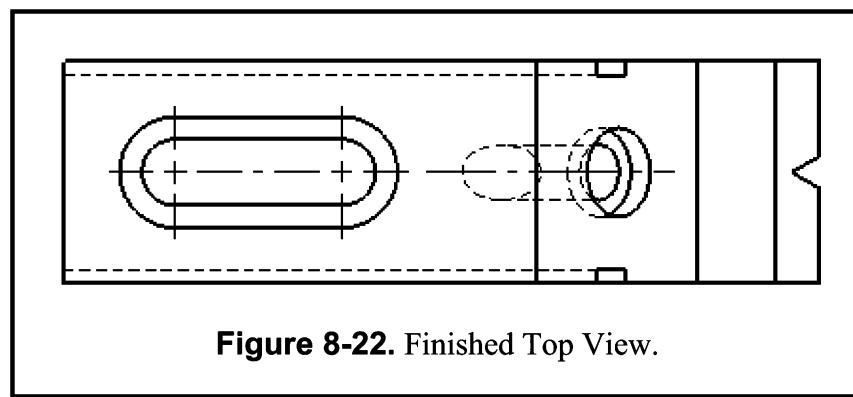
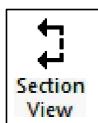


Figure 8-22. Finished Top View.

Section View



Now you will add a section view of the **Top** view. A section view is created by drawing a section line through a model to show internal details. From the **View Layout** tab, select the **Section View** command. In the properties select the Horizontal Section option as shown in **Figure 8-23**.

Now move your cursor to the **Top** view and touch either arc of the slot to add the section line. When the cutting plane snaps to the center of the slot, click to locate the section line.

As you move the cursor, a preview of the section will follow it. Move the section view below the **Top** view and click to locate it.

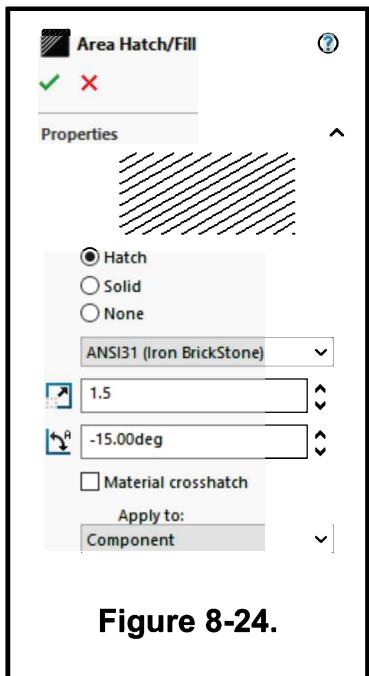


Figure 8-24.

Right click on the **Section A-A** view, and from the context menu select **Tangent Edge, Tangent Edges Removed**. To complete the section view, select the **Centerline** command, and add the centerlines to the view as we did in the **Top** view.

The cross hatching of section lines is not supposed to be parallel or perpendicular to model lines. To change the orientation of the hatching, click to select one of the section view's faces.

In the **Area Hatch/Fill** properties uncheck the **Material crosshatch** option, change the **Hatch Pattern** to **ANSI31 (Iron BrickStone)**, set the scale to **1.5** and the Pattern Angle to **-15°**, as seen in **Figure 8-24**, and click **OK**.

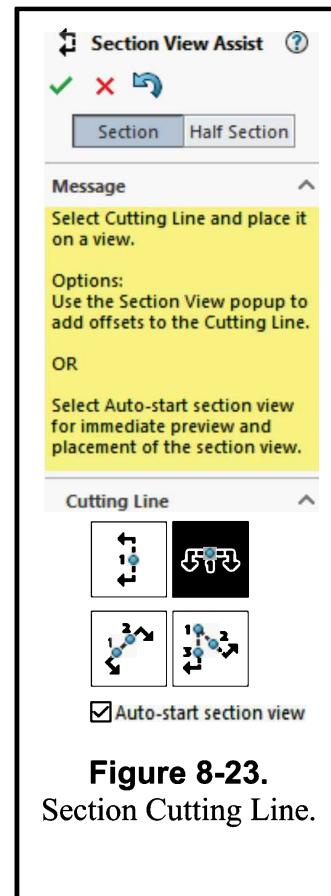
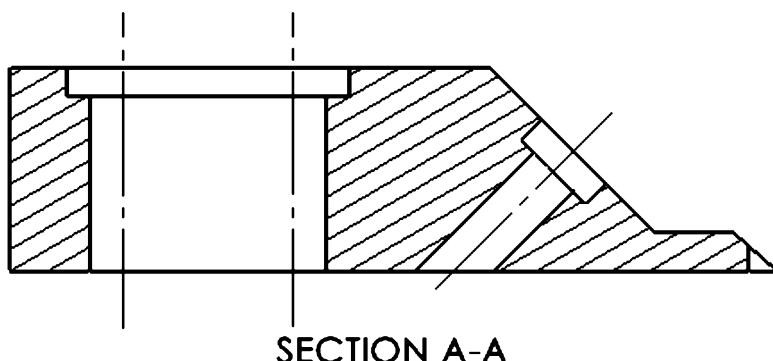
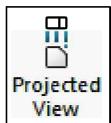


Figure 8-23.

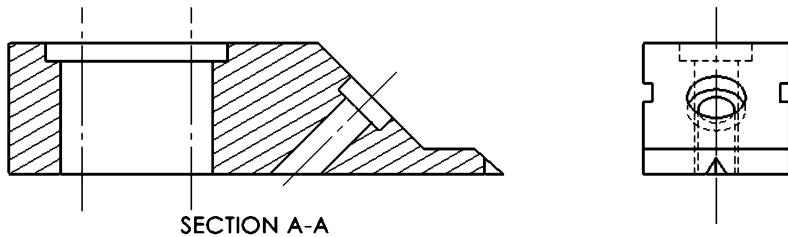
Section Cutting Line.



Projected View



The next view that you are going to add is a projected view to the right of the Section view. From the View Layout tab select the **Projected View** command. When asked to select a view from which to project, select the Section view. The projected **Right** view will be pre-viewed after you move the cursor to the right side. Locate the view next to the Section view and click **OK**.

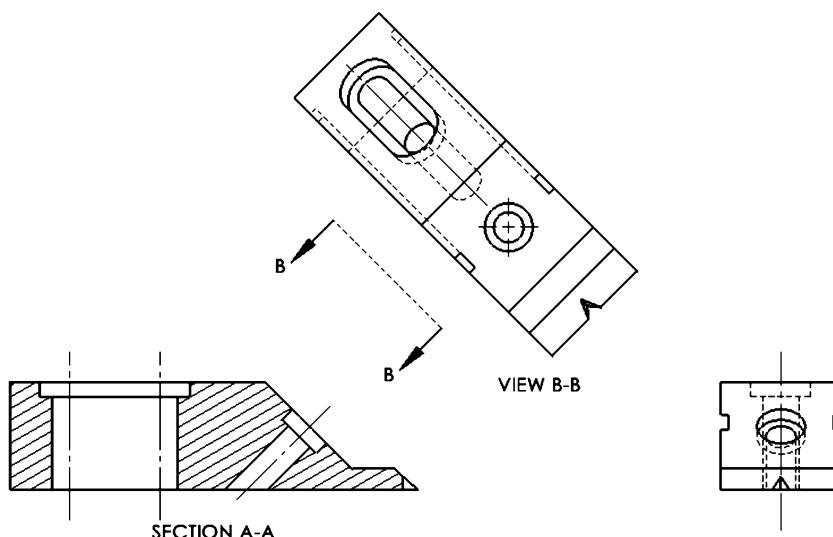


Since the **Right** view was projected from a view with **Hidden Lines Removed**, it will have the same **Display Style**. Select the **Right** view and change it to **Hidden Lines Visible**. To complete the view, add the **Centerlines** as before.

Auxiliary View



After inspecting the drawing views added so far, you will notice that the counterbore hole in the inclined face is not parallel to the page in any view. In order to correctly dimension the features in this face, you need to add an **Auxiliary** view. From the View Layout tab, select **Auxiliary View**. When asked to select a reference edge, select the inclined edge in the section view to make the auxiliary view parallel to this edge. After selecting the edge move your mouse pointer to locate the **Auxiliary** view. After locating the **Auxiliary** view, an arrow indicating the view's label and orientation is added. As you did before, add the **Centerlines** and change the view's display style to **Hidden Lines Visible**. You can also drag the label arrow to resize and locate as needed.



Detail View



When a model has small features that cannot be effectively dimensioned in the standard views, you need to create a **Detail** view, which enlarges a selected area to clearly show and dimension small features. From the View Layout tab select the **Detail** View command. The sketch **Circle** tool is automatically selected, and you are asked to draw a circle around the area to be enlarged. Draw the circle around the “V” cutout at the end of the **Top** view. When the circle is finished, locate the **Detail** view between the **Top** and **Section** views. A note with the view’s label and scale is automatically added. If the **Detail** view is too large or too small, you can select the **Detail** view and change its scale in the view’s properties. Your completed drawing should look like **Figure 8-25**.

Save your finished drawing as **TOE CLAMP VIEWS.slddrw** and print a copy to be turned in to your lab instructor.

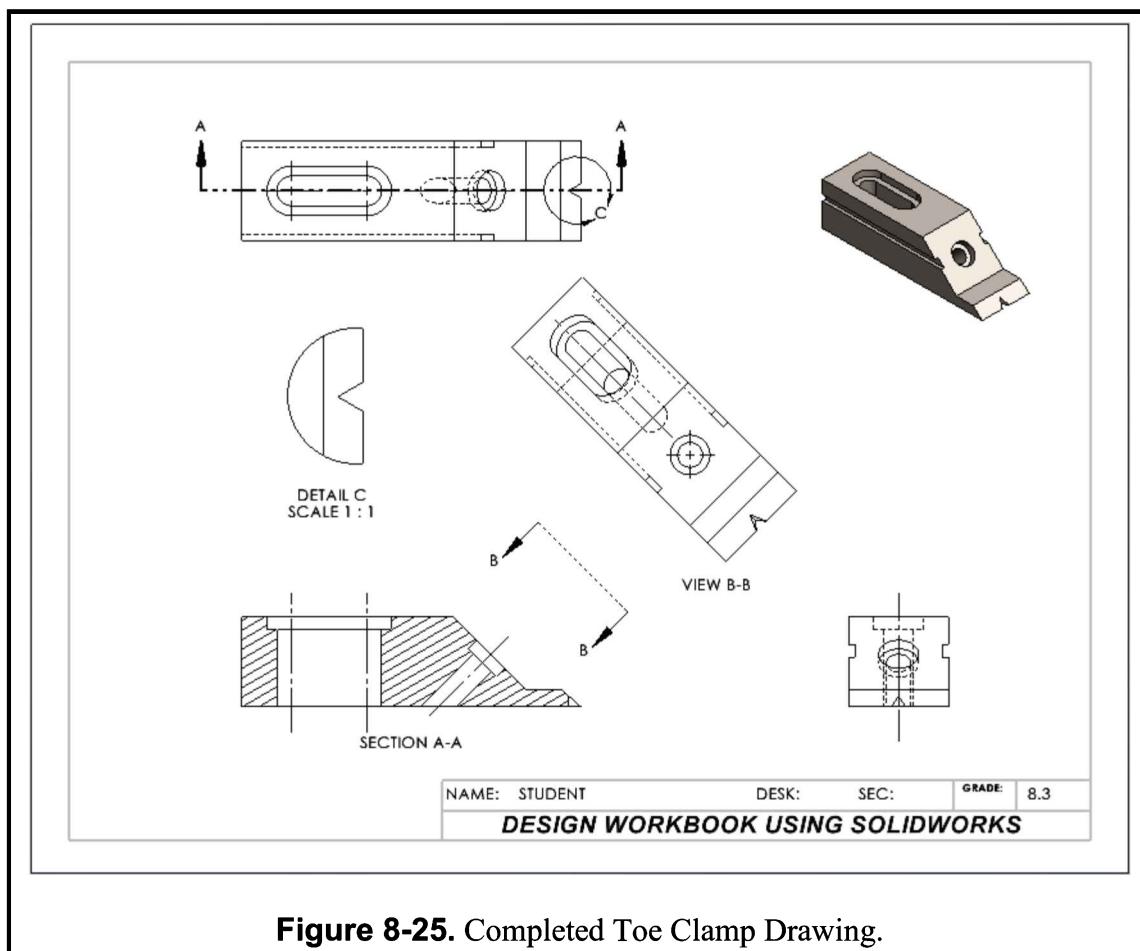


Figure 8-25. Completed Toe Clamp Drawing.

Exercise 8.4: Rapid Prototyping of a Solid Model Part

In this Exercise you will build a model assigned by your instructor and make the **.STL** file needed to make a physical prototype using a 3D printer. The **.STL** file extension is the standard input file format used in 3D printers and other rapid prototyping machines. **.STL** is an acronym for **Stereo Lithography**, the early process used to create rapid prototypes.

SAVE THE SOLID MODEL AS A .STL FILE

After you build your assigned model or open an existing one, select the menu **File, Save As**, and from the File Type option list select **STL** and save your file.

Before you can make your physical 3D printed part, depending on the type of equipment you are using, you will have to open the machine's software and prepare the file that will be sent to your specific machine.

In most 3D printers, commonly known as additive manufacturing, a stock of plastic material in the form of a thin filament is fed through a heated nozzle to melt it. The molten plastic is then deposited one thin layer at a time until the part is complete. The software to prepare the information used by the 3D printer to define the motion of the nozzle to "print" each layer is commonly known as "slicer" software, which reads the **.STL** file and 'slices' the model to define the trajectory of the nozzle to build each slice of the model.

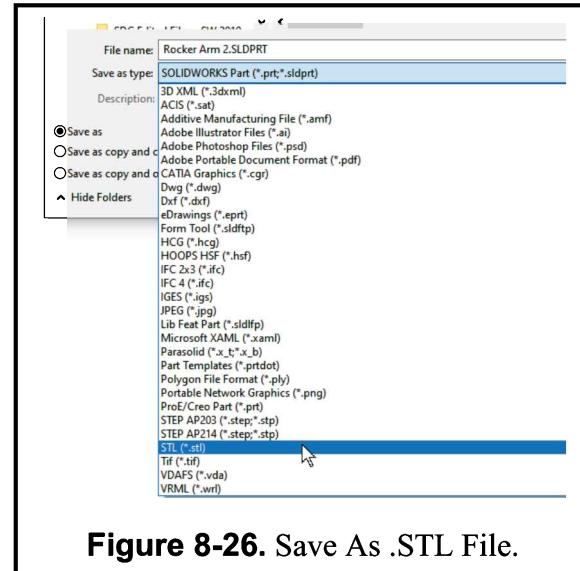


Figure 8-26. Save As .STL File.

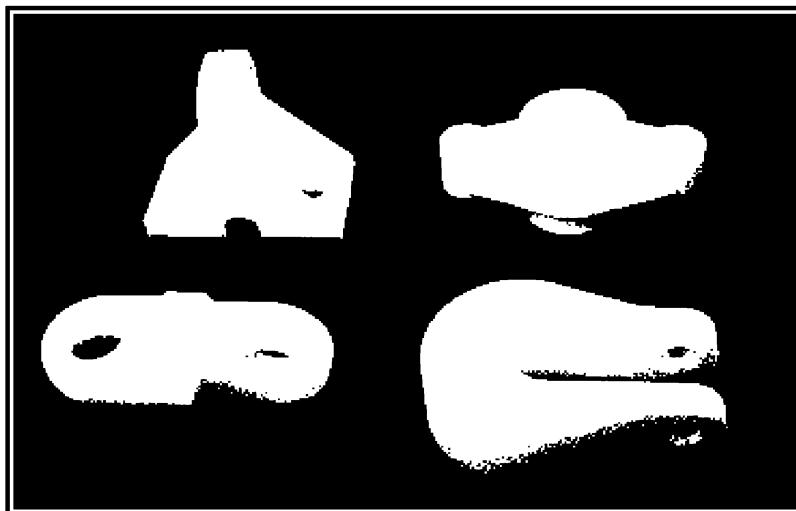
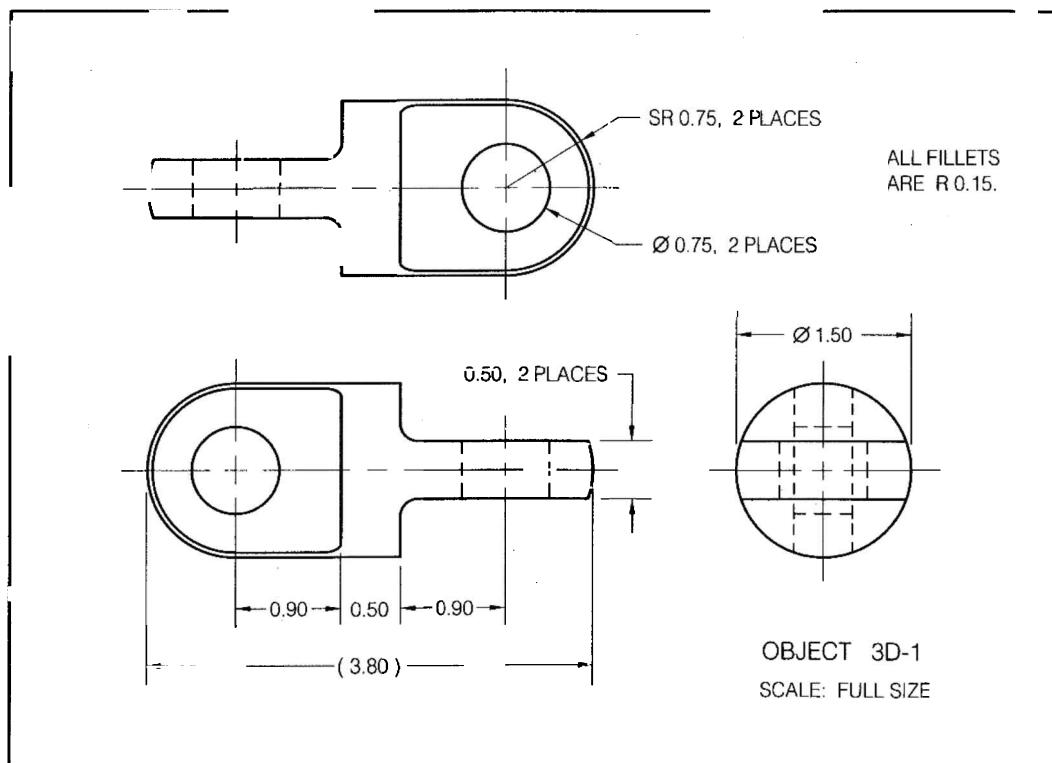
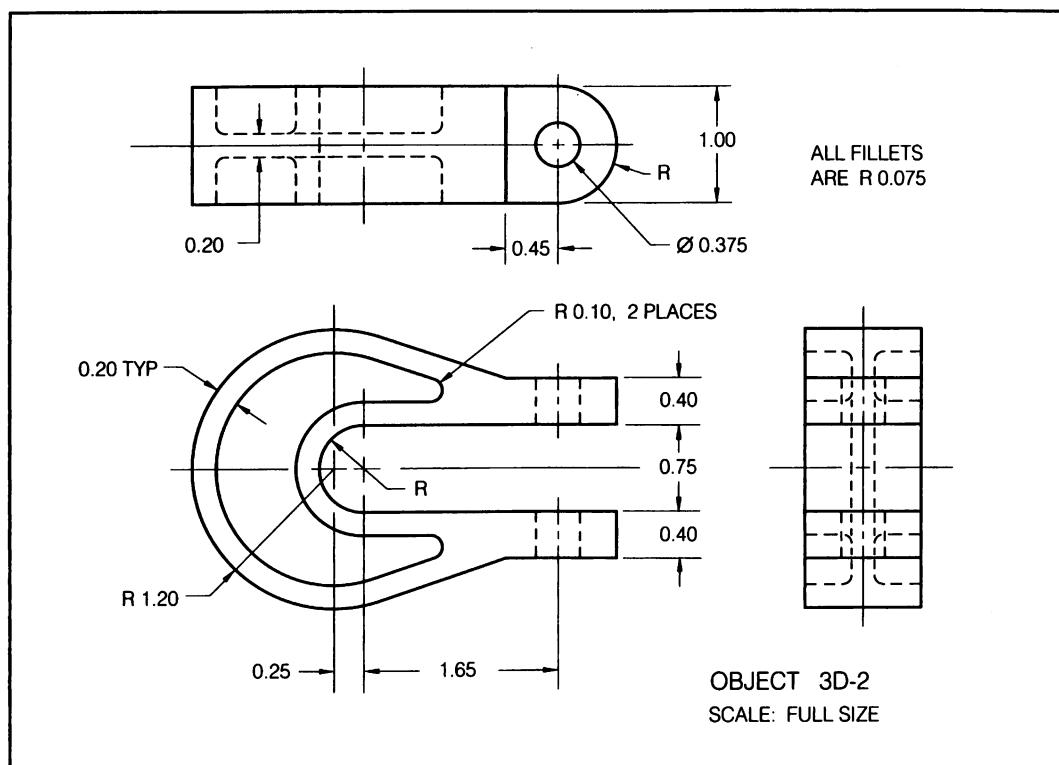


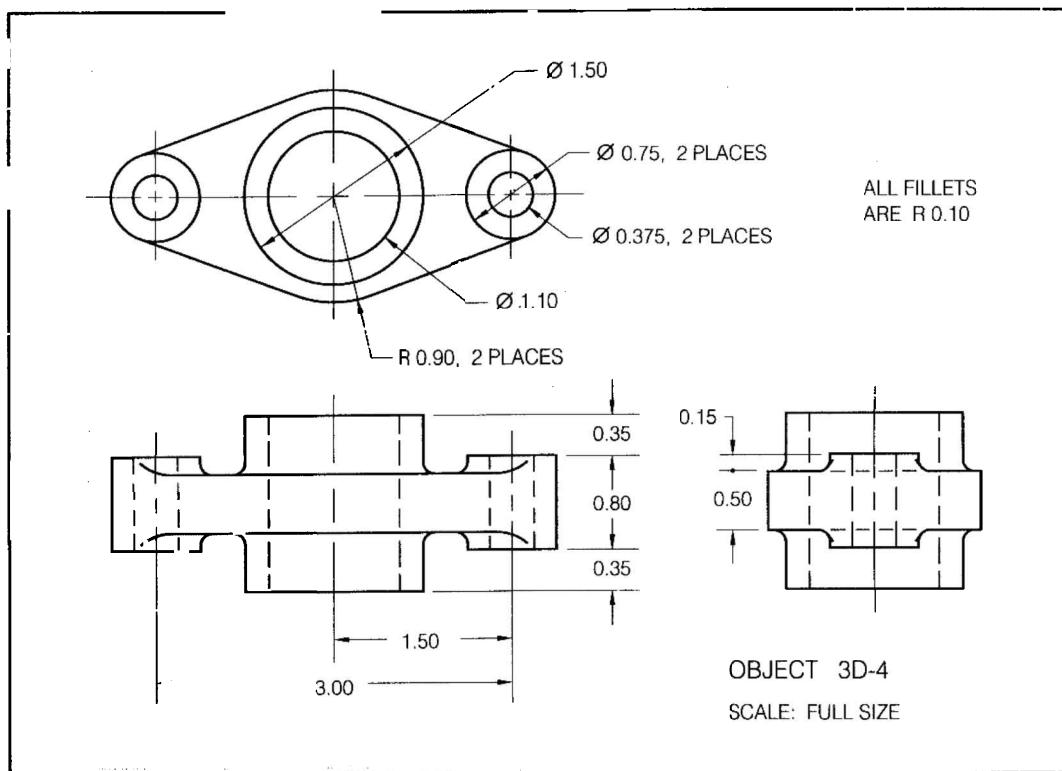
Figure 8-27. Rapid Prototype Models of Parts in Assignments 8.4.1 to 8.4.4.



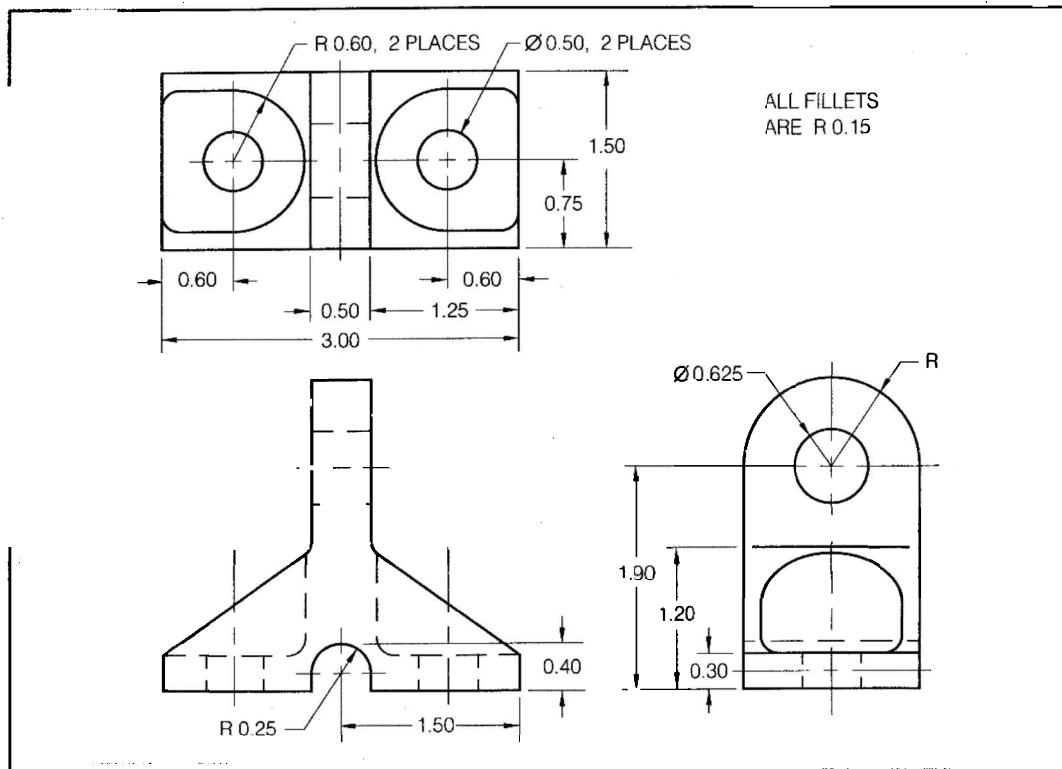
Solid Model Assignment 8.4.1



Solid Model Assignment 8.4.2



Solid Model Assignment 8.4.3



Solid Model Assignment 8.4.4

NOTES:

Design Workbook Lab 9: Section Views in 2D and 3D

In this lab you will learn how to make section views of solid models for inspection and visualization purposes, as well as how to use different options for drawing section views.

3D SECTION VIEWS



To visualize a 3D model's internal features, select the menu **View, Display, Section View**, or the **Section View** command from the View toolbar, as shown in **Figure 9-1**.

Your model will be cut using the last settings used, or if you have not used the **Section View** command in the current session, the **Front Plane** will be used as a default section plane, and the options will be displayed as shown in **Figure 9-2**.

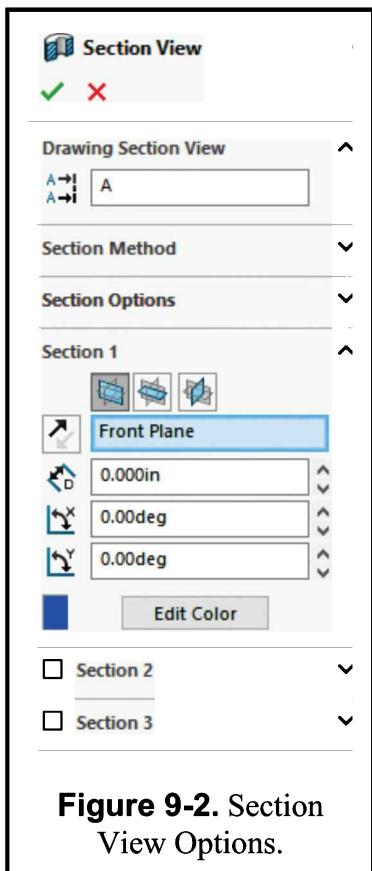


Figure 9-2. Section View Options.

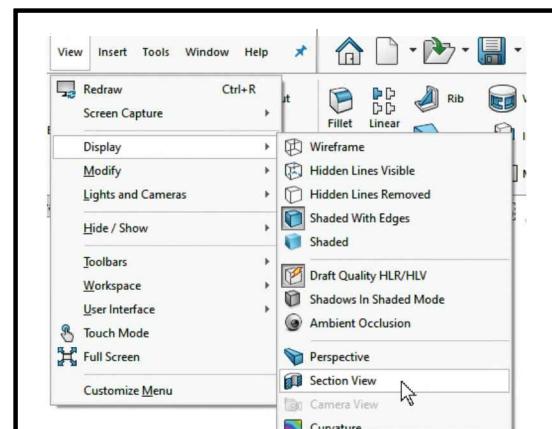


Figure 9-1. Section View Command.

In the **Section View** options you can modify the section plane's orientation, the view's direction, the **distance** from the origin, and **rotation** about the **X** and **Y** axis as needed to inspect internal features of your model.

Additionally, you can click-and-drag the section plane manipulator arrows directly in the screen, as shown in **Figure 9-3**.

If needed, you can use up to three different section planes by activating the **Section 2** and **Section 3** options.

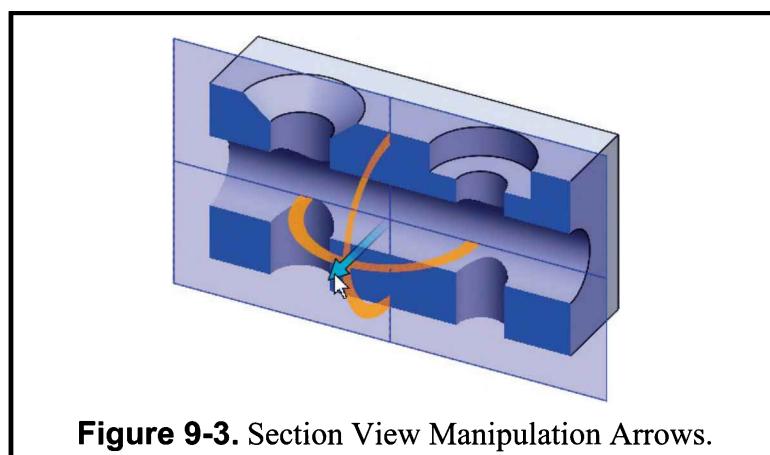


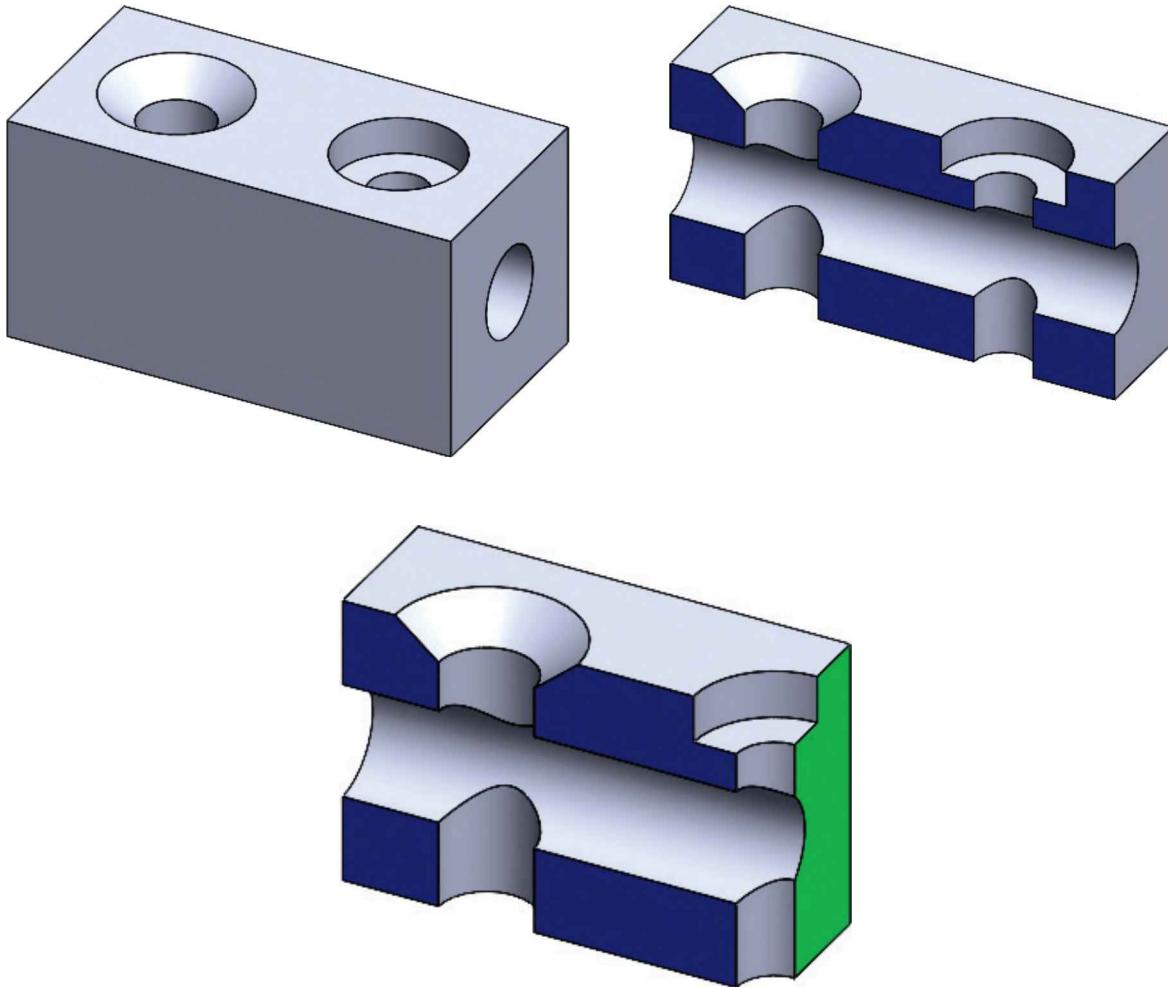
Figure 9-3. Section View Manipulation Arrows.

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You will see a dynamic preview of the section view as you make changes. If you click OK, the section view will remain visible and the Section View command will be activated.

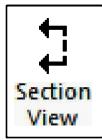
Remember the model has not been cut; it is only a temporary view used to measure and inspect hidden or otherwise difficult to see features of your design. To exit the Section View, select the menu **View, Display** and deselect the **Section View** command, or click on the Section View command again to restore the original 3D model view.



Section View using two section planes

2D DRAWING SECTION VIEWS

In the previous lab you learned how to add a section view in a drawing. In this lab you will learn additional options and types of drawing section views, as well as some advanced options.



After you select the **Section View** command from the View Layout tab, you can see the different options available, depending on the model and type of the section view needed. The section line arrows point in the direction of view. To reverse the Section View's direction, you can select the view and click on the **Flip Direction** button in the view's properties.

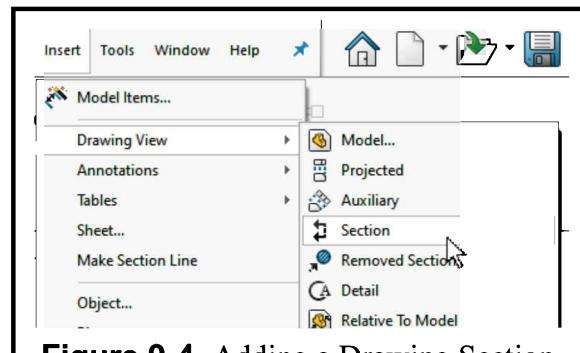
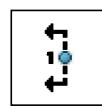
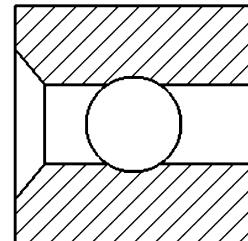
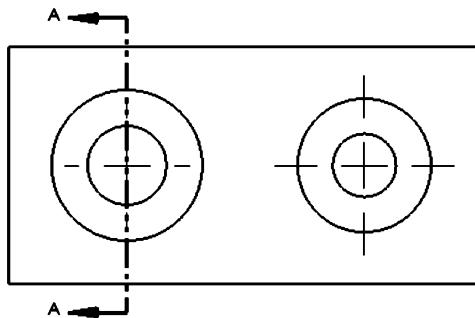


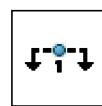
Figure 9-4. Adding a Drawing Section.



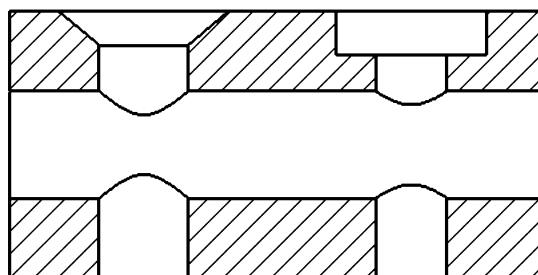
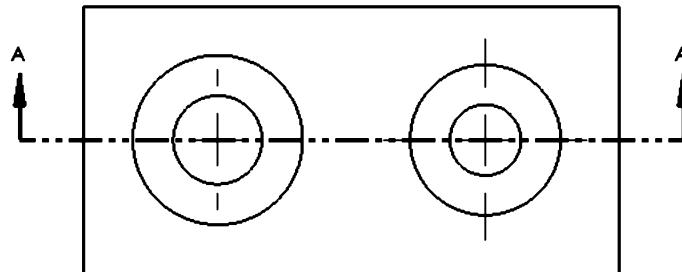
A **Vertical Section** view adds a vertical line to the view to be sectioned. After the section's line is located, you will see a preview of the section, and click to locate the new view.



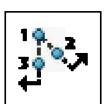
SECTION A-A



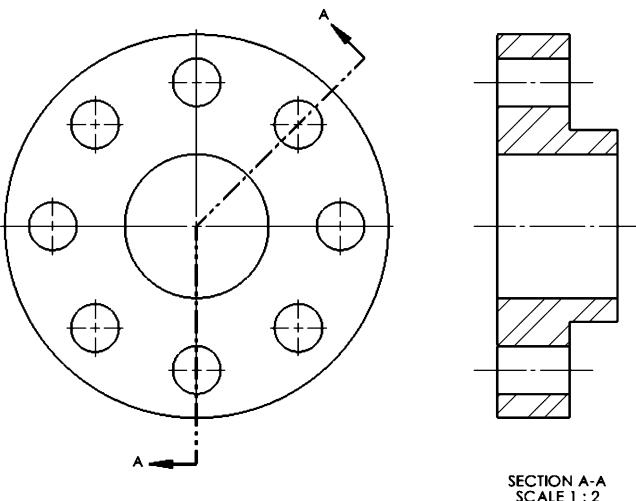
A **Horizontal Section** view adds a horizontal line to the view to be sectioned. After locating the section line, the new section view is created.



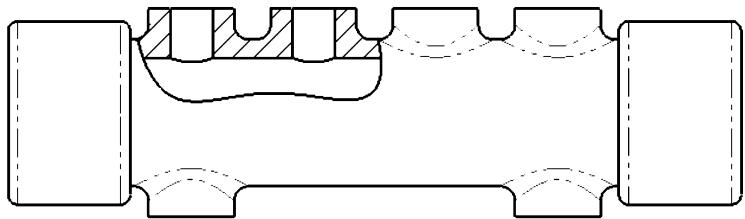
SECTION A-A



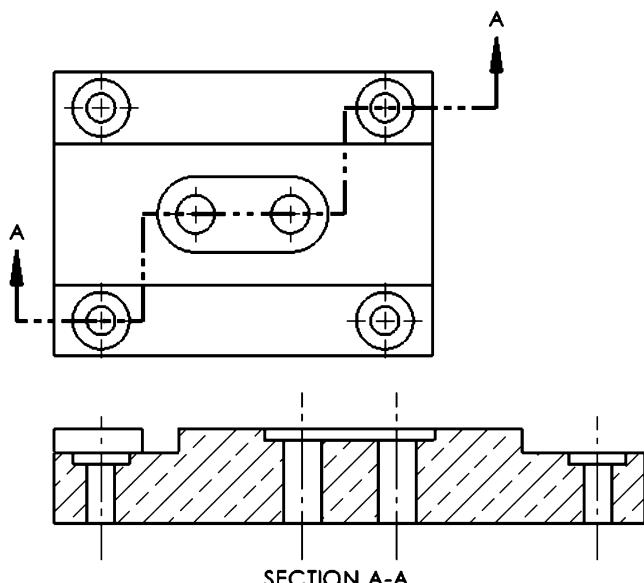
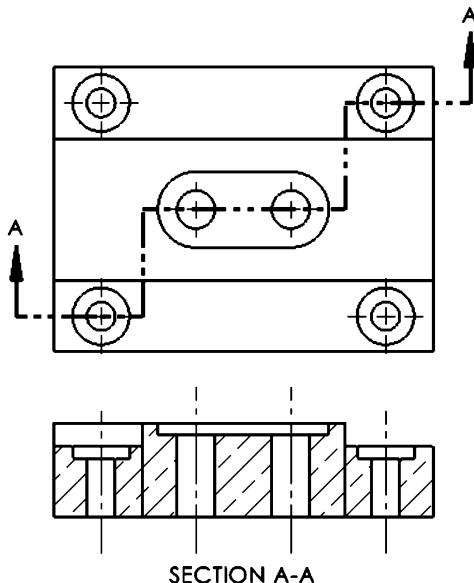
An **Aligned Section** view adds a cutting line made of two non-parallel planes to include features located at an angle. The angled section plane is then rotated and presented in the same orientation as the straight section plane.



A different type of view is the **Broken-Out Section**, which allows you to make a cutout of a main view to see internal features in the same view, without creating a new view. After selecting the **Broken-out Section** you are asked to draw a closed spline curve to define the area to cut out.



Another option is to add an **Offset Section** view. First you need to draw the connected section lines, and then click on the Section Line command. You will be asked to make a **Foreshortened** section normal to the cutting line, or a **Standard** section line, in which the cutting line is unfolded.



Foreshortened Section View

Standard Section View

Exercise 9.1: ROD BASE SECTION VIEW

In Exercise 9.1 you will build the **Rod Base** part using commands learned in previous labs. You will make a 3D section view of the model to visualize the internal features of the model, and then create a 2D drawing with a Top, Isometric and a Section view. To get started, you will build the solid model first.

MAKE THE ROD BASE

Start a new part using the template in **Inches**, add a new sketch in the **Front Plane** and draw the profile in **Figure 9-5**. When finished, use the **Revolved Base** command, and make a **360°** revolution.

To finish the part, add a **0.125" x 45° Chamfer** to the top inner edge, and the three **0.0625"** fillets indicated in **Figure 9-6**.

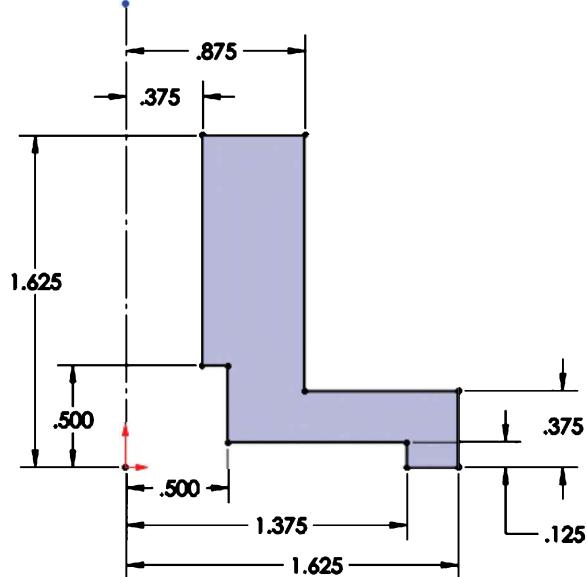


Figure 9-5. Sketch for the Rod Base.

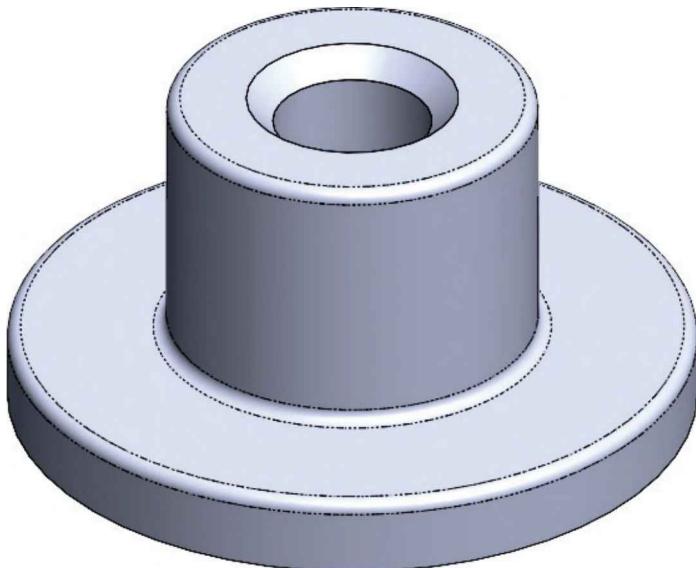
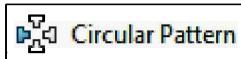


Figure 9-6. Part with Chamfer and Fillets.

Change to a **Top** view, add a sketch on the flat face and draw the profile indicated.

Add a **2.25"** diameter circle in the origin. Change it to **Construction** geometry and use it to locate the **0.25"** hole and make a Through All **Extruded Cut**.



Next, use the **Circular Pattern** command to make **4** equally spaced copies of the hole. Your part will look like **Figure 9-9**.

The last feature will be a small hole that goes through one side of the top cylindrical face. Change to a **Right** view, add a new sketch in the **Right Plane** and draw the profile shown in **Figure 9-8**.

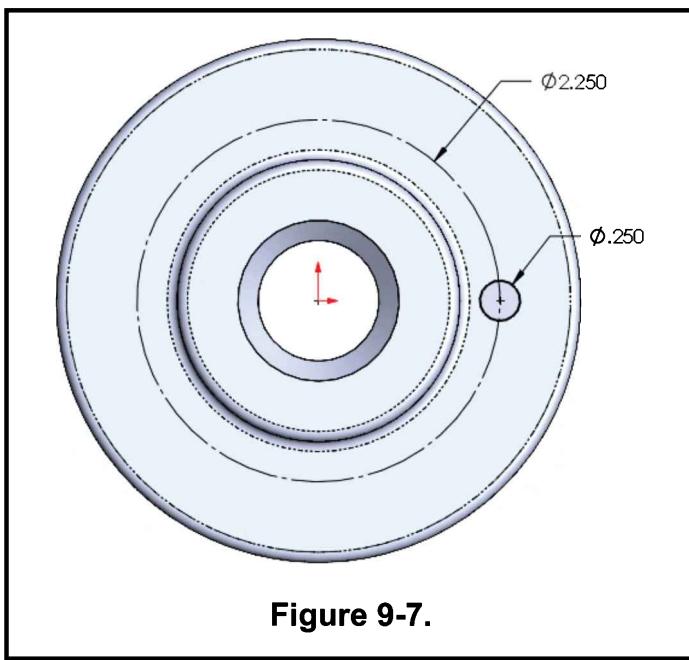


Figure 9-7.

Draw and dimension the circle, and to fully define your sketch add a **Vertical** relation between the center of the circle and the Origin. Change to an Isometric view and make an **Extruded Cut** using the Through All end condition to the right side of the part.

Set the part's material to **Manganese Bronze** from the **Copper Alloys** library and save your file as **Rod Base.sldprt**.

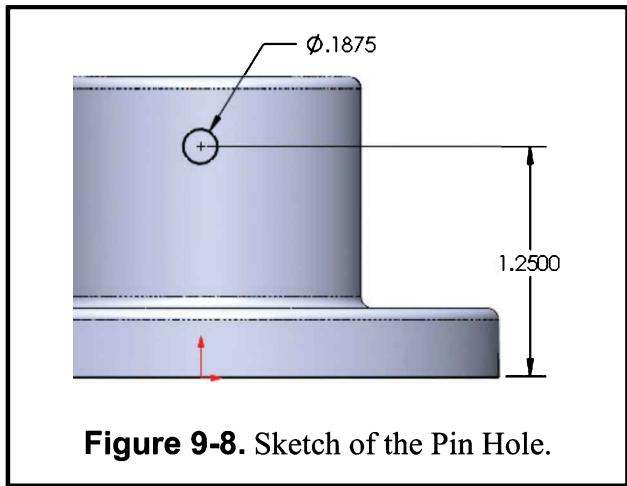


Figure 9-8. Sketch of the Pin Hole.

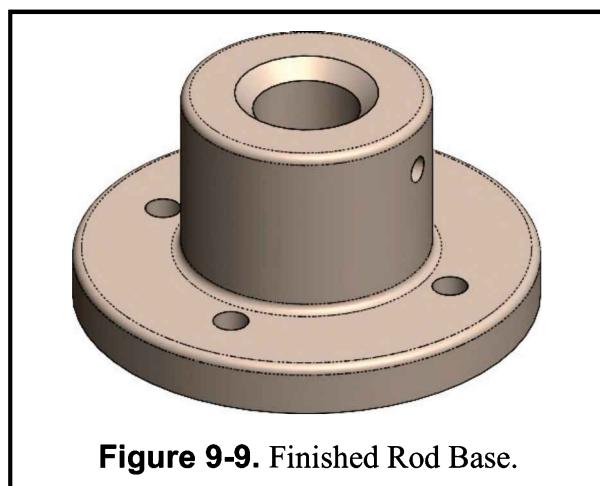


Figure 9-9. Finished Rod Base.

3D SECTION VIEW OF THE ROD BASE



You will now make a 3D section view of the **Rod Base**. Select the menu **View, Display, Section View**, or the **Section View** command from the **View** toolbar.

The Front Plane is the default option for Section 1. Leave the Offset Distance at **0.00"** and click **OK** to continue. If needed, click on the **Reverse Direction**  button to change the section view's orientation. Your sectioned model will look as shown in **Figure 9-10**.

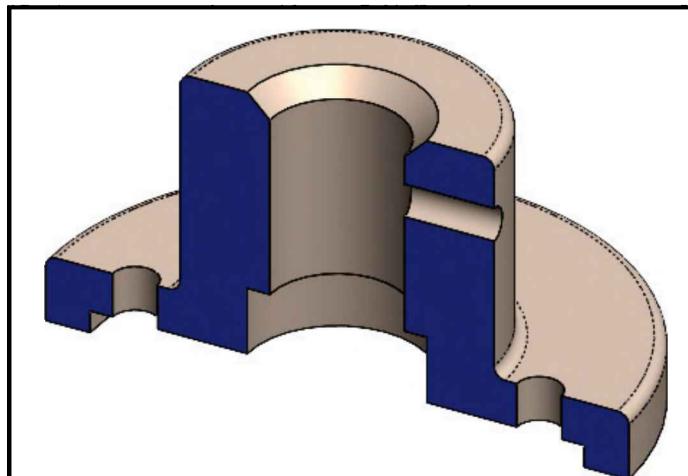


Figure 9-10. 3D Section View of the Rod Base.

When ready, click on the **Section View** command again to turn it off and save your model before making the detail drawing.

ROD BASE DETAIL DRAWING

Using the **Inches** template make a new drawing. Before adding drawing views, there are some system settings that can be changed to control the default options of new drawing views. Select the menu **Tools, Options, System Options**, and go to the **Drawings, Display Style** section, as shown in **Figure 9-11**.

In the **Display Style** section set the default to **Hidden lines visible**, and in the **Tangent Edges** section select the **Removed** option. Click **OK** to continue.

Remember, the settings listed in the **System Options** tab control the behavior of all documents, and the settings listed under the **Document Properties** tab only affect the current document.

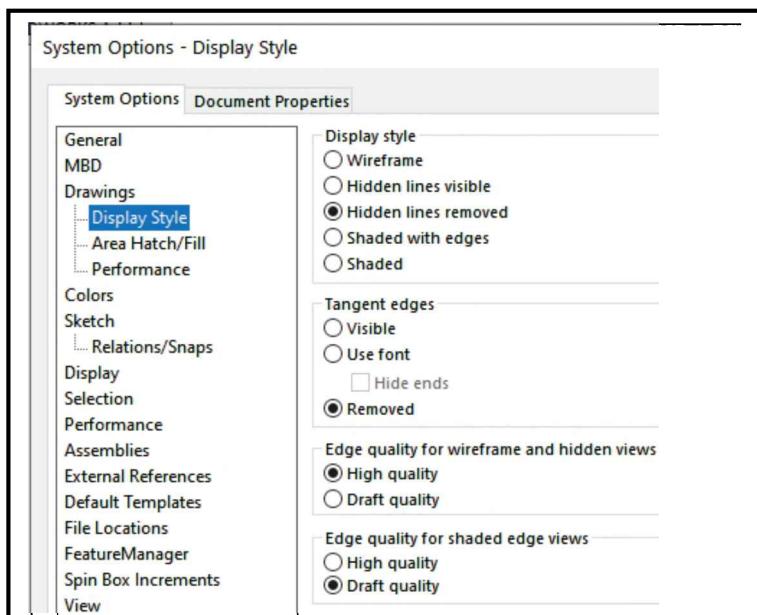


Figure 9-11. Drawing Display Style Options.

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After setting the system options go back to the drawing sheet and select the **View Palette** tab in the **Task Pane**. In the **View Palette** you can see a preview of each view orientation of the part currently open in **SOLIDWORKS**, or you can select a different part from the drop-down list at the top of the **View Palette**.

Add the **Front** view by dragging the **Front** preview to the lower left of the drawing sheet; since the “Auto-Start projected view” option is activated, immediately after the **Front** view is added, you can move the cursor around the first view to project additional views.

Move up and click in the sheet to add a **Top** view, then move up and right, and click to add an **Isometric** view, and click **OK** to finish projecting views.

Since the Front view will be replaced by a section of the **Top** view, select the **Front** view, and **delete** it. Your drawing should look like **Figure 9-12**. If needed, right click on **Sheet 1** in the FeatureManager and select **Sheet Properties**, change the sheet’s scale to **1:1** and click **OK**. This option will change the scale of all views in the drawing sheet.

Note: For a view’s scale to be controlled by the sheet’s scale, select a view, and make sure the option **Use sheet scale** is selected in the view’s properties.

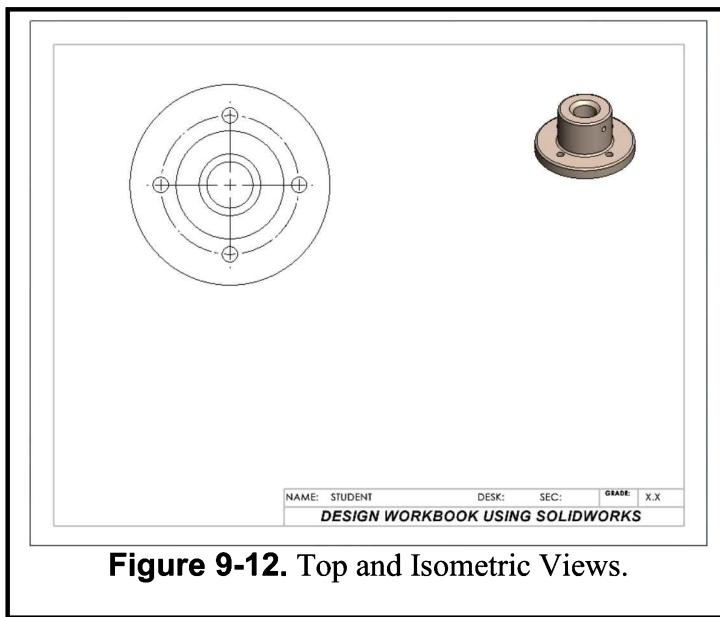
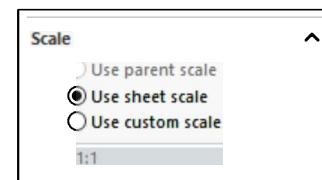
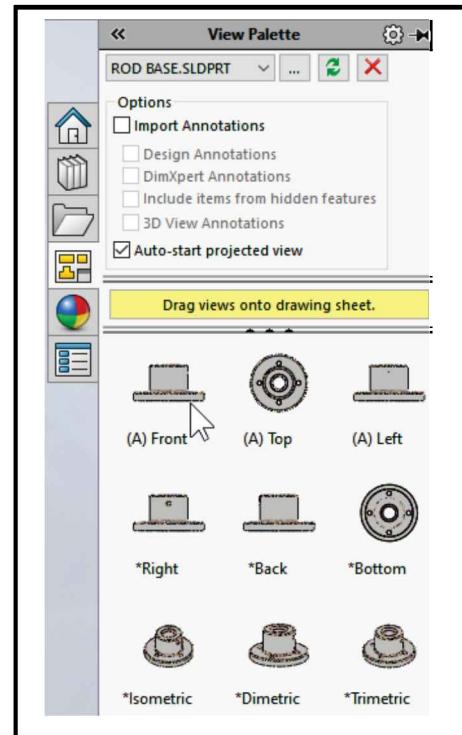
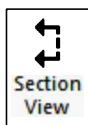


Figure 9-12. Top and Isometric Views.

Select the **Isometric View**, in its properties make sure the **scale** is set to **1:2**, and **Display Style** is set to **Shaded with edges** mode.

Right Click in the isometric view, select **Tangent Edge** and **Tangent Edges Visible**. Move the view to the upper right-hand corner of the drawing sheet. Now move the image to the upper left of the drawing sheet.

ADD A SECTION VIEW OF THE ROD BASE



To add a section view, select the **Section View** command from the View Layout tab. In the Section View properties select the **Horizontal** section line option. Move your cursor to the center of the **Top View**; when the **Section Line** snaps to the center, click to locate the section line, move your cursor down to where the Front view used to be, and click to finish the command.

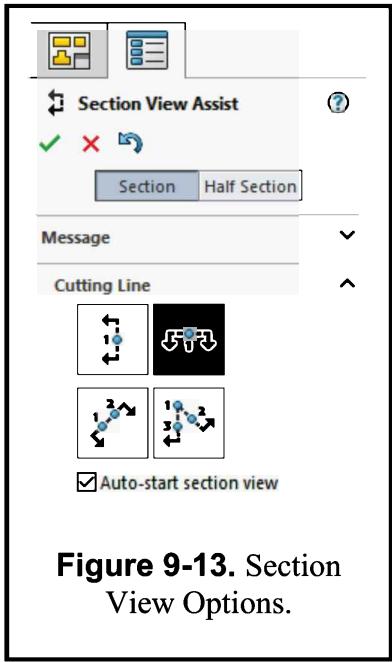


Figure 9-13. Section View Options.

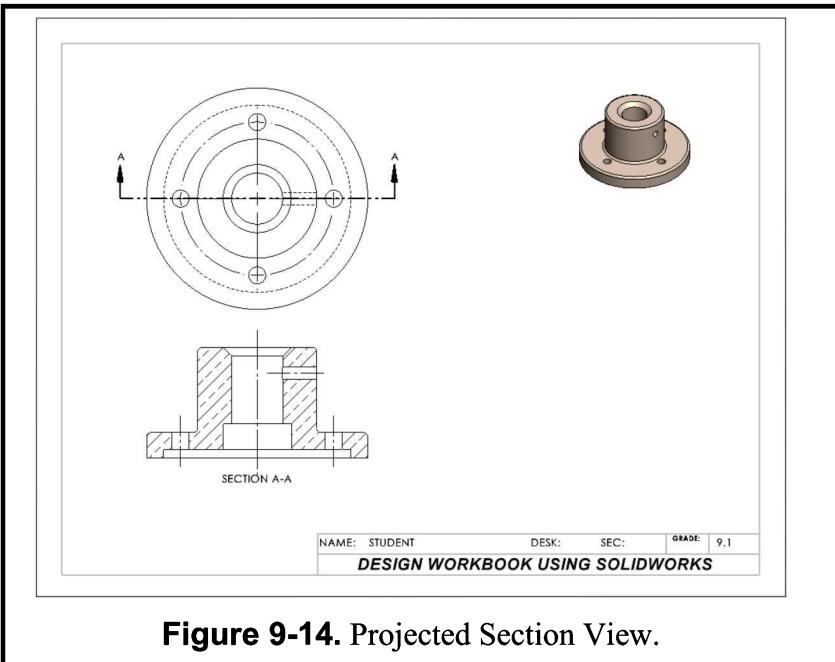


Figure 9-14. Projected Section View.

Notice the section view is aligned vertically with the top view, and parallel to the section line. The section line is automatically labeled **A-A**, and the section view is also labeled **Section A-A**, as indicated in **Figure 9-14**.

If the section line arrows are pointing down, select the section view and click in the **Flip Direction** button. Also, if the section view has the **Hidden Lines Visible** display style, change it to **Hidden Lines Removed**. Finally, select the **Centerline** command from the Annotations tab, in the Auto Insert section activate the option **Select View** and select the section view to add centerlines.

Now you need to add a Right view. From the View Layout tab, select the **Projected view** command. Select the section view, move to the right to project the **Right** view, and click to locate the view on the sheet. By projecting the view, the **Right** view will be aligned horizontally with the section view and inherit its display style properties.

Change the **Right** view's display style to **Hidden Lines Visible**, add the view's centerlines and add a note to indicate the part's name as **ROD BASE**, the scale and material. Your finished drawing will look like **Figure 9-15**.

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When you are finished save your drawing as **ROD BASE.slddrw**, print a hard copy to submit to your instructor.

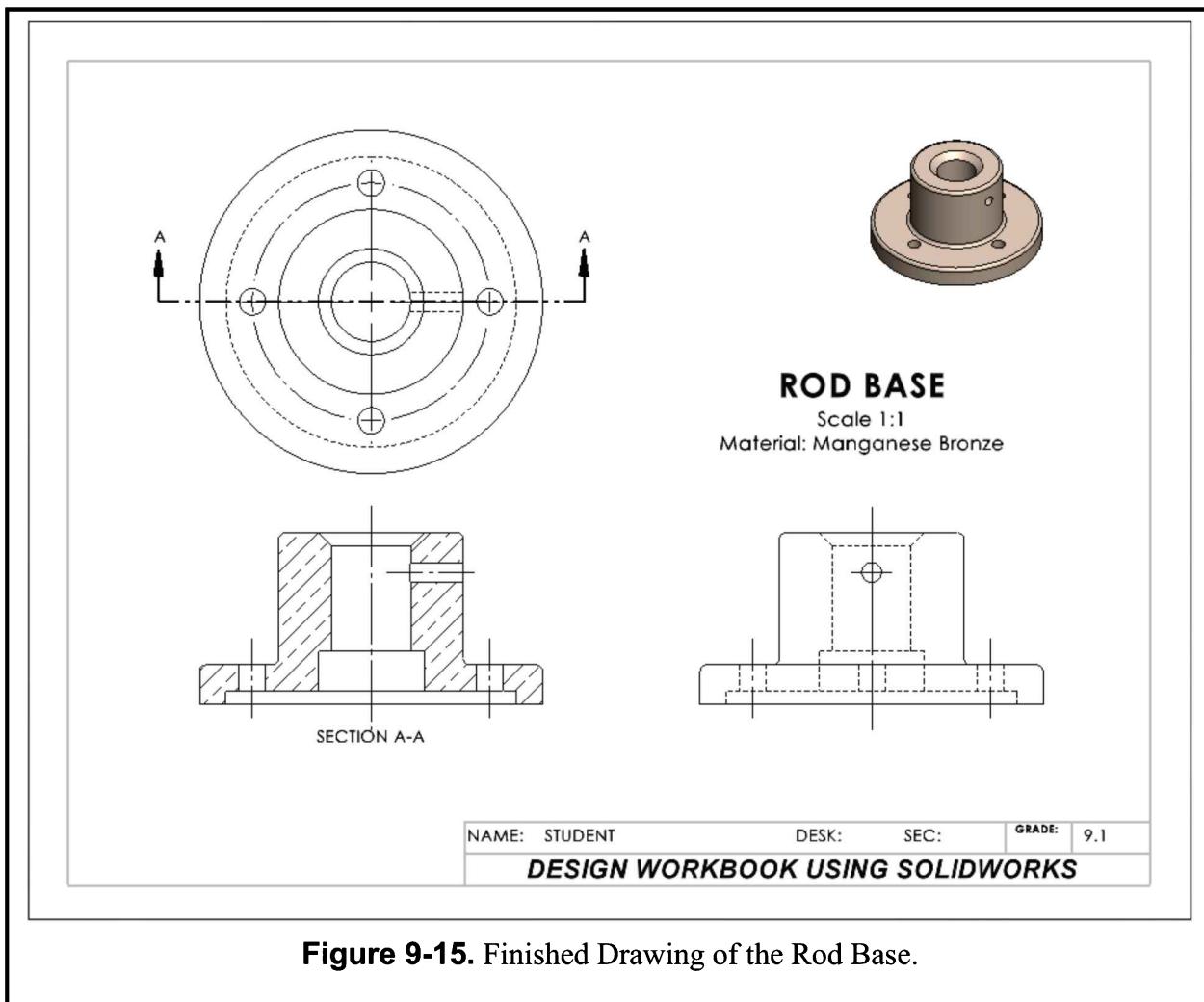


Figure 9-15. Finished Drawing of the Rod Base.

Exercise 9.2: TENSION CABLE BRACKET SECTION VIEW

In Exercise 9.2 you will build the **Tension Cable Bracket**. You will make a 3D section view of the model to visualize the internal features of the model, and then create a 2D drawing with a Top, Isometric and a Section view. To get started, you will build the solid model first.

BUILDING THE TENSION CABLE BRACKET

Start a new part using the **ANSI-INCHES** template, add a new sketch in the **Top Plane** and draw the profile shown in **Figure 9-16**. Extrude it upward **0.25"** and rename the feature "Base Plate."

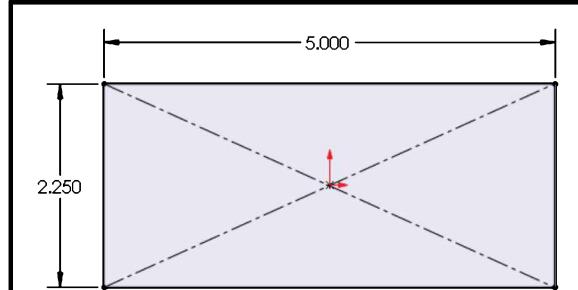


Figure 9-16. Initial Sketch Profile.

Now add a sketch in the top face of the Base Plate and draw the profile shown in **Figure 9-17**. Add a **Horizontal** relation between one hole and the origin to fully define the sketch. Use the **Dynamic Mirror** command to simplify your work.

Make an **Extruded Cut** using the Through All end condition, activate the **Draft** option and set the angle to **15°** to make two tapered holes.

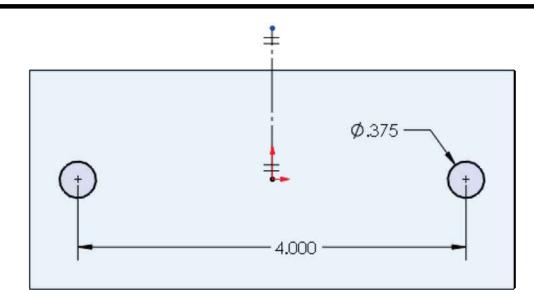


Figure 9-17. Sketch for Holes.

Now add a new sketch on the Top face. Draw a circle centered at the origin with a diameter of **1.75"**. Make an **Extruded Boss 0.875"** in Direction 1, and **0.875"** in Direction 2.

In the next step change to a Top view, add a sketch on the topmost face, draw a circle at the origin and dimension it **1.00"** in diameter.

Make an **Extruded Cut** using the Through All end condition, turn on the **Draft** option with **7.5°** and click **OK** to finish.

Change to a **Hidden Lines Visible** mode to see the tapered hole, as shown in **Figure 9-19**.

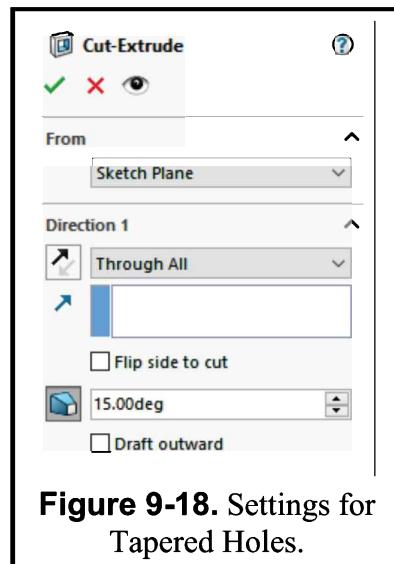


Figure 9-18. Settings for Tapered Holes.

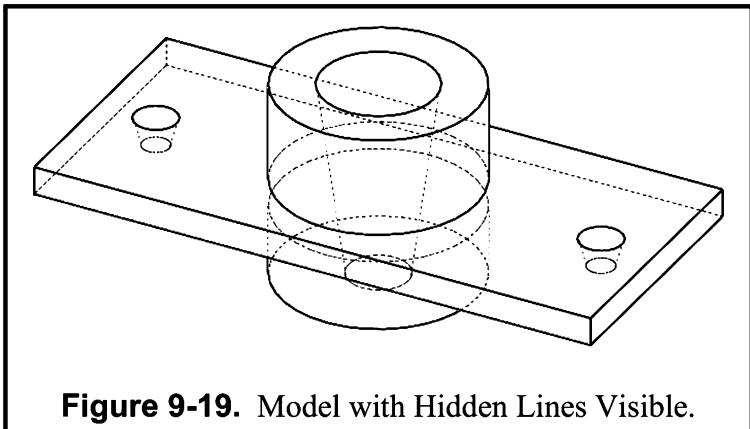


Figure 9-19. Model with Hidden Lines Visible.

Select the **Front Plane** in the first reference from the fly-out FeatureManager, set the offset distance to **0.75"** going to the front of the part and click **OK** to create the new plane.

Change to a **Front** view, select **Plane1** in the screen or the FeatureManager and add a new sketch on it.

Draw the profile shown in **Figure 9-20** using the **Dynamic Mirror** command and **Hidden Lines Visible** display mode.

Now make a **0.25"** extrusion using the **Blind** end condition going towards the center of the part.



Select the **Mirror** command from the Features tab and mirror the triangles about the **Front Plane**. Your part should now look like **Figure 9-21**.

In the next step you will add two triangular supports. These supports will be offset from the center, and you will need to add a reference plane.

Select the **Reference Geometry** command and click on **Plane** from the drop-down list.

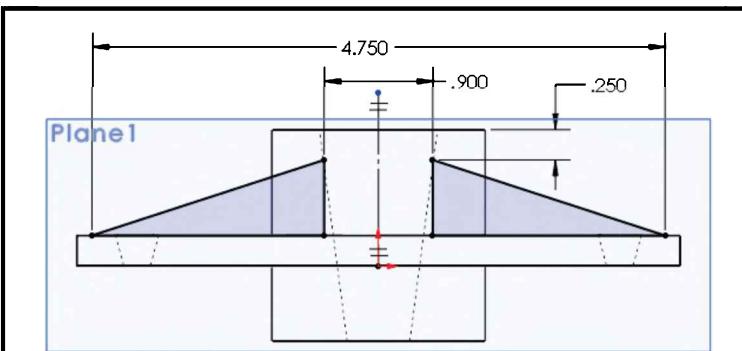


Figure 9-20. Triangular Support Sketch.

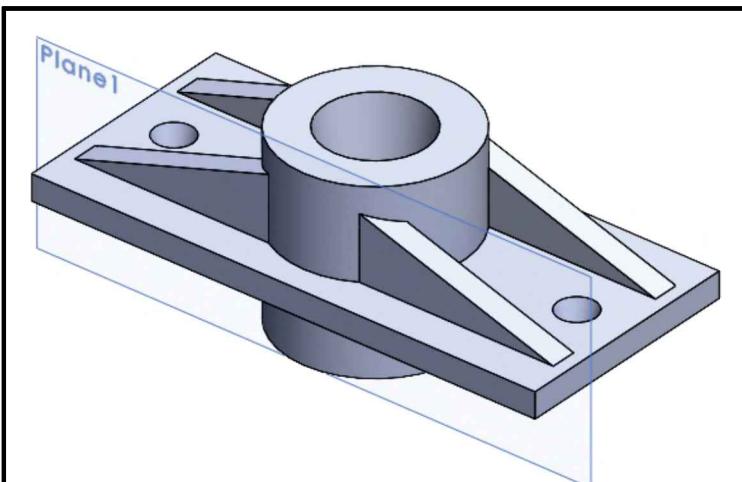


Figure 9-21. Triangular Supports.

To finish the Tension Cable Bracket, add a **Chamfer** to the top inside edge of the cylindrical boss. Make the chamfer using the **Distance-Angle** option and size it **0.125" x 45°**.

Next, add a **0.0825" Fillet** to the following edges:

- Outer top edge of upright boss.
- All intersections of the upright boss with the base plate, above and below the base plate.
- The top outer edges of the base plate.
- The top edges of the four triangular supports.
- The vertical edges of the rectangular base.

When finished adding the fillets, set the material to **Cast Carbon Steel** and save your part as **Tension Cable Bracket.sldprt**.

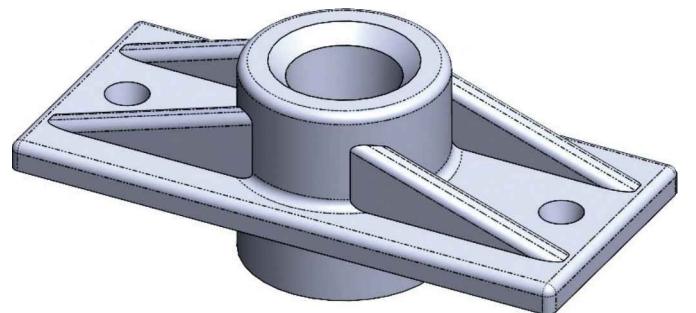


Figure 9-22. Finished Tension Cable Bracket.

3D SECTION VIEW OF THE TENSION CABLE BRACKET



You will now make a 3D section view of the **Tension Cable Bracket**. Select the menu **View, Display, Section View**, or the **Section View** command from the **View** toolbar.

The Front Plane is the default option for Section 1. Leave the Offset Distance at **0.00"** and click OK to continue. If needed, click on



the **Reverse Direction** button to change the section view's orientation. Your sectioned model will look as shown in **Figure 9-23**.

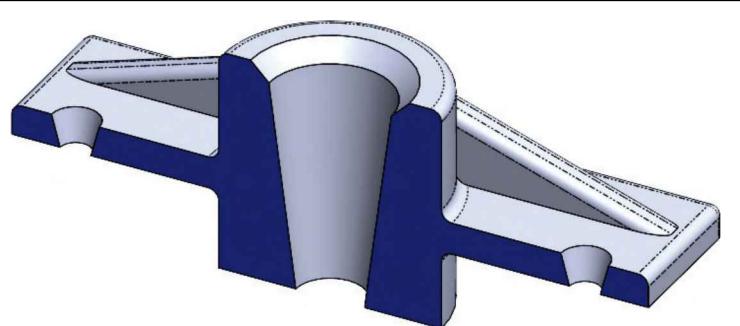


Figure 9-23. 3D Section View of the Cable Bracket.

When ready, click on the **Section View** command again to turn it off and save your model before making the detail drawing.

CABLE TENSION BRACKET DETAIL DRAWING

At this time, you will make the detail drawing of the **Cable Tension Bracket**. If you changed the **System Options** in exercise 9-1, those settings are already set, and you do not need to change them again. Remember that **System Options** affect all SOLIDWORKS documents and are not document specific.

Start a new drawing using the **Inches** template. Select the **View Palette** tab from the **Task Pane** on the right side of the screen and drag the **Top** and **Isometric** views separately onto the drawing, as shown in **Figure 9-24**.

Select the **Isometric** View and from the view's properties make sure the **Scale** is set to **1:2** and the Display Style to **Shaded with Edges** mode. Right click on the view and set the **Tangent Edges** to **Tangent Edges with Font**.

Now select the **Top view**, change its **Scale to 1:1** and change the Display Style to **Hidden Lines Visible**. Arrange the views to match the image on **Figure 9-24**.

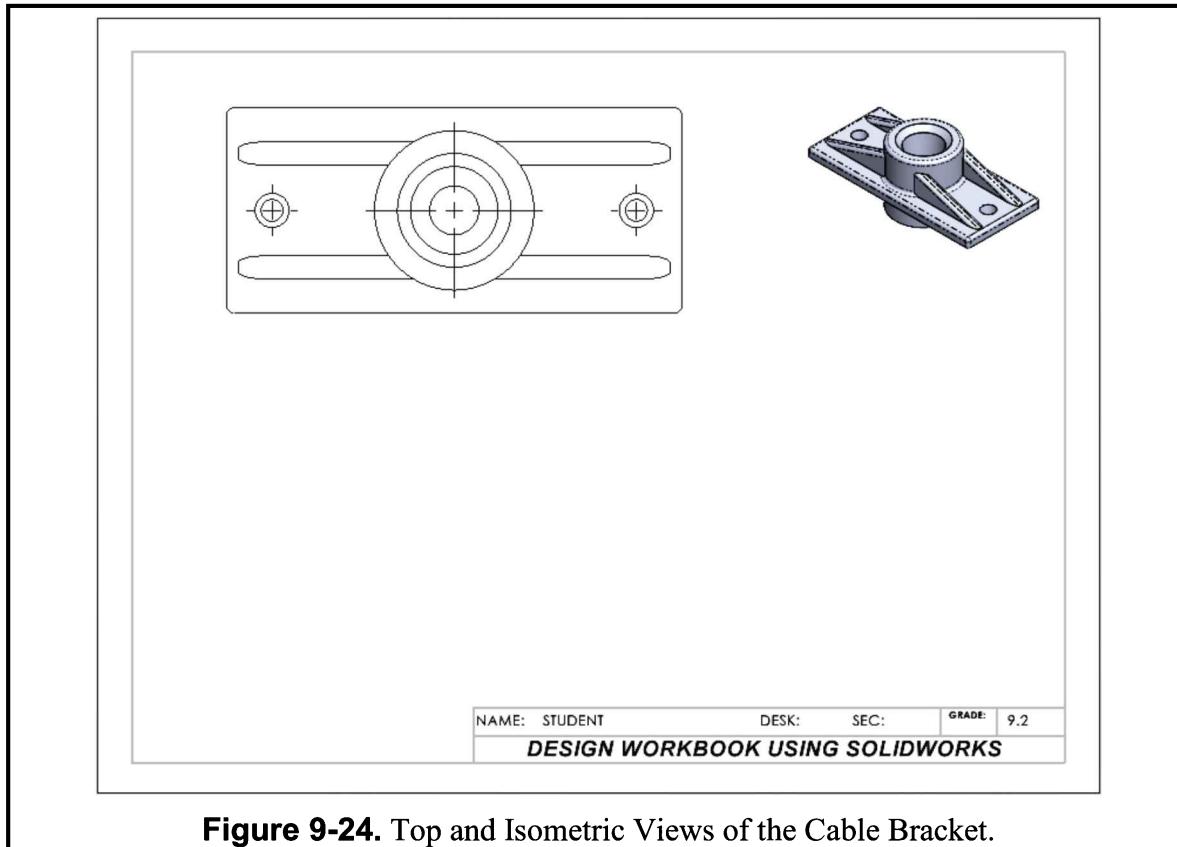
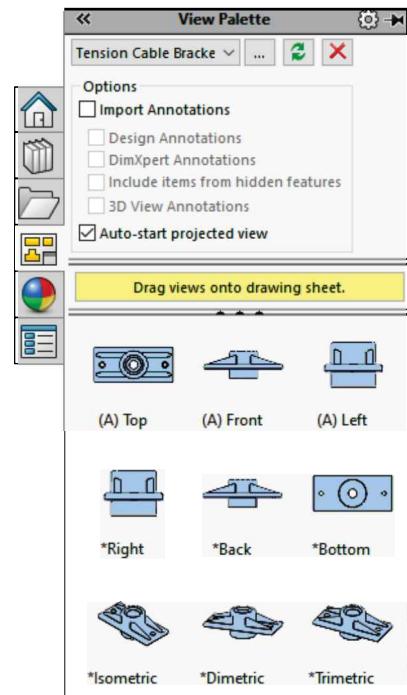
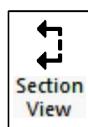


Figure 9-24. Top and Isometric Views of the Cable Bracket.

ADD A SECTION VIEW OF THE CABLE BRACKET

 To add a section view, select the **Section View** command from the View Layout tab. In the Section View properties select the **Horizontal** section line option. Move your cursor to the center of the **Top View**; when the **Section Line** snaps to the center, click to locate the section line, move your cursor down to place the section view in place of the **Front** view, and click to locate it. 

Remember the section view is aligned vertically with the top view, and parallel to the section line. The section line is automatically labeled **A-A**, and the section view is labeled **Section A-A**. If the section line arrows are pointing down, click in the **Flip Direction** button. Finally, use the **Centerline** command to add the missing centerlines to the Section View.

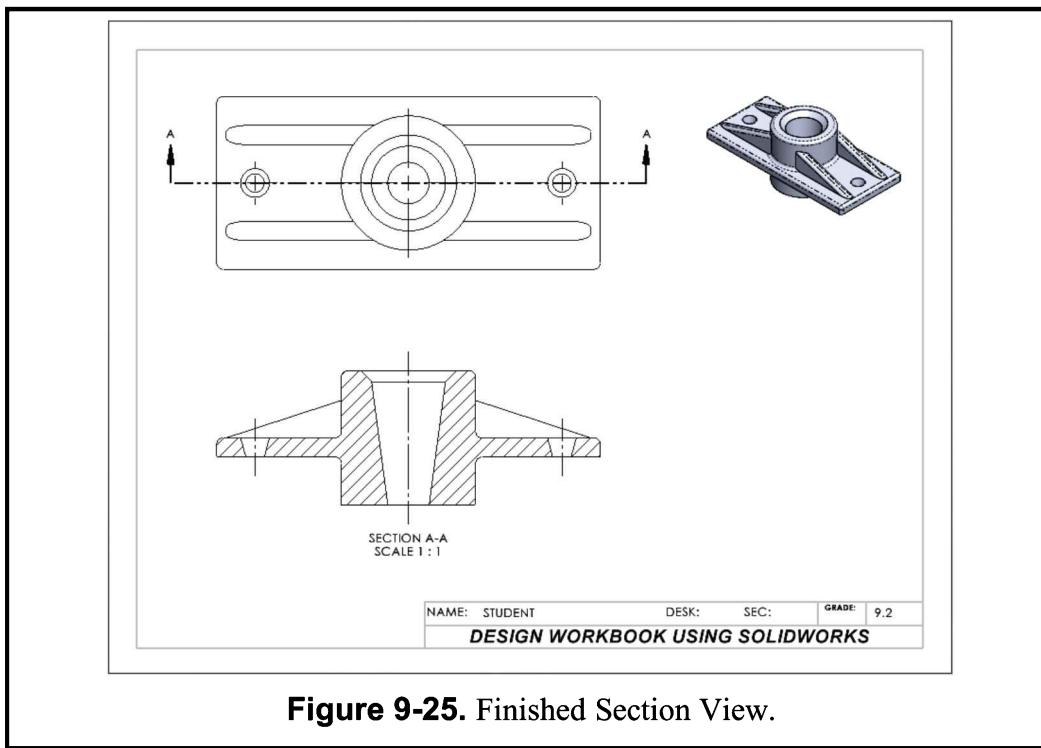


Figure 9-25. Finished Section View.

Now you need to add a **Right** view. From the View Layout tab, select the **Projected View** command. Select the section view and move to the right to project the **Right** view. When you reach the desired location, click to position the view. By projecting the view, the **Right** view will be aligned horizontally with the section view.

Change the **Right** view to **Hidden Lines Visible**. From the Annotation tab select the **Note** command to add the drawing's title **TENSION CABLE BRACKET** and **SCALE 1:1** as shown in **Figure 9-26**.

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When finished save your drawing as **TENSION CABLE BRACKET.slddrw** and print a copy for your lab instructor.

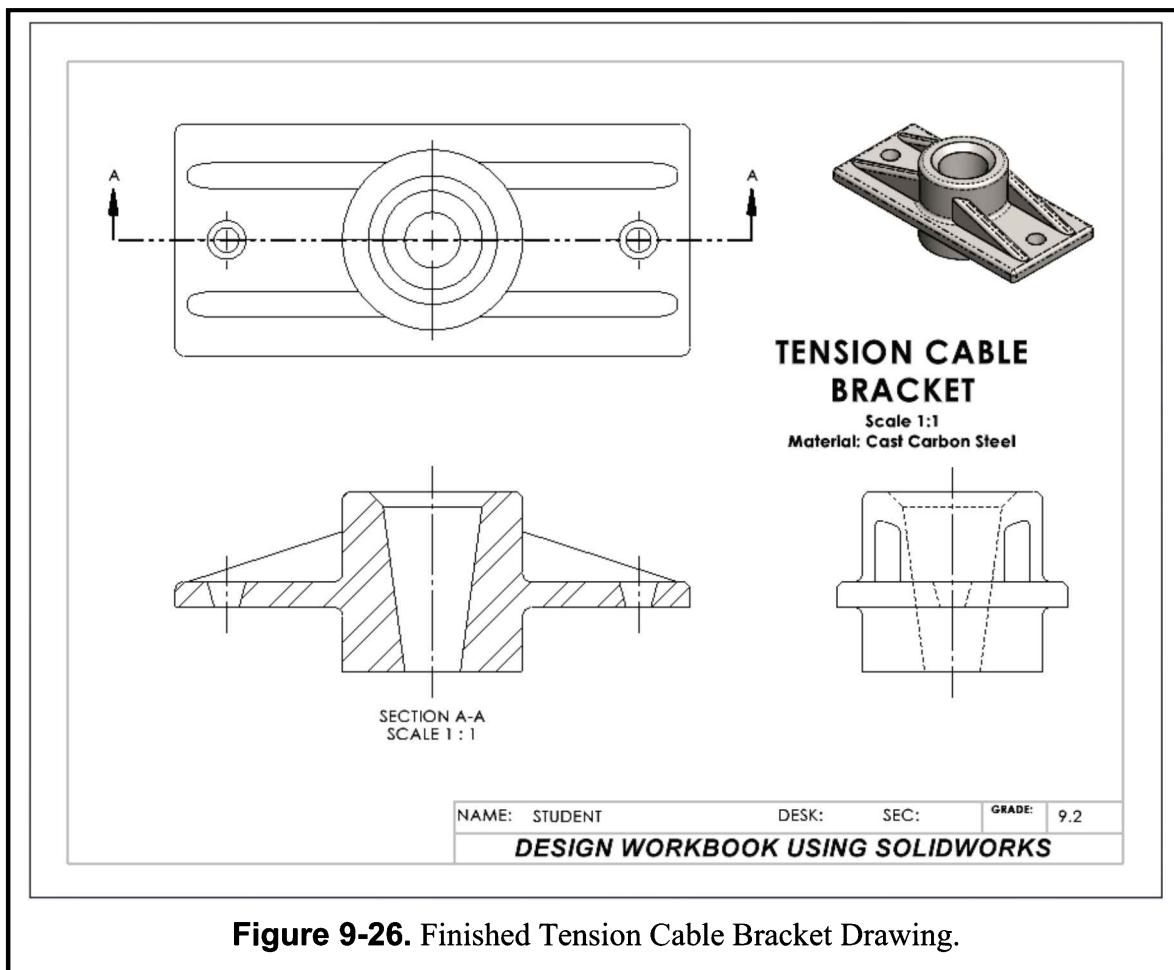


Figure 9-26. Finished Tension Cable Bracket Drawing.

Exercise 9.3: MILLING END ADAPTER

Section Views

In Exercise 9.3, you will model a Milling End Adapter using the commands learned so far. You will make a section view of the 3D model to visualize the internal features of the model, and then you will create a detail drawing with a section view. To get started, you will first build the solid model part.

BUILDING THE MILLING END ADAPTER

Start a new part using the **ANSI-INCHES** template, add a new sketch in the **Front Plane**, and draw and dimension the profile shown in **Figure 9-27**. Remember to add a horizontal **Centerline** starting at the origin. You will need to add a horizontal relation between the top endpoints of the **0.125"** wide slots. Change the units of measure to four decimal places.

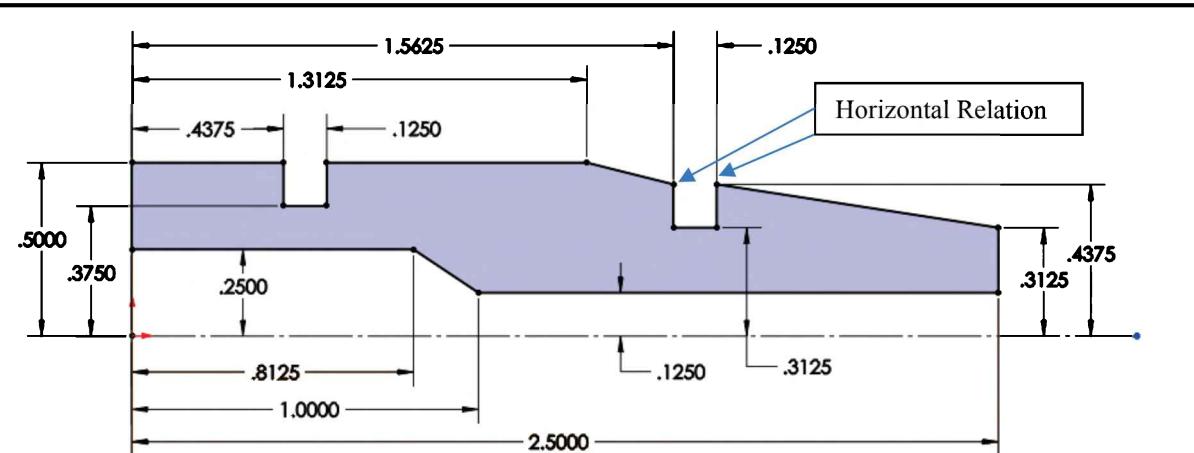


Figure 9-27. Sketch for the Milling End Adapter.

Select the **Revolved Base** command and revolve the profile **360°**. Now add a **Chamfer** to both outside edges of the part. Use the **Distance-Angle** option and make the chamfer **0.0625" x 45°**.

Your part will look like **Figure 9-28**.

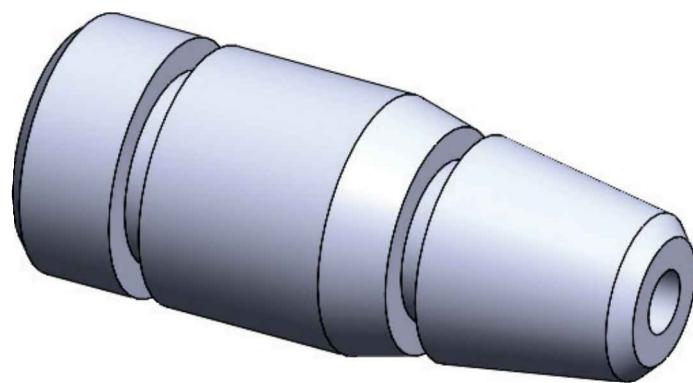


Figure 9-28. The Revolved Part with Chamfers.

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The adapter needs to have two screw holes on the outer surface. In order to locate the holes, you need to add a new plane first. Select the **Reference Geometry** command and click on **Plane**.

In the **First Reference** select the **Top Plane** and select the **Parallel** option. For the **Second Reference** select the outside face of the Milling Adapter. Using these options will create a new plane parallel to the **Top Plane**, and **Tangent** to the selected cylindrical face. If needed, use the **Flip offset** option to locate the new plane above the part, and not below. Click **OK** to finish.

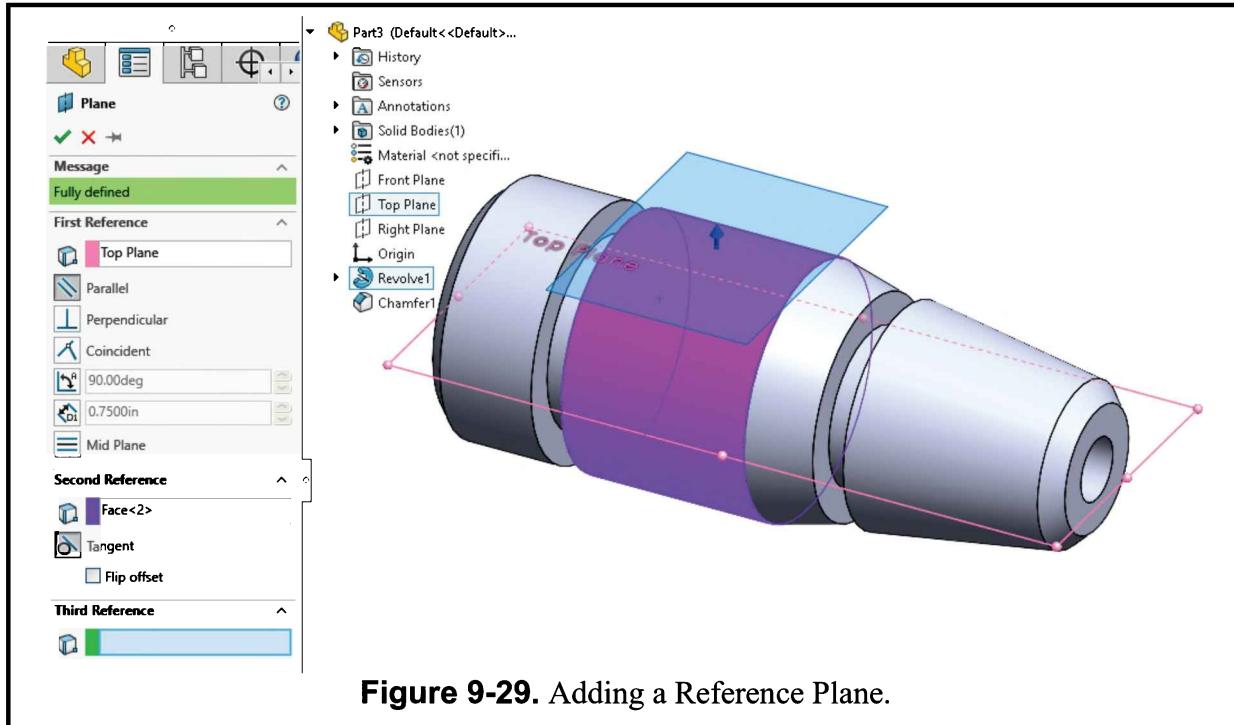


Figure 9-29. Adding a Reference Plane.

Change to a **Top** view, add a new sketch in the new plane (**Plane1**) and draw the profile indicated in **Figure 9-30**. Add an **Equal** geometric relation to make both circles the same and dimension the diameter of either one **0.125"**.

To fully define the sketch, add a **Horizontal** relation between the circles' center and the origin.

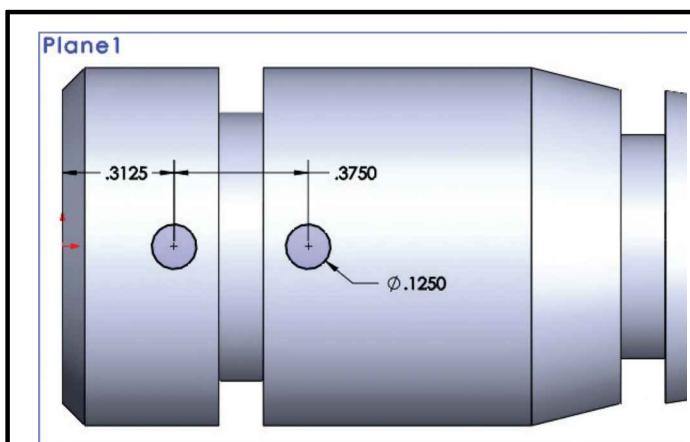


Figure 9-30. Sketch for the Screw Holes.

Now make an **Extruded Cut** down through the cylinder wall using an **Up to Next** end condition.

The **Up to Next** end condition will make the cut until it finds the next model face, in this case the *inner* face of the Milling Adapter. This results in the holes only going through the top half of the part.

Now your model of the Milling End Adapter is complete, and it should look like **Figure 9-36**.

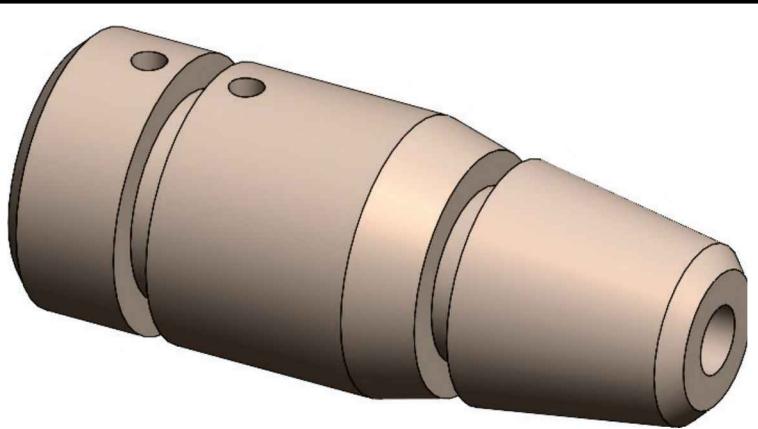


Figure 9-31. Finished Milling End Adapter Part.

Optionally, you can add the sketch in the **Top Plane** and make the **Extruded Cut** going up, but we chose this option to learn a different approach and additional command options.

To complete the model, assign the **Manganese Bronze** material from the **Copper Alloys** library, and save your part as **Milling End Adapter.sldprt**.

MAKING A 3D SECTION VIEW OF THE MILLING END ADAPTER

Using the same process as the previous two exercises, select the **Section View** command, using the **Front Plane**.

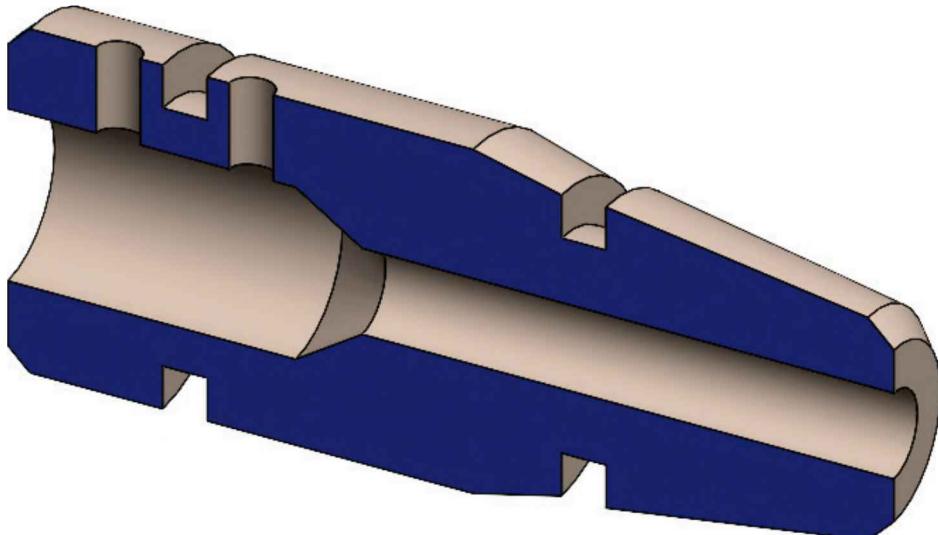


Figure 9-32. 3D Section View of the Milling End Adapter.

MILLING END ADAPTER DETAIL DRAWING

Now you will make the detail drawing of the **Milling End Adapter**. If you changed the **System Options** in exercise 9-1, those settings are already set, and you do not need to change them again. Remember that **System Options** affect all SOLIDWORKS documents and are not document specific.

Start a new drawing using the **Inches** template. Select the **View Palette** tab from the **Task Pane** on the right side of the screen and drag the **Front** view onto the drawing sheet.

Make sure the option **Auto-start projected view** is activated, and project the **Top**, **Right** and **Isometric** views, as shown in **Figure 9-33**.



Select the **Front** view and change it to **Hidden Lines Visible** mode. The rest of the views will also change because they are set to use the parent view's style.

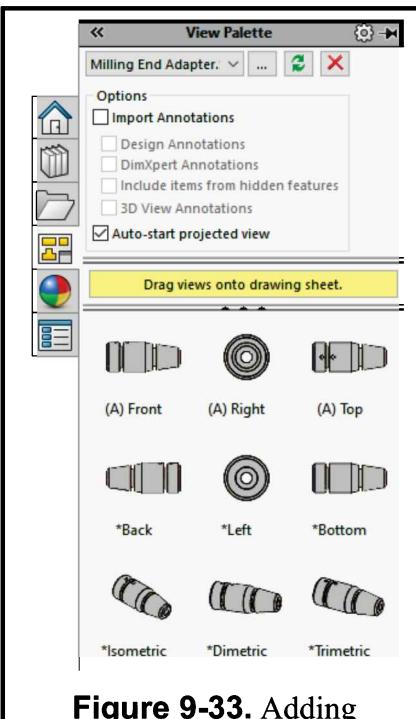


Figure 9-33. Adding Drawing Views.

In the FeatureManager right-click on **Sheet1**, select **Properties** and change the sheet's scale to **2:1**. All four views will change scale. Now select the **Isometric** view, change its scale to **1:1** and display mode to **Shaded with Edges**. Finally, add the missing **Centerlines** to all views as show in **Figure 9-34**.

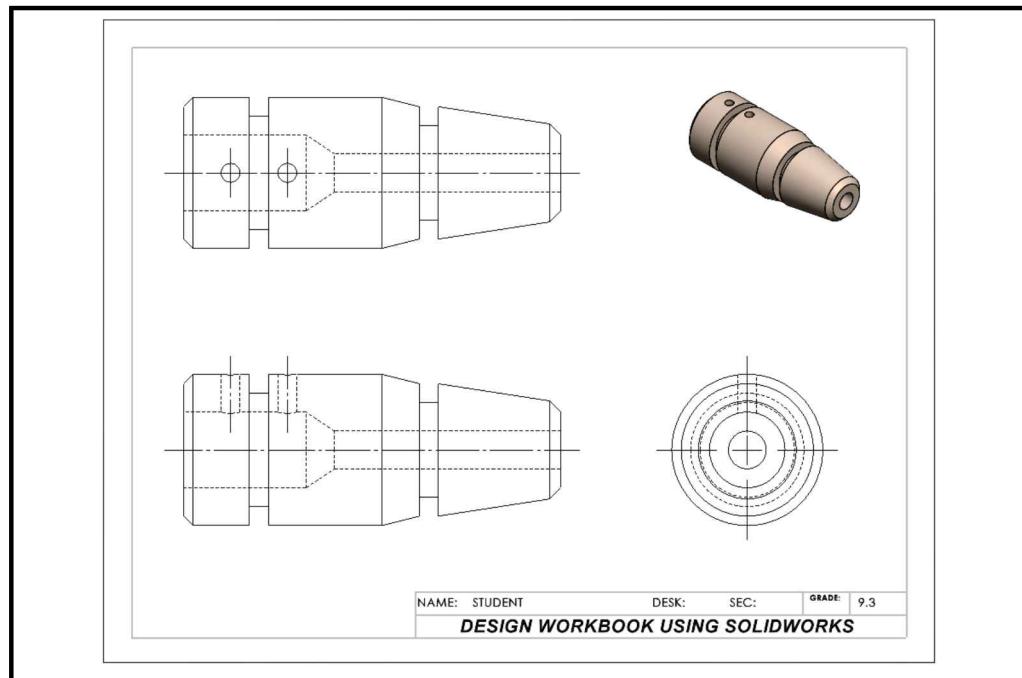


Figure 9-34. Milling End Adapter Drawing.

BROKEN-OUT SECTION VIEW



A **Broken-Out section** shows a partial section to a defined depth in a view. To create it, select the **Broken-Out Section** command from the View Layout tab. The **Spline** sketch tool will be automatically activated.

Go to the **Front** view and draw a closed profile to define the area for the section, as suggested in **Figure 9-35**. When the spline (or any other closed profile) is completed, the **Broken-Out Section** properties are displayed.

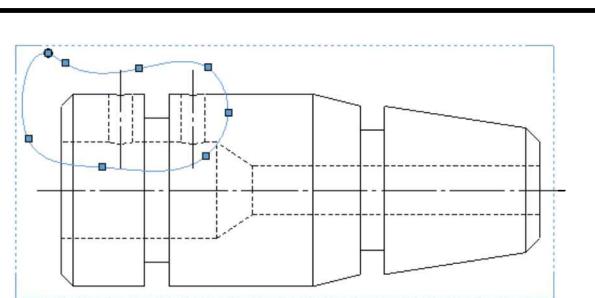


Figure 9-35. Closed Spline on the Front View for the Broken-Out Section.

Turn on the **Preview** checkbox to preview the results. In the **Depth** distance box, enter **0.500** inches. This places the cut depth plane through the middle of the part. You will see the depth planes in the **Top** and **Right** views, including the arrows indicating the direction of cut, as shown in **Figure 9-36**. Click **OK** to finish.

Note: You can draw the closed spline before and drag the nodes to adjust the section area. When the closed profile covers the area of interest, pre-select the closed profile, and then select the **Broken-Out Section** view command.

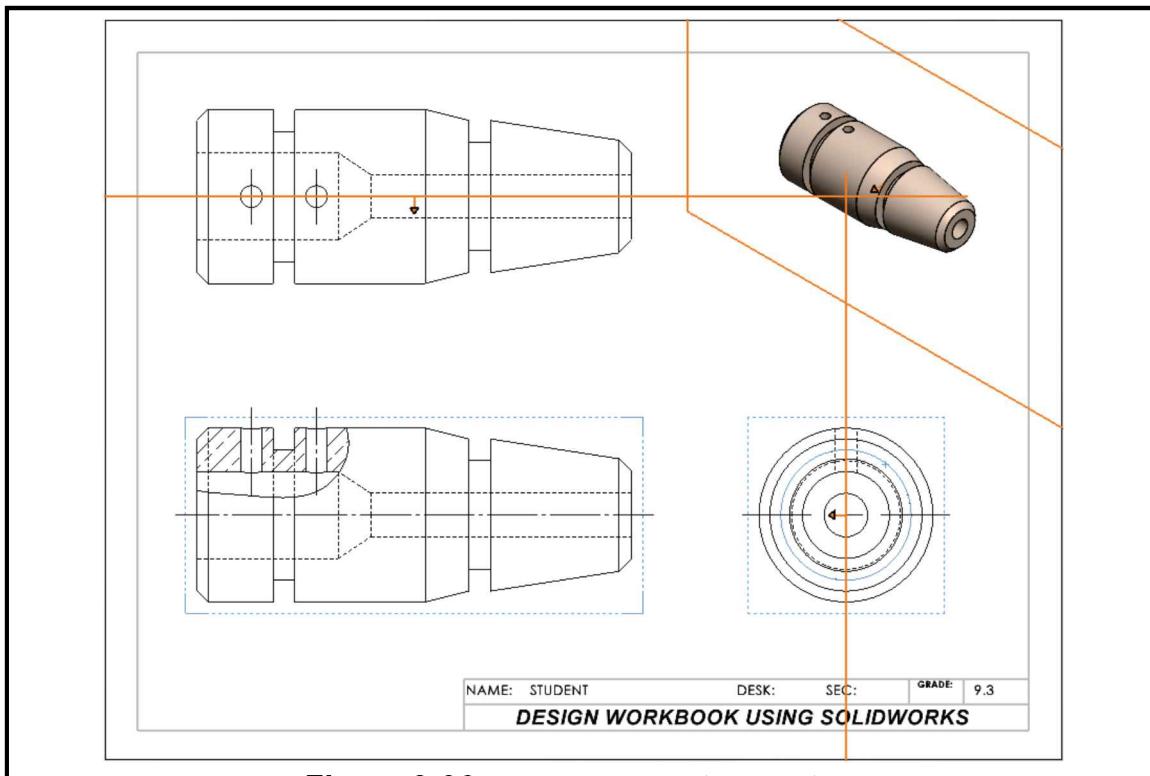


Figure 9-36. Broken-out Section Preview.

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With the drawing views complete, add use the Note command to add the part's name, drawing scale and material, as shown in **Figure 9-37**.

When you are finished, save your drawing as **MILLING END ADAPTER.slddrw** and print a copy to submit to your instructor.

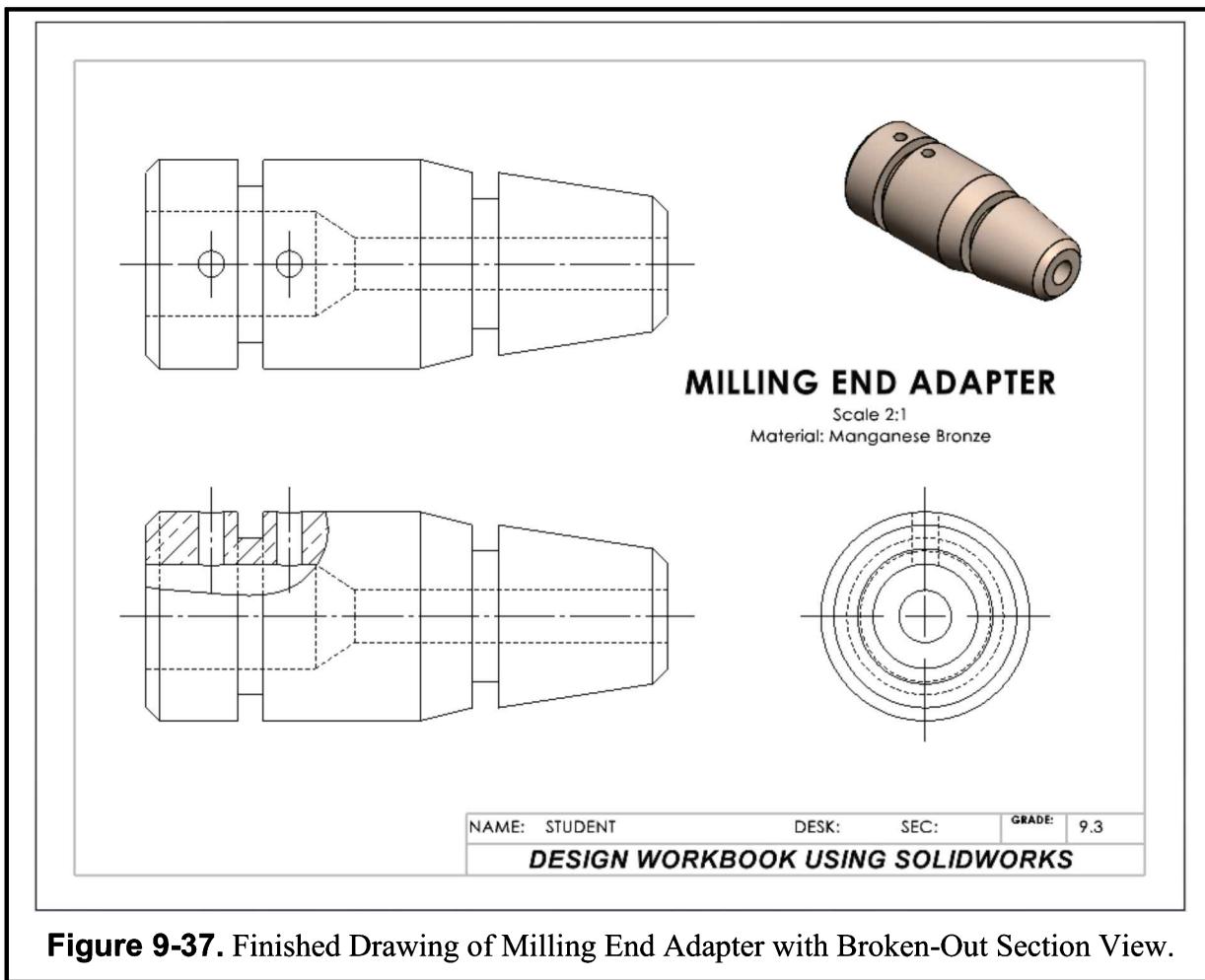


Figure 9-37. Finished Drawing of Milling End Adapter with Broken-Out Section View.

Exercise 9.4: PLASTIC REVOLVING BALL ASSEMBLY Section View

In Exercise 9.4, you will build the three parts of the Plastic Revolving Ball assembly. You will make a 3D section view of the model to visualize and inspect internal features of the assembly. Next you will make an assembly drawing and create a 2D section view of the assembly. To get started, build the solid model of each part, and then create an assembly of the three parts.

Building the Ball

Use the **ANSI-INCHES** template and change the units of measure to four decimal places. Add a new sketch in the **Front Plane** and draw the profile shown in **Figure 9-38**. The arc center is in the **Centerline**.

Add a **Centerline** starting at the origin and dimension as indicated. Use the **Revolved Boss** command, make a full revolution and click **OK** to finish.

Set the material to **PF** from the **Plastics** library and save your part as **Plastic Ball.sldprt**, as shown in **Figure 9-39**.

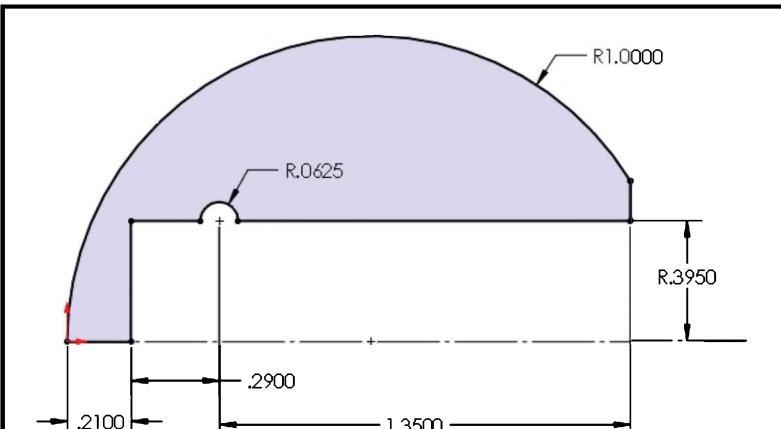


Figure 9-38. The Sketch for the Ball.

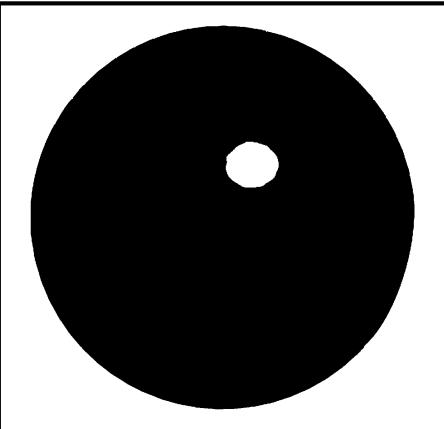


Figure 9-39. The Finished Plastic Ball.

Building the Steel Shaft

Start a new part using the **ANSI-INCHES** template, add a new sketch in the **Front Plane** and draw the profile shown in **Figure 9-40**. Select the two horizontal lines at the top and add a **Collinear** relation between them to fully define the sketch.

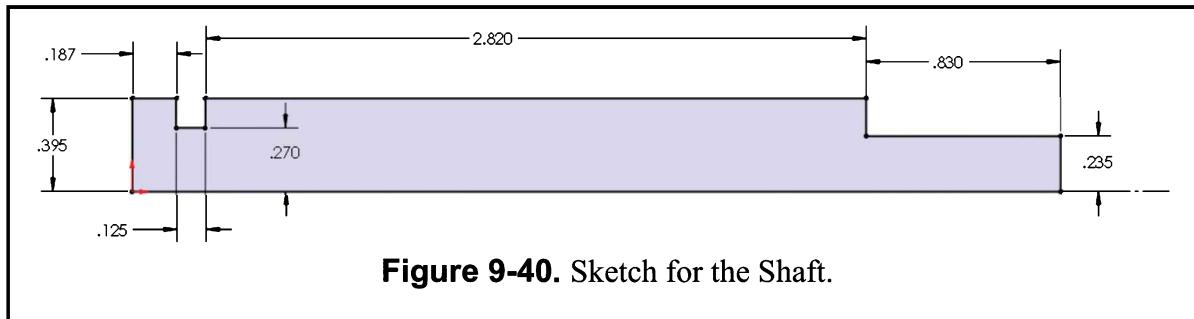


Figure 9-40. Sketch for the Shaft.

Select the **Revolved Boss** command and revolve the sketch **360°** and set the material to **AISI 316 Annealed Stainless Steel (SS)**, from the **Steel** library.

Change to a Right view, select the small end of the shaft, and add a new sketch. Draw a **Hexagon** as shown in **Figure 9-41** and make a **Cut Extrude** using the **Blind** end condition **0.375"** deep. Finally add a **0.0375" x 45° Chamfer** on both ends of the shaft. Your part should look like **Figure 9-42**. Save your part as **Steel Shaft.sldprt**.

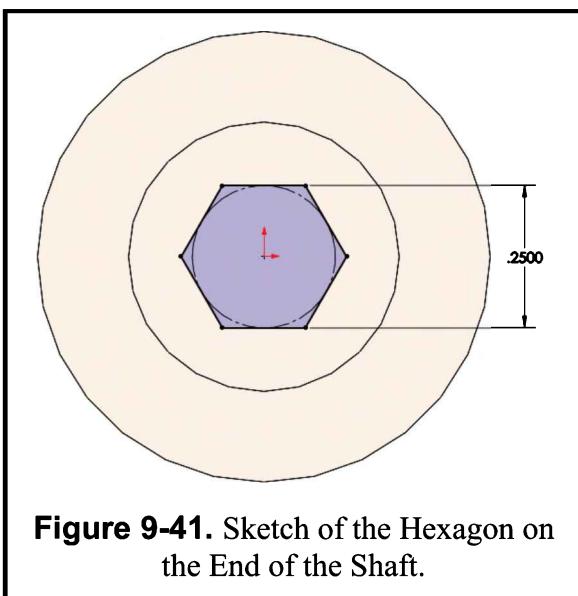


Figure 9-41. Sketch of the Hexagon on the End of the Shaft.

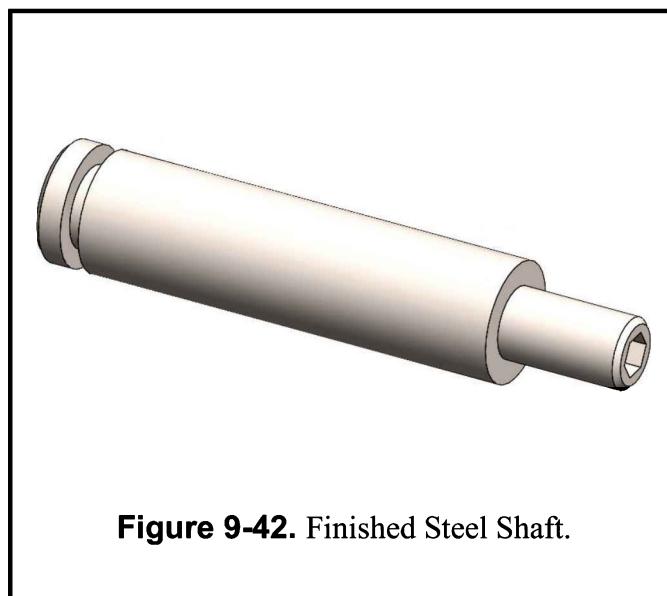


Figure 9-42. Finished Steel Shaft.

Building the Snap Ring

Start a new part using the **ANSI-INCHES** template, add a sketch in the **Front Plane**, draw the profile shown in **Figure 9-43**, and change the units of measure to **four** decimal places. Remember to add a **Vertical** relation between the arc's center and the origin to fully define the sketch.

Select the **Revolved Boss** command and revolve the sketch 360° to create the Snap Ring. Set the material to **Alloy Steel** from the **Steel** library.

Next, change to a **Right** view, select the **Right Plane**, and start a new sketch. Draw the sketch profile shown in **Figure 9-44** using the Dynamic Mirror command and add a **Cut Extrude** using the End Condition **Through all - Both**.

Your part should look like **Figure 9-45**. **Save** your part as **Snap Ring.sldprt**.

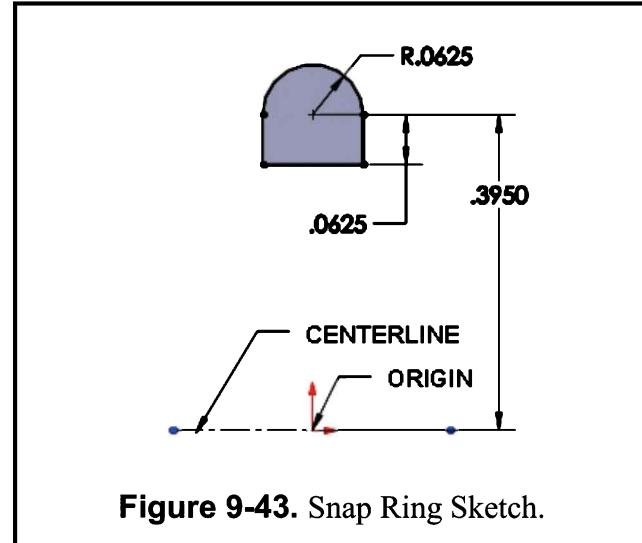


Figure 9-43. Snap Ring Sketch.

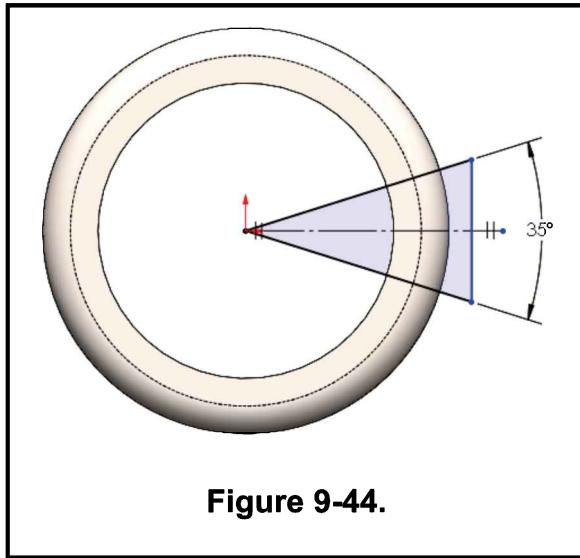


Figure 9-44.

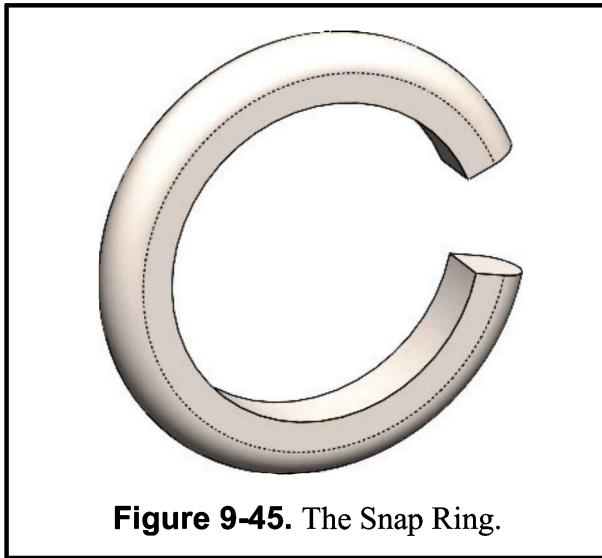


Figure 9-45. The Snap Ring.

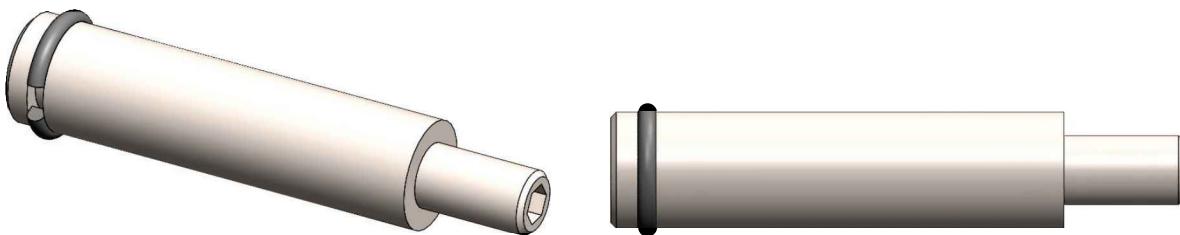
Building the Assembly



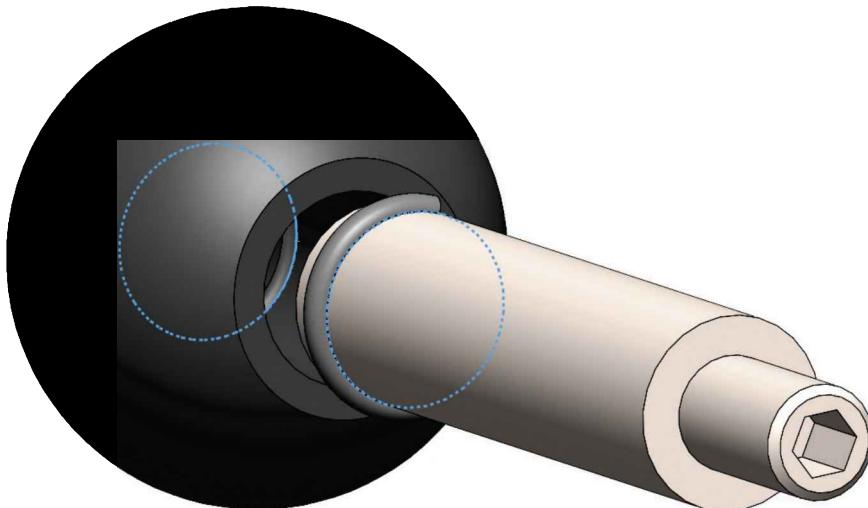
After the three parts are completed, make a new assembly and from the **Begin Assembly** command, select the **Steel Shaft** and click **OK** to add it at the assembly's origin. Next, use the **Insert Components** command, add the **Snap Ring** part and locate it near the shaft's end with the groove.



Use the **Mate** command and make the inside cylindrical face **Concentric** with the groove's cylindrical face, and finally add a **Coincident** mate between a flat face of the **Snap Ring** and the corresponding flat face of the shaft's groove, as shown below. At this point you can drag the **Snap Ring** and it will rotate about the shaft's axis.



Now use the **Insert Components** command to add the **Plastic Ball** and locate it at the end of the shaft with the snap ring. Make the inside face of the **Plastic Ball Concentric** with the cylindrical face of the **Steel Shaft**. If needed, move the ball away from the shaft, and rotate as shown next. Now add a new **Coincident** mate between the inside edge of the groove in the **Plastic Ball**, and the matching edge of the shaft's groove. When finished, save your assembly as **Plastic Revolving Ball.sldasm**.



DISPLAY A SECTION VIEW OF THE ASSEMBLY

After the assembly is complete, click on the **Section View** command and use the **Front Plane** as the section plane. Before you click **OK** to finish the section view, feel free to click and drag the distance arrow to see the effect in the model.

In the **Section View** options turn off the **Keep cap color** option to use the component's color in the section plane. Click **OK** to evaluate the assembly, and finally turn off the **Section View** to continue.

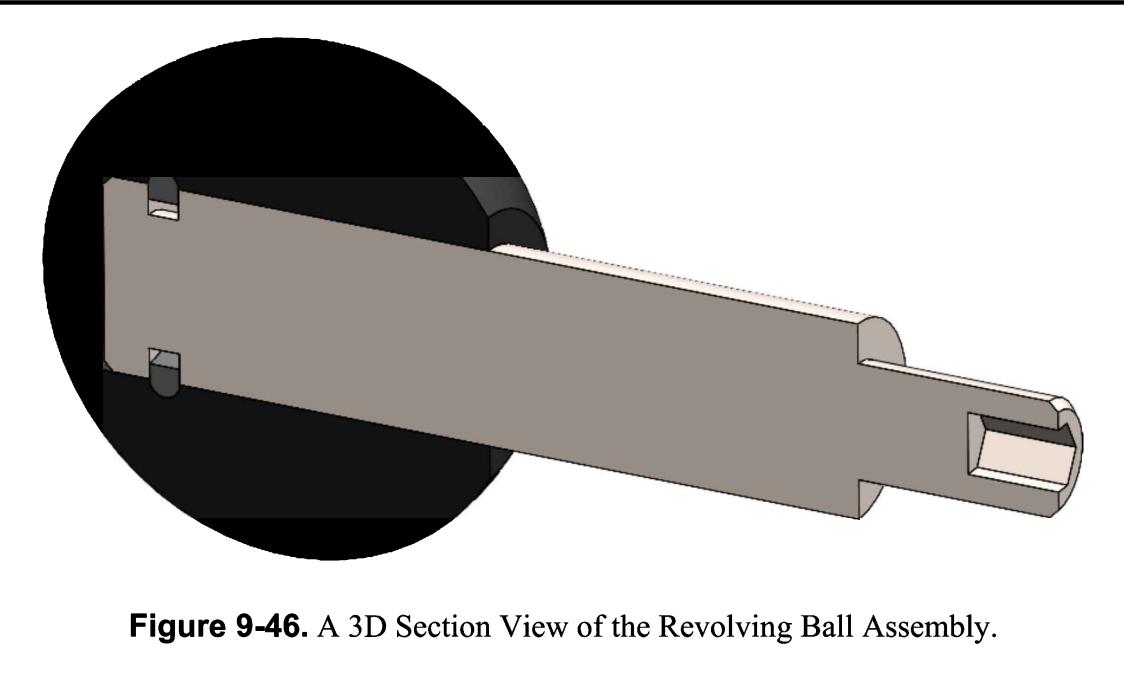
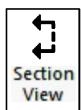


Figure 9-46. A 3D Section View of the Revolving Ball Assembly.

CREATING A DRAWING OF THE REVOLVING BALL ASSEMBLY

Make a new drawing using the **Inches** based template and select the **View Palette** tab. If needed, select the **Plastic Revolving Ball** assembly from the drop-down list, turn off the **Auto-Start projected view** option, and add the **Top** and **Isometric** views to your drawing. Select the **Isometric View**, set the view's scale to **1:2** and change the display mode to **Shaded with Edges**. Now select the **Top view** and set its scale to **1:1** with **Hidden Lines Visible** mode.

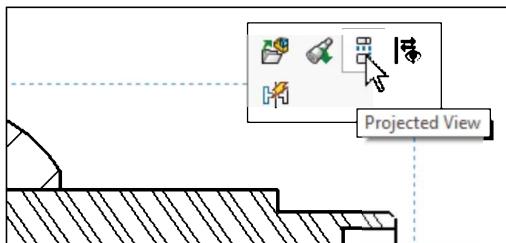
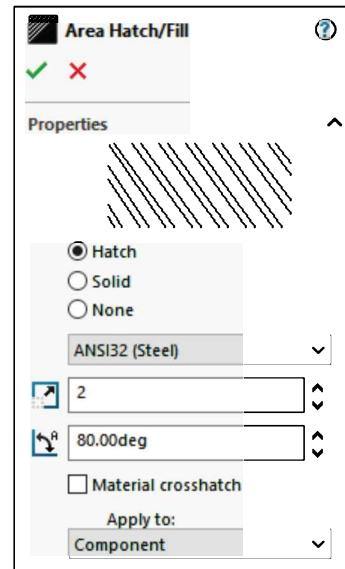


To add the section view, select the **Section View** command, select the **Horizontal Section** option, and locate your cursor at the axis of the shaft in the **Top** view. When the section line snaps to the axis click to locate it. The **Section Scope** dialog is displayed, where you can select the assembly components that will *not* be cut by the section line. In this case you will not select any component and click **OK** to continue. Move the **Section** view below the **Top** view and click to locate it.

The section line will be automatically labeled **A-A** in the **Top** view and should be pointing up. If the arrows are pointed downward, select the flip direction box. The section view is automatically labeled **Section A-A**, as shown in **Figure 9-46**.

Once the section view is placed, you will notice that the **Area Hatch** lines may not be at different angles and the scale of the section lines may not be appropriate.

Select the section area of a component to view its properties, select the option **Apply to: Component**, turn off the **Material crosshatch** option, and change the **Hatch Pattern**, **Scale** and **Angle** to modify the density and direction of the crosshatch for each component.



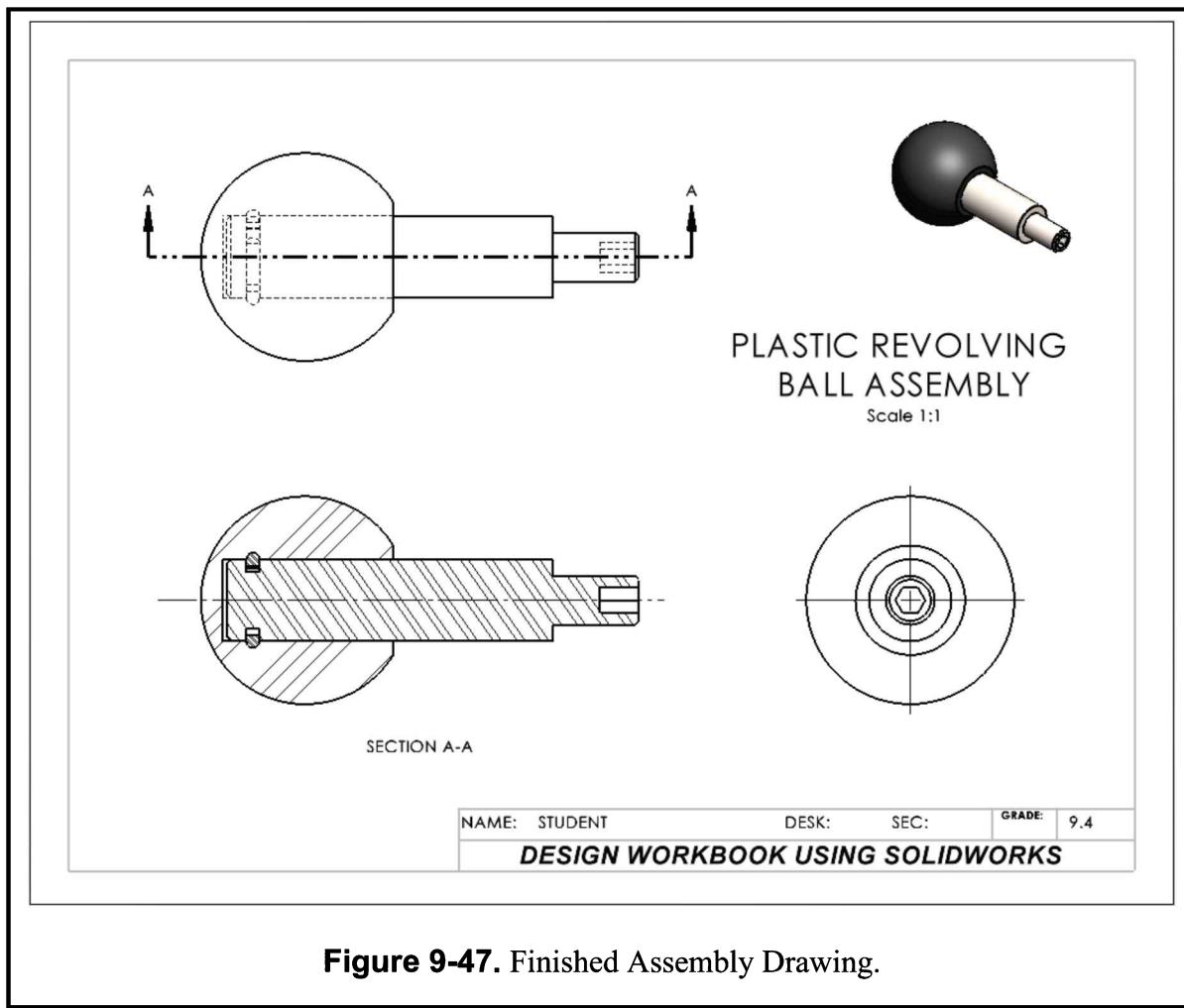
Now you need to add a **Right** view to your drawing. Select the **Projected View** command from the View Layout tab, select the Section view, and locate the **Right** view to the right. Optionally, select the Section view, and from the pop-up toolbar select the **Projected View** command.



When the **Right** view is projected, it will inherit the **Display Style** of the **Section** view. Select the **Right** view and change it to **Hidden Lines Visible**. Use the **Center Mark** command from the Annotations tab and add the missing center marks. Select the outer edge of the plastic ball to locate the center mark. If needed, use the **Centerline** command to add a centerline to the **Section** view as before.

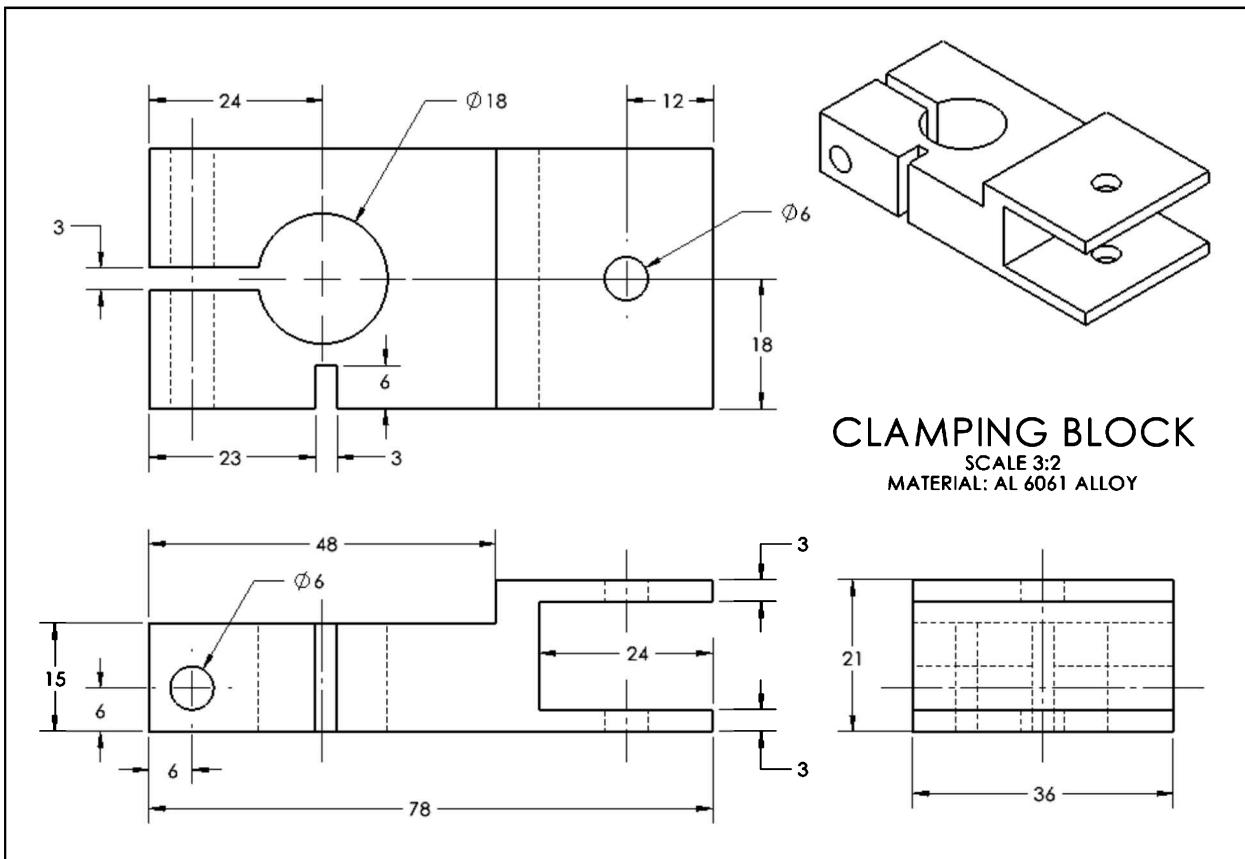
Add a note to your drawing with the name and scale of the assembly with the same font settings used in previous drawings.

When finished, save your drawing as **Plastic Revolving Ball.slddrw** and print a copy for your instructor.



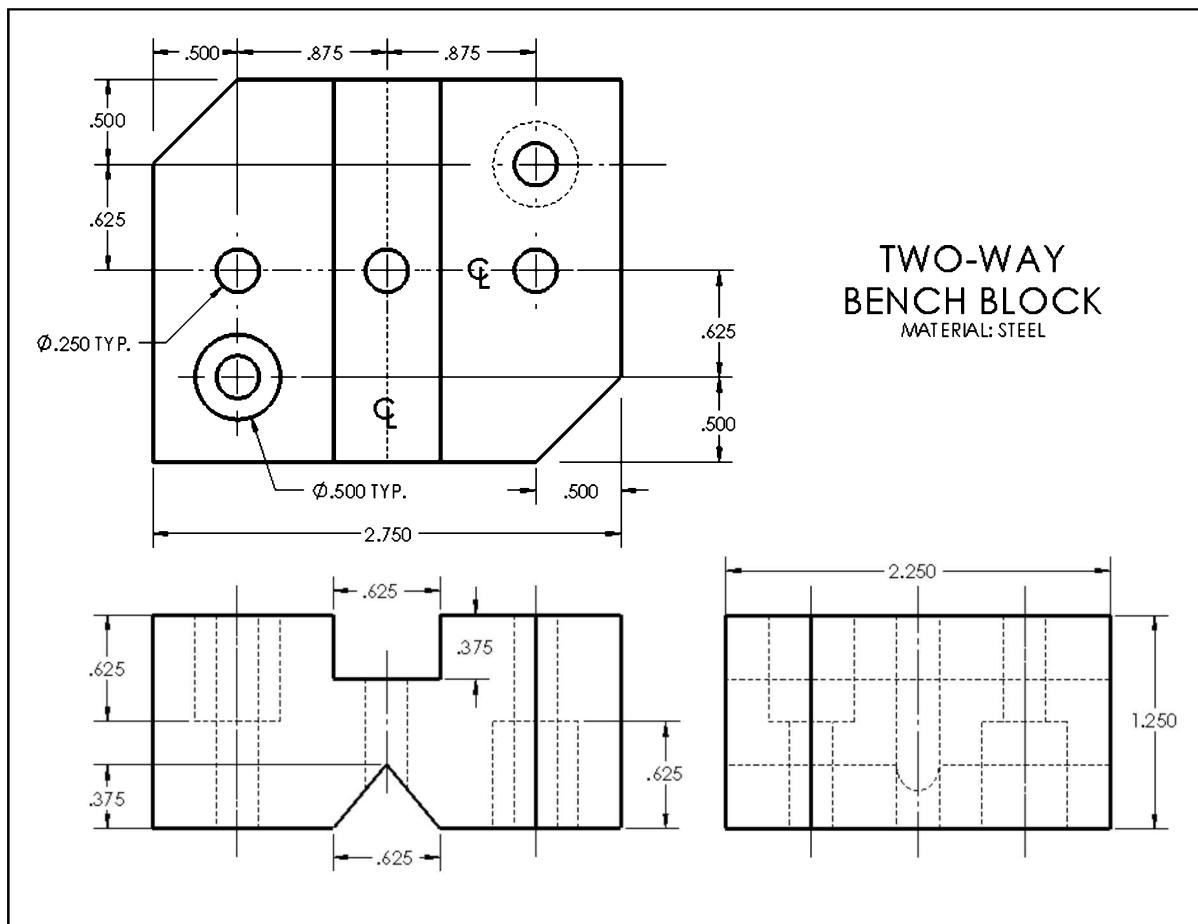
Supplementary Exercise 9-5: CLAMPING BLOCK

Build a solid model of the figure below. Make a drawing and provide a **Section** in the place of the front view. Add a small isometric view in the upper right-hand corner of the sheet. Add the Title, Scale and Material annotations. The part's dimensions are given in millimeters.



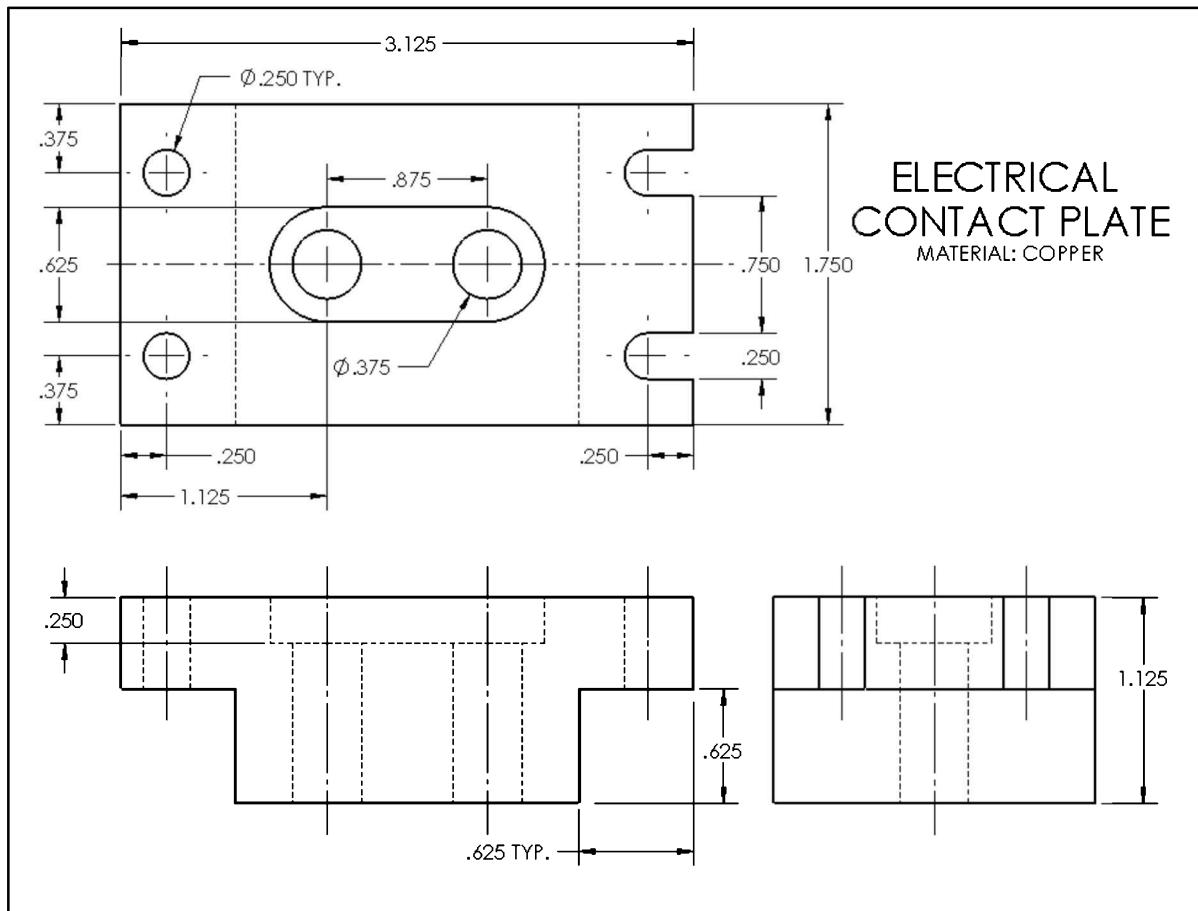
Supplementary Exercise 9-6: TWO WAY BENCH BLOCK

Build a solid model of the figure below and make a drawing. In place of the **Front** view add a section view of the **Top** view. Draw an offset line through the two counterbored holes and the center hole. Next select the menu **Insert, Drawing View, Section View** and locate it in place of the **Front** view. From the **Section** view, project the **Right** view. Finally add an **Isometric** View in the upper right corner. Add notes with the Title and Scale as in previous exercises. Dimensions are given in Inches.



Supplementary Exercise 9-7: ELECTRICAL CONTACT PLATE

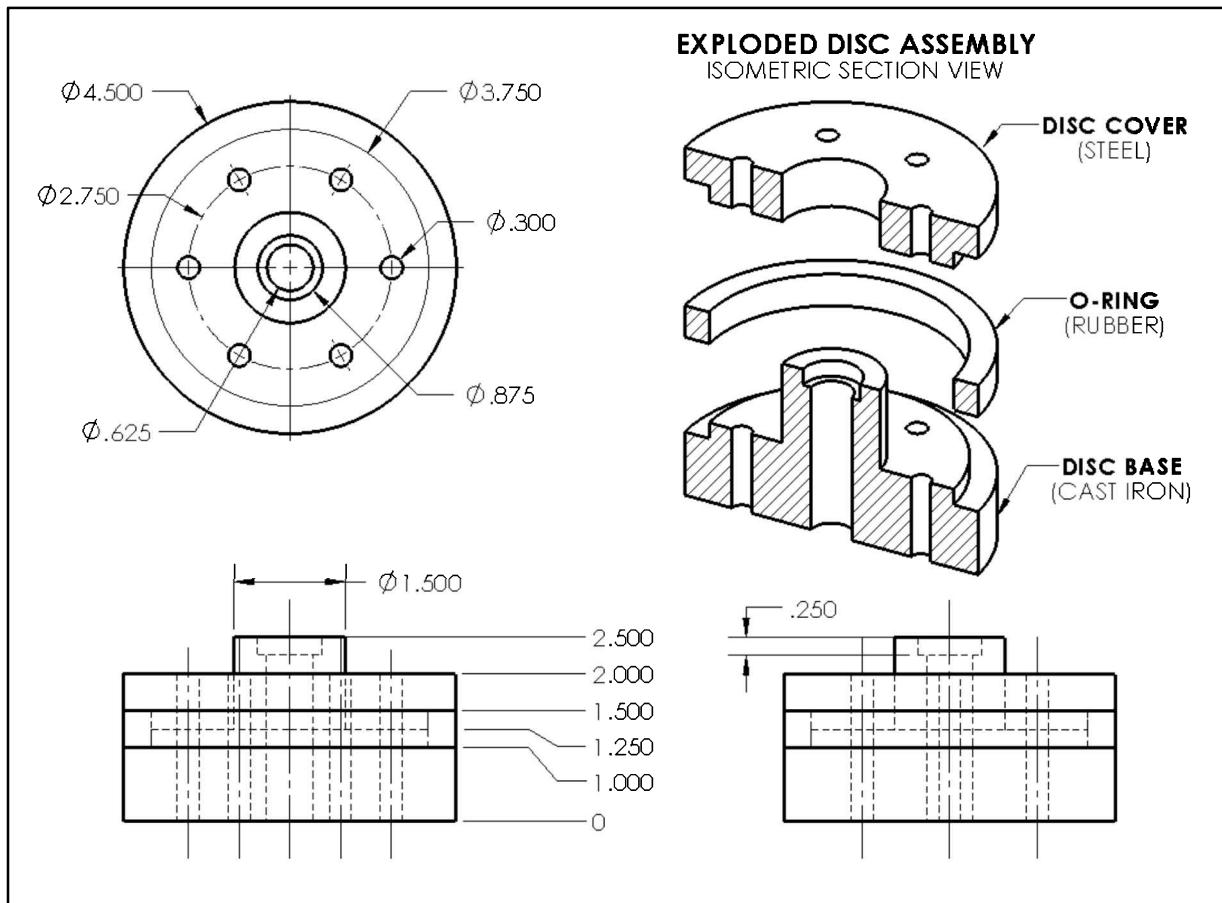
Build a solid model of the figure below and make a drawing. In place of the Front view add a section view of the Top view. Draw an offset line through one of the small holes on the left, then through the slot with the two holes in the center and through one of the slots on the right. Next select the menu **Insert, Drawing View, Section View** and place it in place of the Front view. From the Section view, project the Right view. Finally add an Isometric View in the upper right corner. Add notes with the Title and Scale as in previous exercises. Dimensions are given in Inches.



Supplementary Exercise 9-8: DISC ASSEMBLY

Build solid models of the **Disk Assembly Pieces** below. Build a fully constrained Assembly of the parts and make a drawing of it.

Add a **Section View** in the place of the **Front** view. Finally, add an **Isometric** View in the upper right corner of the unexploded assembly in the upper right corner of your drawing. Add notes with the Title and Scale as in previous exercises. Dimensions are given in Inches.



NOTES:

Design Workbook Lab 10: Manufacturing Detail Drawings

In this lab you will learn how to create a fully annotated 2D detail drawing of a part or assembly, including all the necessary views required in a manufacturing environment. In each exercise you will build the part(s), make the detail drawing, import the solid model dimensions into the drawing, and manually add any missing dimensions and annotations.

The following introduction will teach you how to use the necessary tools to make a complete part or assembly drawing using SOLIDWORKS.

THE DRAWING SHEET AND SHEET FORMAT

In chapter 1 you created a 2D drawing template called **TITLEBLOCK-INCHES.drwdot**, shown in **Figure 10-1**. In this drawing template you added a Title Block with information about the drawing such as student name, desk, section, exercise number, etc.

In industrial and manufacturing drawings, a title block usually has more information including the drawing's scale, material, weight, vendor, part number, notes, revision level, etc. to correctly fabricate the part or assembly, and it is common practice to add multiple drawing Sheets as needed.

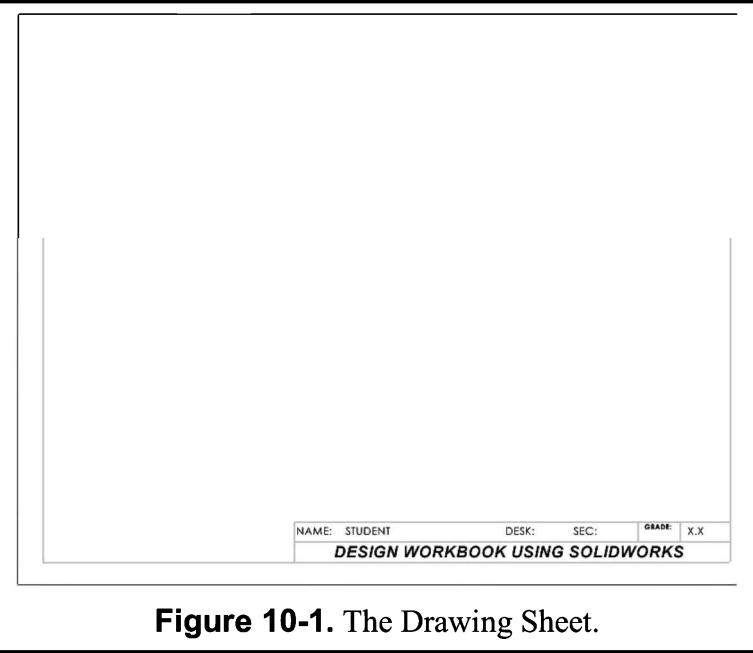


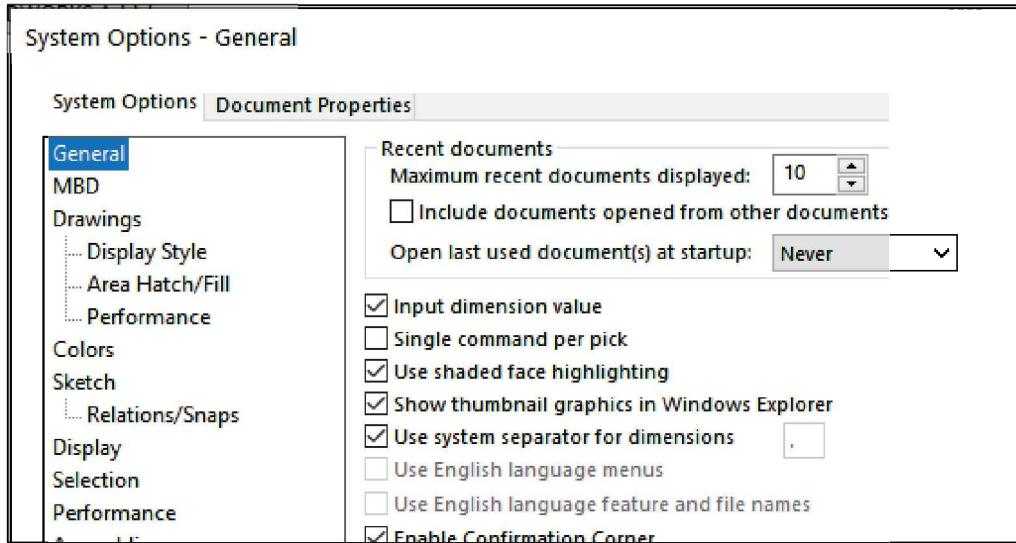
Figure 10-1. The Drawing Sheet.

The **Sheet Format** is in a “locked” layer in the drawing. The purpose of this is to prevent you from accidentally modifying it while making a 2D drawing. The **Sheet Format** can be easily accessed to make changes if needed.

Note: When you modify the **Sheet Format**, the drawing views are automatically hidden.

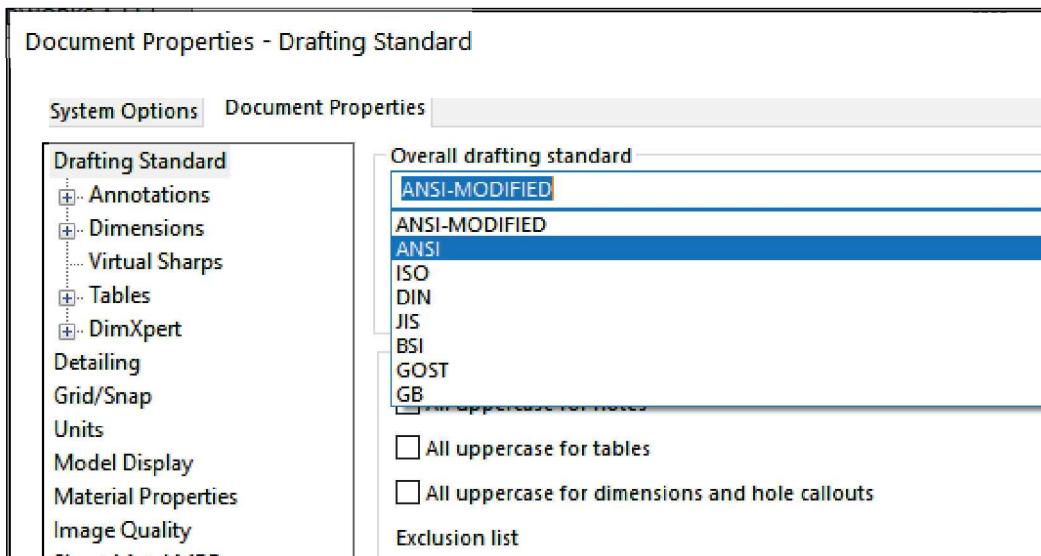
DETAIL DRAWING OPTIONS

As previously covered, the options set in the **System Options** tab control the behavior and functionality of all the SOLIDWORKS documents regardless of the model open, including Sketch options, display style, file locations, performance settings, etc.



The options set in the **Document Properties** tab are specific to the model currently open in SOLIDWORKS, including the units of measure, material, and the **Drafting Standard** used.

Selecting a **Drafting Standard** in a document will automatically adjust all annotation styles, line types and weights, arrow styles and size, symbols, etc. Since most companies use one of the multiple drafting standards available in SOLIDWORKS, making changes to these options is not a common practice in order to prevent deviations from industry standards.



ADDING MODEL VIEWS TO A DRAWING

When you have a part or assembly open, and start a new drawing, select the **View Palette** tab where you can see a preview of each orientation of the model currently open in **SOLIDWORKS**, or you can select a different model from the drop-down list at the top of the **View Palette**.

It is advisable to turn on the **Auto-Start Projected View** option at the top as shown in **Figure 10-2**. From here you can drag any view orientation needed, and automatically project the other views from it. If this option is not activated, you will need to add the different views individually, or use the **Projected View** command as before.

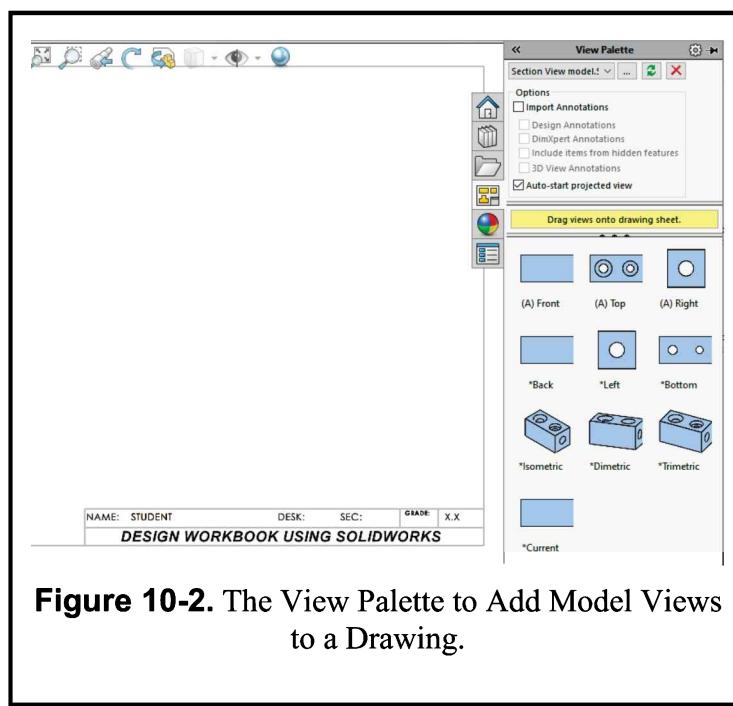


Figure 10-2. The View Palette to Add Model Views to a Drawing.

From the **View Palette** drag the **Front** view onto the drawing and move the cursor around to see the other projected views. Click above to add a **Top** view, on the upper right to add an **Isometric** view, and to the right side to add a **Right** view, as shown in **Figure 10-3**.

The projected views will inherit the Display Style of the original view. Select the Front view and change it to Hidden Lines Visible or Hidden Lines Removed to see the effect.

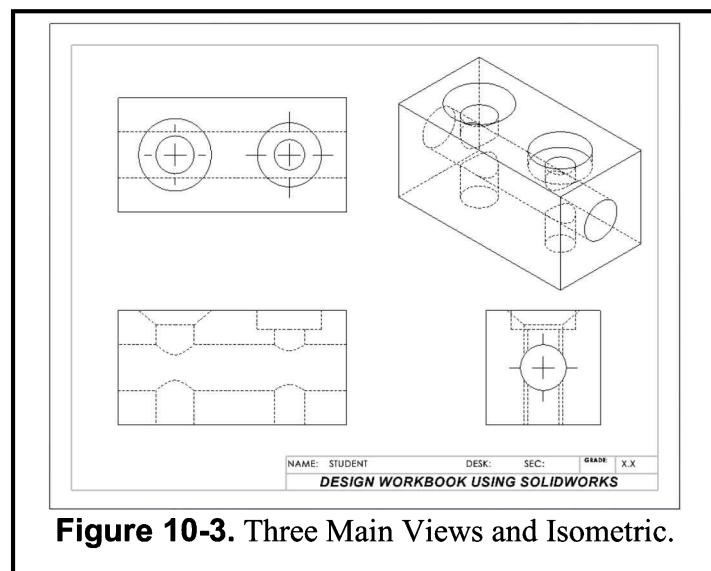


Figure 10-3. Three Main Views and Isometric.

If the drawing views are too big for the sheet, right click on *Sheet1* in the FeatureManager, or in the drawing (*Not a model view*) and select **Properties** to change the scale of the entire sheet.

Select the **Isometric** view. In the view properties change the view's scale to make it smaller and change the Display Style to **Shaded with Edges** mode.

Remember that you can click-and-drag a view to move it in the drawing sheet, in order to have enough space for dimensions and annotations.

If the orthographic views were projected from one view, they will remain aligned vertically or horizontally.

DIMENSIONING THE DRAWING



After adding the necessary views to your drawing, you need to add dimensions and annotations to the different views. When you make a 3D part, sketch and feature dimensions were added. These are called **Parametric** dimensions and can be imported into a detail drawing, making it easier to fully annotate it. When parametric dimensions are changed in the part *or* the drawing, both the 3D model and the 2D drawing views are updated accordingly. To import the 3D model dimensions and annotations into a 2D drawing, select the **Model Items** command from the **Annotation** tab.

In the **Model Items** command options, you can choose to import annotations into a selected view or all drawing views, the annotations of the entire 3D model or only a selected feature, and the type of dimensions, annotations and reference geometry, as shown in **Figure 10-4**.

Some of the annotations available to import into a drawing include:

	Add all dimensions added in a sketch and those created by 3D features.
	Dimensions that had been manually excluded to be imported into a drawing.
	The number of instance copies in patterns.
	Dimensions with tolerance values.
	Dimensions of Hole Wizard profiles
	Dimensions for Hole Wizard locations
	Hole specifications and count

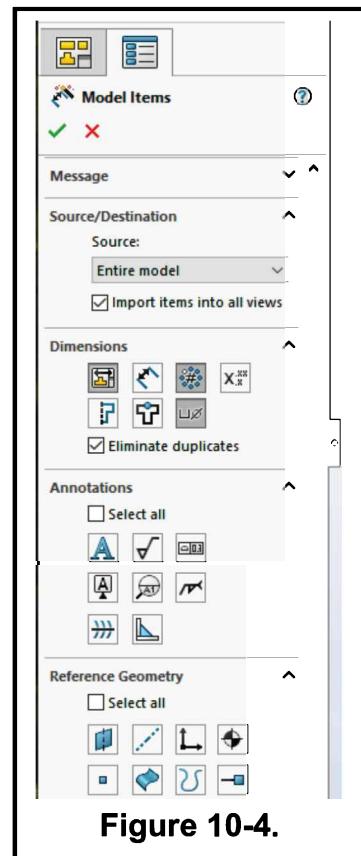


Figure 10-4.

If the imported 3D model dimensions do not fully describe the detail drawing, you can manually add the missing necessary dimensions. By default, manually added dimensions have a parenthesis, which is interpreted as a reference dimension, meaning its value is not exact. A dimension's parentheses can be turned on or off individually in the dimension's properties.

To turn off the parentheses for all manually added dimensions, go to the menu **Tools**, **Options**, **Document Properties**, select the **Dimensions** section and turn off the “**Add Parentheses by default**” option.

Remember this is a document option, not a system setting. You have to set this option for each drawing individually, or if you want it in all drawings, it has to be changed in the template.

After the model dimensions and hole callout annotations are imported, the dimensions can be arranged in the different views for visibility.

To **Move** a dimension from one view to a different one that would better describe the model, you must drag the dimension to the other view while holding the **Shift** key.

After you add the **Center Mark** and **Centerlines** to the drawing, and arrange the dimensions, the drawing looks like **Figure 10-5**.

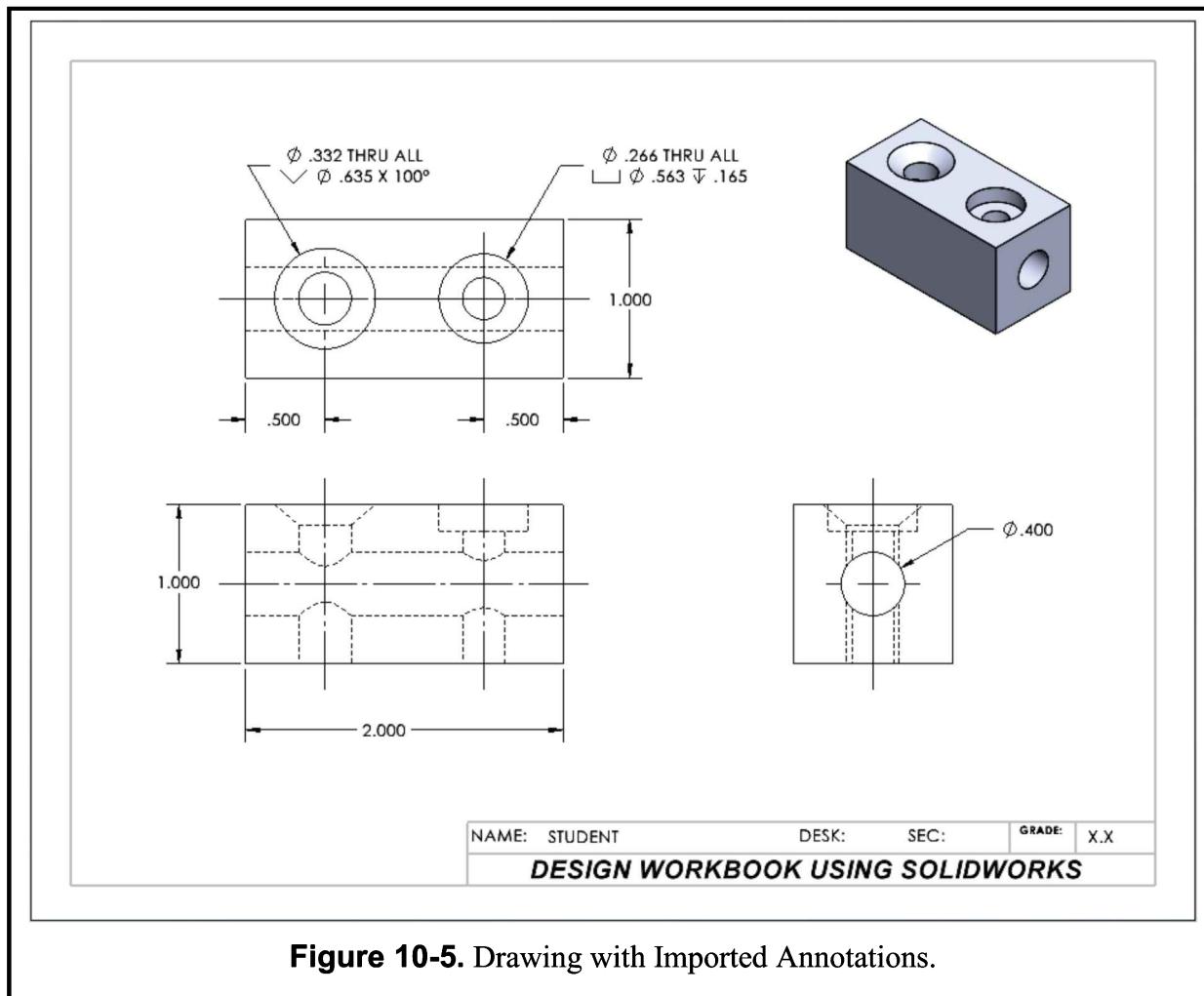


Figure 10-5. Drawing with Imported Annotations.

Now you can manually add the vertical location dimensions of the holes using the **Smart Dimension** tool and turn off the parentheses. Using the **Note** command add the **Title**, **Scale** and **Material** for the part. The finished drawing looks like **Figure 10-6**.

Design Workbook Using SOLIDWORKS 2020

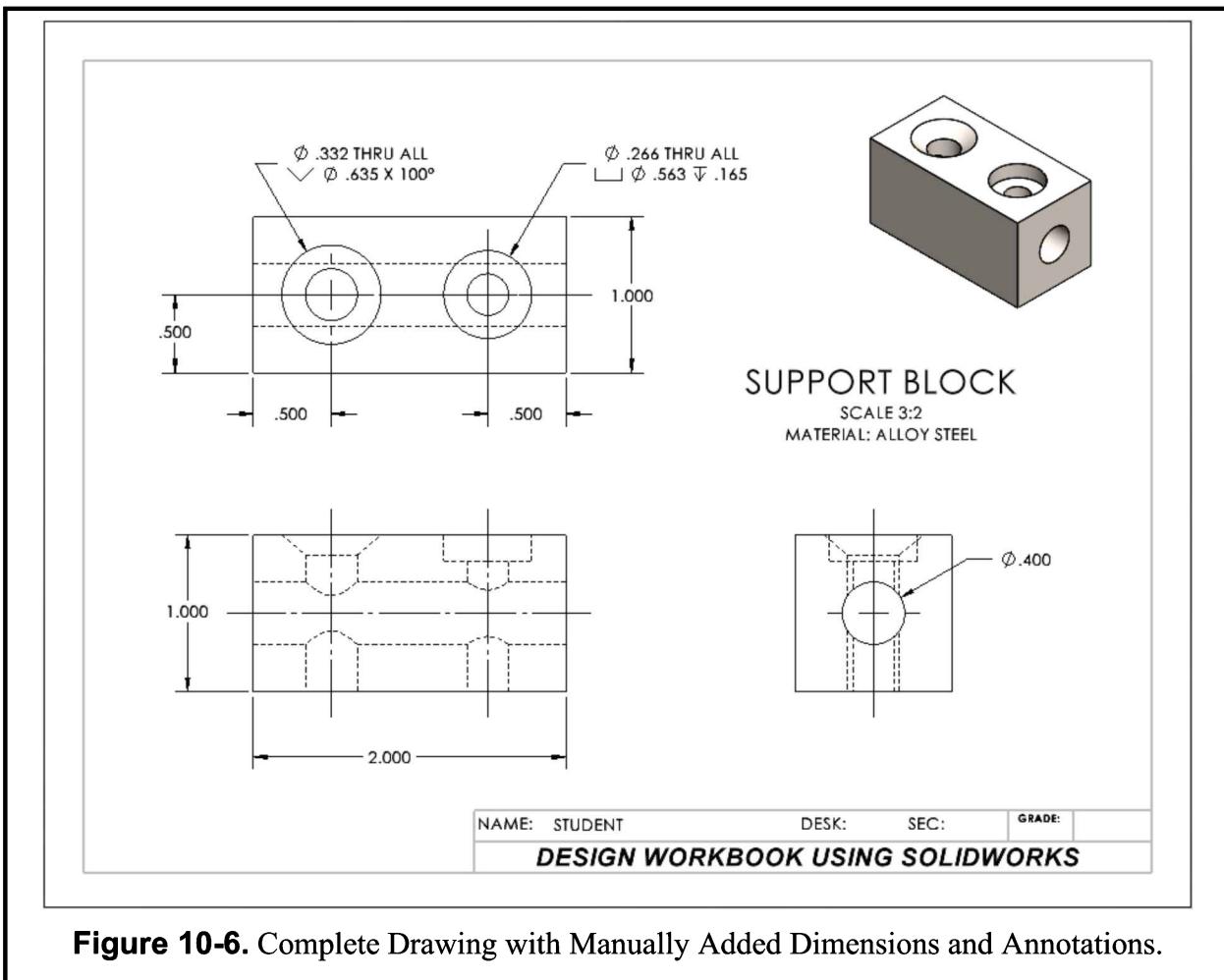


Figure 10-6. Complete Drawing with Manually Added Dimensions and Annotations.

Exercise 10.1: GUIDE BLOCK Drawing

In Exercise 10.1, you will make a solid model of the Guide Block using previously learned commands. Next, you will make a drawing with three orthographic views and an Isometric of the part. You will import the 3D model dimensions and complete the drawing with the title annotations.

BUILDING THE GUIDE BLOCK PART

Start a new part using the **ANSI-INCHES** template, add a new sketch in the **Top Plane** and draw the profile shown in **Figure 10-7**. Use the **Dynamic Mirror** command to maintain symmetry.

When finished, make a **0.75"** **Extruded Boss** in the upward direction.

Now change to a **Back**-view orientation, select the back face, and add a new sketch in it. Draw a rectangle and dimension it as shown in **Figure 10-8**.

Add a vertical **Centerline** through the origin and **Mirror** the rectangle to the other side. Change to an **Isometric** view orientation and **Extrude** the sketch **1.00"** towards the front of the part.

The last feature is a slot that goes all the way through from front to back. Change to a **Front** view orientation, select the front face of the part, and add a new sketch.

Add a rectangle and dimension it as shown in **Figure 10-9**. To maintain symmetry, add a  **Midpoint** relation between the centerline's endpoint and the lower horizontal line of the rectangle.

Now use the **Extruded Cut** command to cut the sketch using a **Through All** condition. When finished, change the part's material to **AISI 1020** from the **Steel** library.

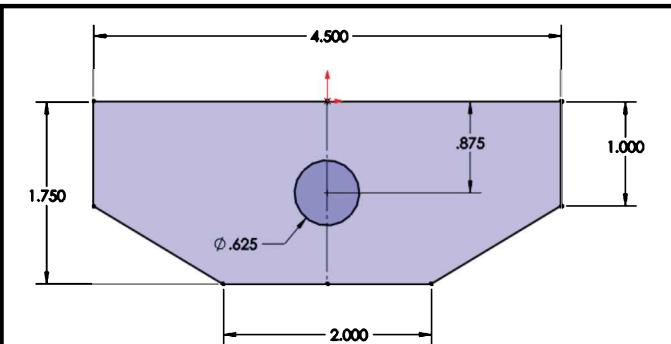


Figure 10-7. Sketch for the Base Feature.

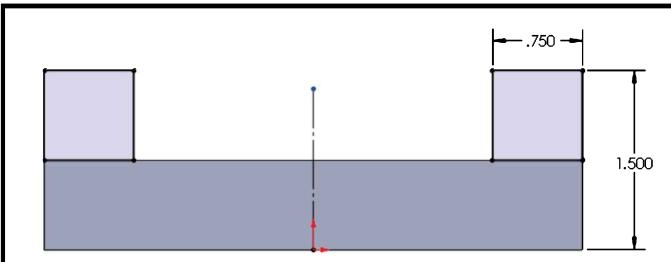
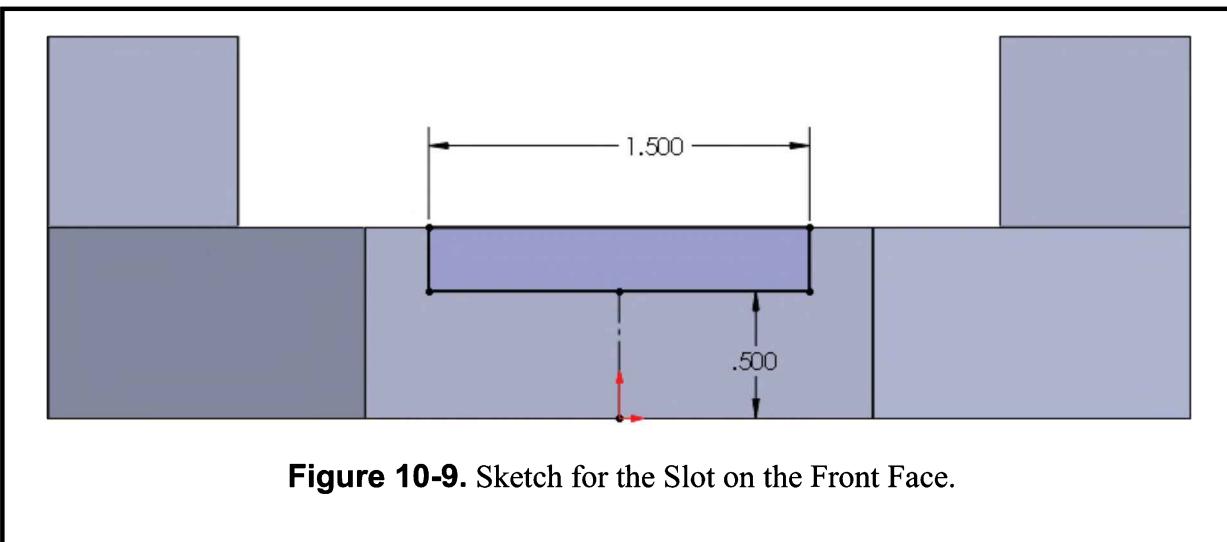
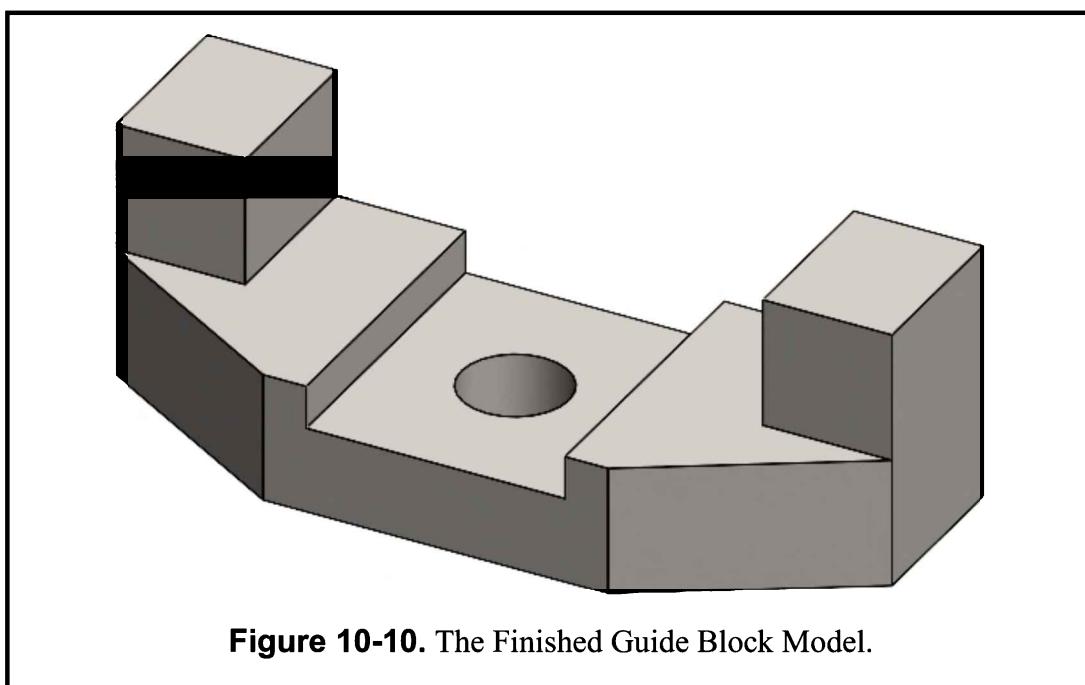


Figure 10-8. Sketch on the Back Face.



The Guide Block model is now complete, as shown in **Figure 10-10**. Save your part as **GUIDE BLOCK.sldprt** to continue. You will now make the detail drawing.



MAKE A MULTI-VIEW DRAWING

With the Guide Block still open, start a new drawing using the Inches template. Open the **View Palette**, make sure the **Auto-Start Projected view** option is turned on and the **Guide Block** is selected in the drop-down list at the top.

Select the **Front** view and drag it into the drawing sheet, move up to add the **Top** view, to the top-right to add an **Isometric** view, and to the right to add a **Right** view. If needed, right click in **Sheet1** on the FeatureManager, select **Properties** and change the scale to 1:1. Select the Front view and change it to **Hidden Lines Visible** mode; the rest of the projected views will change at the same time.

Select the **Isometric** view and change its scale to 1:2 and **Shaded with Edges** display mode. Use the Centerline command to add the missing centerlines to the Front and Right views. Your drawing now looks like **Figure 10-11**.

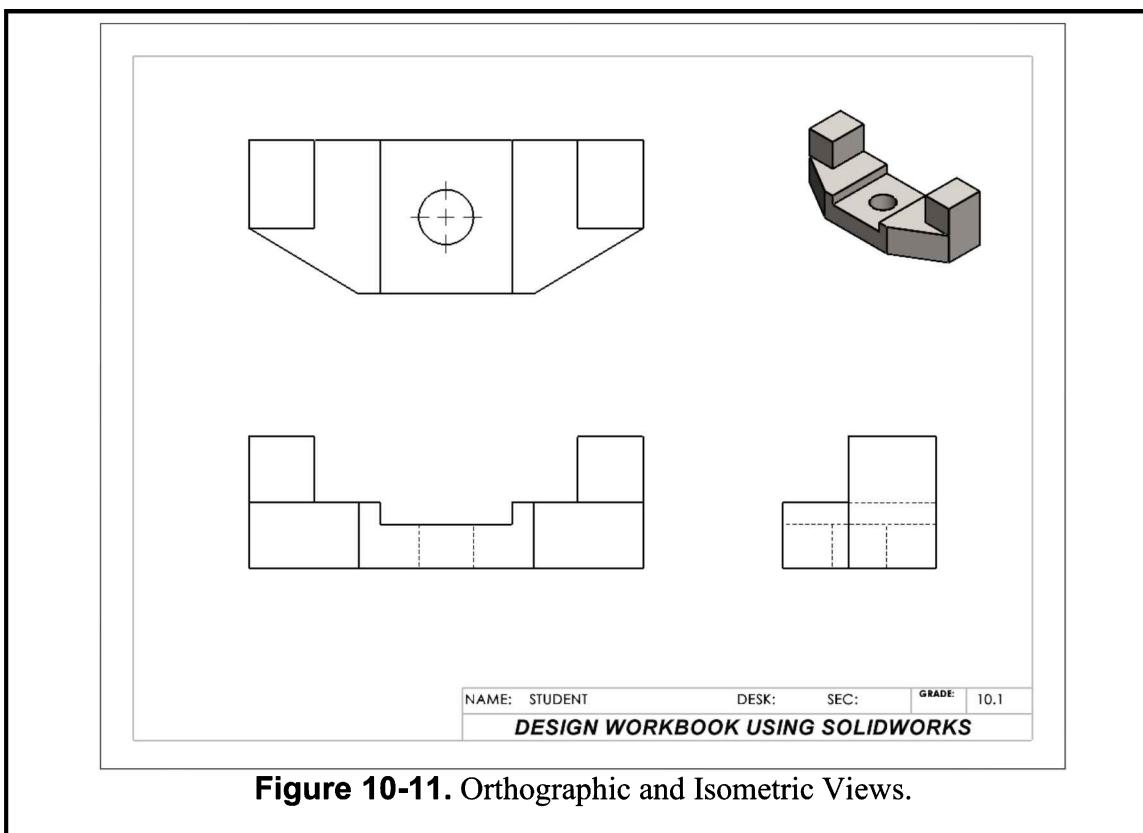


Figure 10-11. Orthographic and Isometric Views.

DIMENSIONING THE DRAWING



From the **Annotations** tab, select the **Model Items** command. In the Source section select **Entire Model**, turn on the option **Import items into all views** and click **OK**. Your 2D drawing with the imported 3D model dimensions should look like **Figure 10-12**.

To distribute the dimensions in the drawing, select the **0.750"** vertical dimension in the **Front** view, hold down the **Shift** key and drag it to the **Right** view. In the **Top** view, there are two **1.00"** dimensions, and one of them is redundant. Select one and delete it.

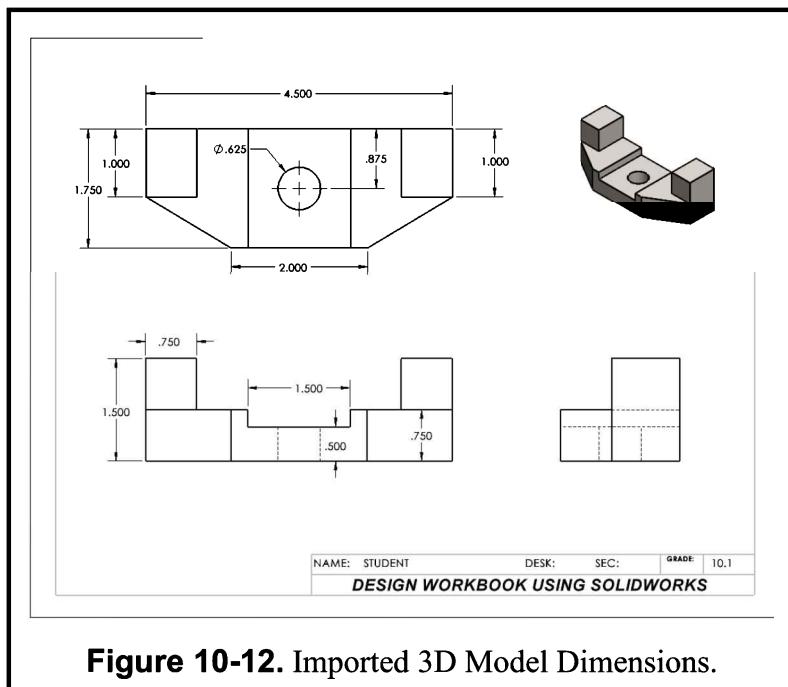


Figure 10-12. Imported 3D Model Dimensions.

Since the part was made symmetric about the origin, some dimensions are not present, but can be added manually using the **Smart Dimension** tool. Add the missing dimensions to match the completed drawing in **Figure 10-13**. Remember to turn off the parentheses to the manually added dimensions in the dimension's properties. Finally add a **Note** with the Title, Scale and Material. Save your drawing as **GUIDE BLOCK** and **Print** a copy to submit to your instructor.

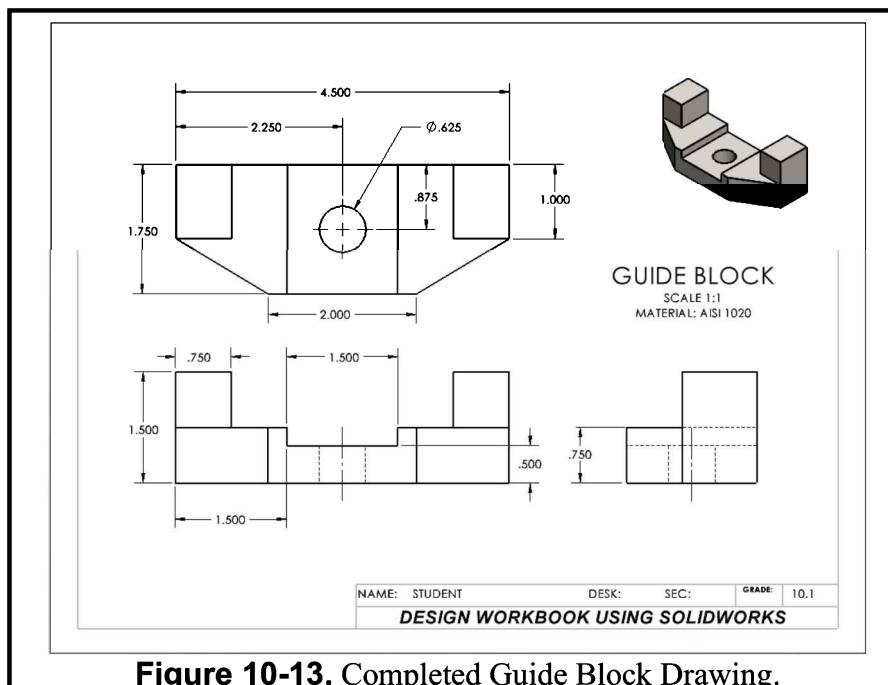


Figure 10-13. Completed Guide Block Drawing.

Exercise 10.2: PIPE JOINT Drawing

In this Exercise 10.2, you will build a solid model of a Pipe Joint. This exercise is dimensioned using Millimeters. Next, you will make a drawing with three orthographic views and an Isometric of the part. You will import the 3D model dimensions and complete the drawing with annotations.

BUILDING THE PIPE JOINT MODEL

Start a new part using the **ANSI-METRIC** template, add a new sketch in the **Top Plane** and draw the profile indicated in **Figure 10-13**.

Draw the two large circles, add a relation to make their centers **Horizontal**, and then draw the two lines. Add a **Tangent** relation between both circles and each line.

Add a vertical centerline that extends beyond the large diameter circle and dimension the sketch.

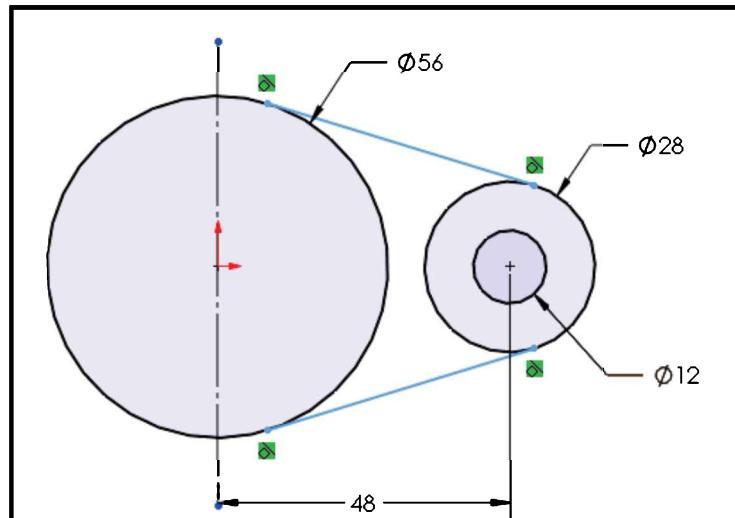


Figure 10-14. Sketch for the First Feature.



Next use the **Trim Entities** tool and remove the left and right side of the **56 mm** diameter circle, and the left side of the **28 mm** diameter circle.

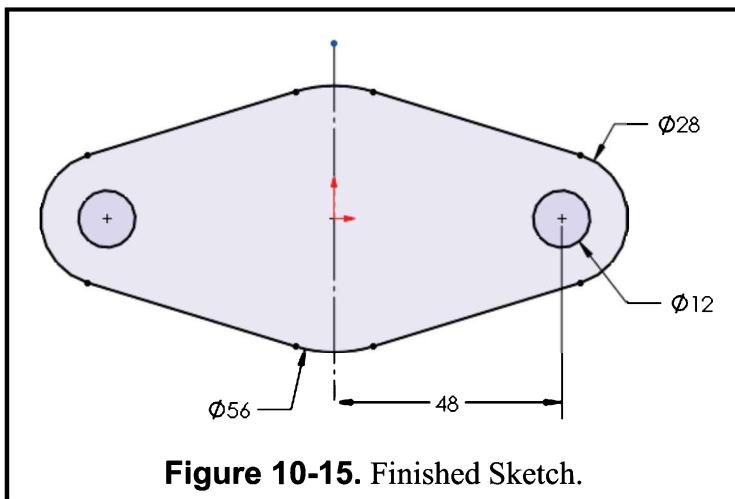


Figure 10-15. Finished Sketch.

Use the **Mirror Entities** command to mirror the entire sketch about the vertical centerline as shown in **Figure 10-15**.

Finally **Extrude** the Sketch 16 mm up to complete the base feature for the Pipe Joint, as shown in **Figure 10-16**.

Next, add a new sketch in the top face of the part, draw two circles **concentric** to the holes in the first feature, add an **Equal** sketch relation and dimension one of them **18 mm** in diameter. Make an **Extruded Cut 6 mm** deep into the base to make the counterbore holes.

Add a new sketch in the top face and draw a circle at the origin. Add an **Equal** relation to the **56 mm** diameter edge and add a **24 mm Extruded Boss**.

In the topmost face, add a new sketch, draw a circle at the origin and dimension it **32 mm** diameter. Make an **Extruded Cut** using a **Through All** end condition to make the center hole. Your part should now look like **Figure 10-17**.

Finally add a **3 mm** Fillet to the edges indicated in **Figure 10-17**, and a **3 mm x 45° Chamfer** to the top inside edge. Set the part's material to **Wrought Stainless Steel**.

The finished Pipe Joint part is shown in **Figure 10-18**. Save your part as **PIPE JOINT.sldprt** to continue. Now you will now make the detail drawing.

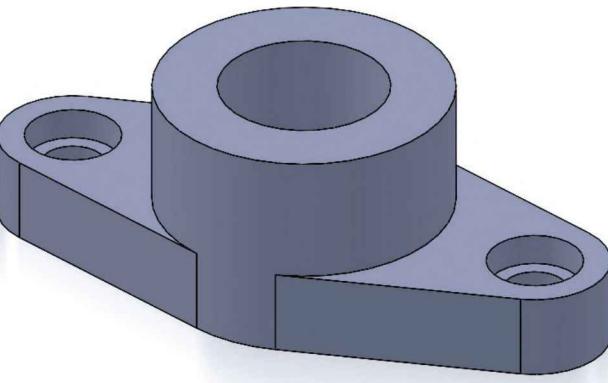


Figure 10-16. The Pipe Joint Model.

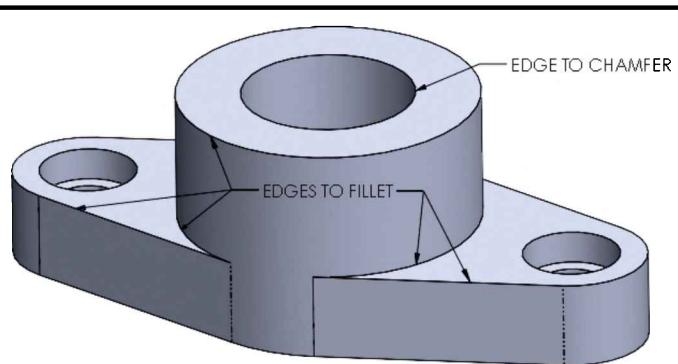


Figure 10-17. The Edges to Fillet and Chamfer.

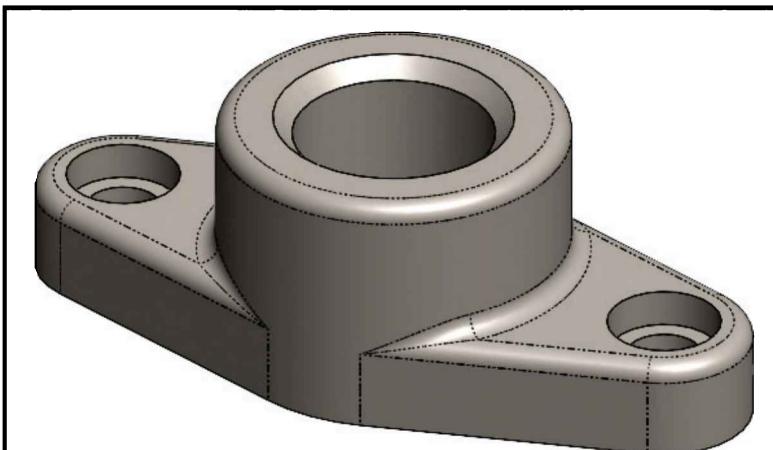


Figure 10-18. The Finished Pipe Joint Model with Fillets.

MAKE A MULTI-VIEW DRAWING

With the Pipe Joint open, start a new drawing using the **Metric** template. Open the **View Palette**, and for this exercise make sure the **Auto-Start Projected view** option is turned off and the **Pipe Joint** is selected in the drop-down list.

From the **View Palette** drag the **Top** and **Isometric** views. Select the **Isometric** view and change it to **Shaded with Edges** mode.

If needed, right click on the **Top** view, select **Tangent Edge**, **Tangent Edges Removed**, and change the **Top** view's scale to **1:1**, as shown in **Figure 10-19**.

From the View Layout tab select the **Section View** command.

Activate the **Horizontal** section view option and click in the **Top** view when the section line snaps in the middle of the part. Immediately locate the section view below in place of the **Front** view.

Now select the **Section** view, from the View Layout tab select the **Projected View** command and locate the projected **Right** view on the right side of the sheet.

Select the **Right** view and set the Display Style to **Hidden Lines Visible**. Add the **Centerlines** to continue. The final drawing views layout is shown in **Figure 10-20**.

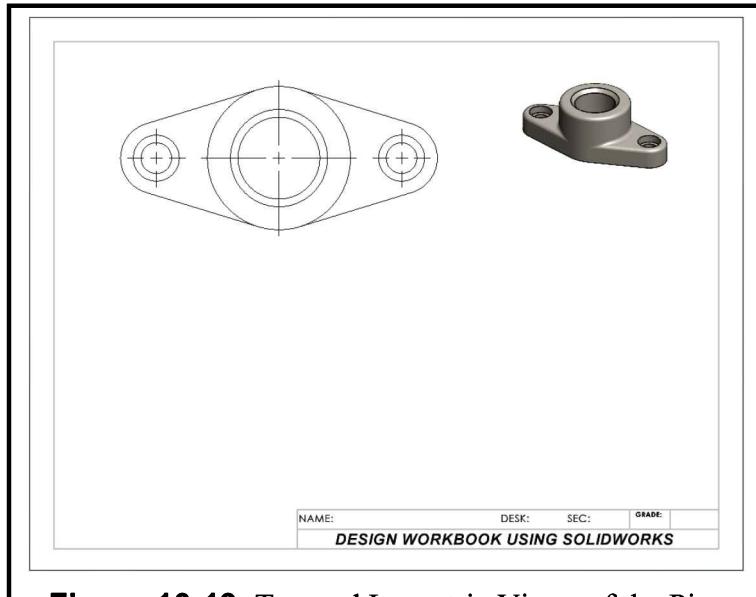


Figure 10-19. Top and Isometric Views of the Pipe Joint.

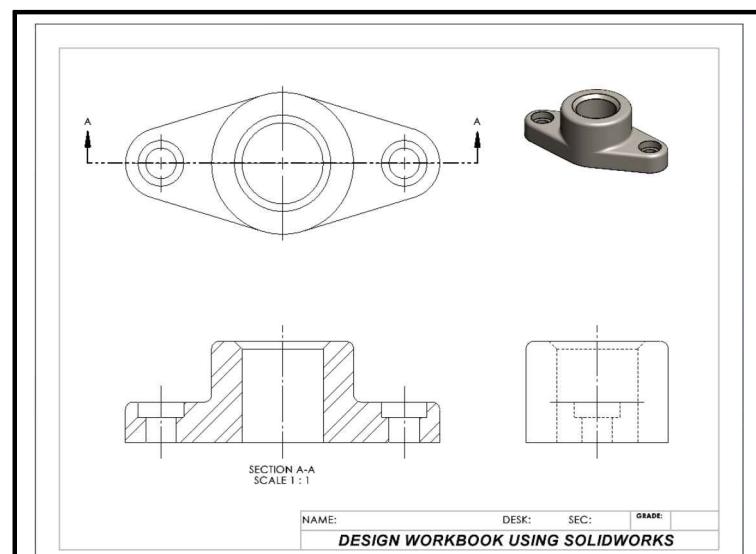


Figure 10-20. Layout of Pipe Joint Drawing.

DIMENSIONING THE DRAWING



From the Annotations tab, select the **Model Items** command. In the Source section select **Entire Model**, turn on the option **Import items into all views** and click **OK**.

When you import dimensions, they are first added to the **Detail** views, then **Section** views, and finally to main views. Move the **28mm** diameter and the **48 mm** dimensions to the **Top** view. Remember to hold down the **Shift key** when you move a dimension to a different view. Delete the dimensions of the counterbore hole (both diameters and the depth). Your 2D drawing should look like **Figure 10-21**.

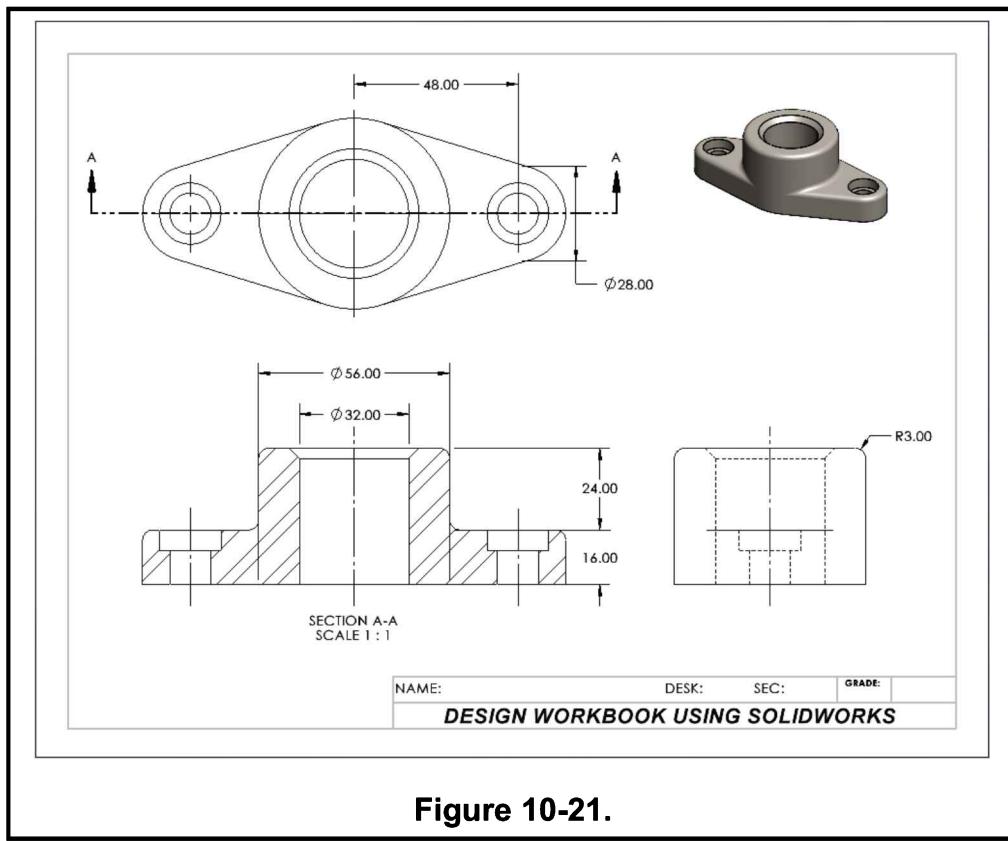
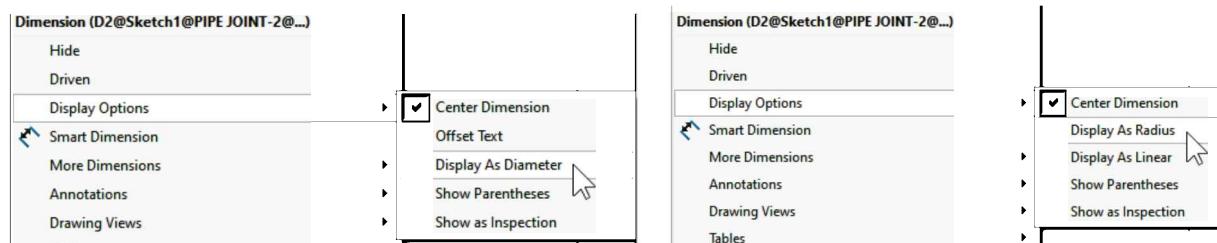
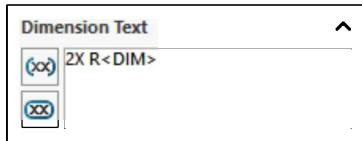


Figure 10-21.

After moving the **28mm** Diameter dimension it will be shown as a linear dimension. To change it to a radial dimension right click on it, and select **Display Options, Display as Diameter**. To show it as a **Radius** dimension, right click on it again and select **Display Options, Display as Radius**.

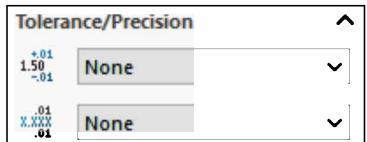




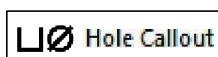
Since this dimension references both sides of the part, select the **R14.00**, and in the dimension's properties add a “**2X**” before “**R<DIM>**” in the **Dimension Text box**. The final dimension should read “**2X R14**”. With the **Smart Dimension** tool, add a dimension between the two counterbore holes.



From the Drop-down **Smart Dimension** command, select **Chamfer Dimension**. In the **Section** view select the chamfer edge, then the vertical edge of the hole and locate the Chamfer dimension.



To change how the chamfer dimension is displayed, select the dimension, and from the properties, under Tolerance/Precision change the **Unit Precision** value to **None**.



Finally, from the Annotation tab select the **Hole Callout** command and click on the outer edge of the counterbore hole in the **Top** view. Locate the annotation to finish. As with the radial dimension, add a “**2X**” in the **Dimension Text box**.

Next use the **Note** command to add a note with title, scale, material, “**ALL FILLETS & ROUNDS -R3**”, and “**UNITS MM**”, as shown in **Figure 10-22**. Change the font and size as needed.

Save your drawing as **PIPE JOINT.slddrw** and print a copy to submit to your lab instructor.

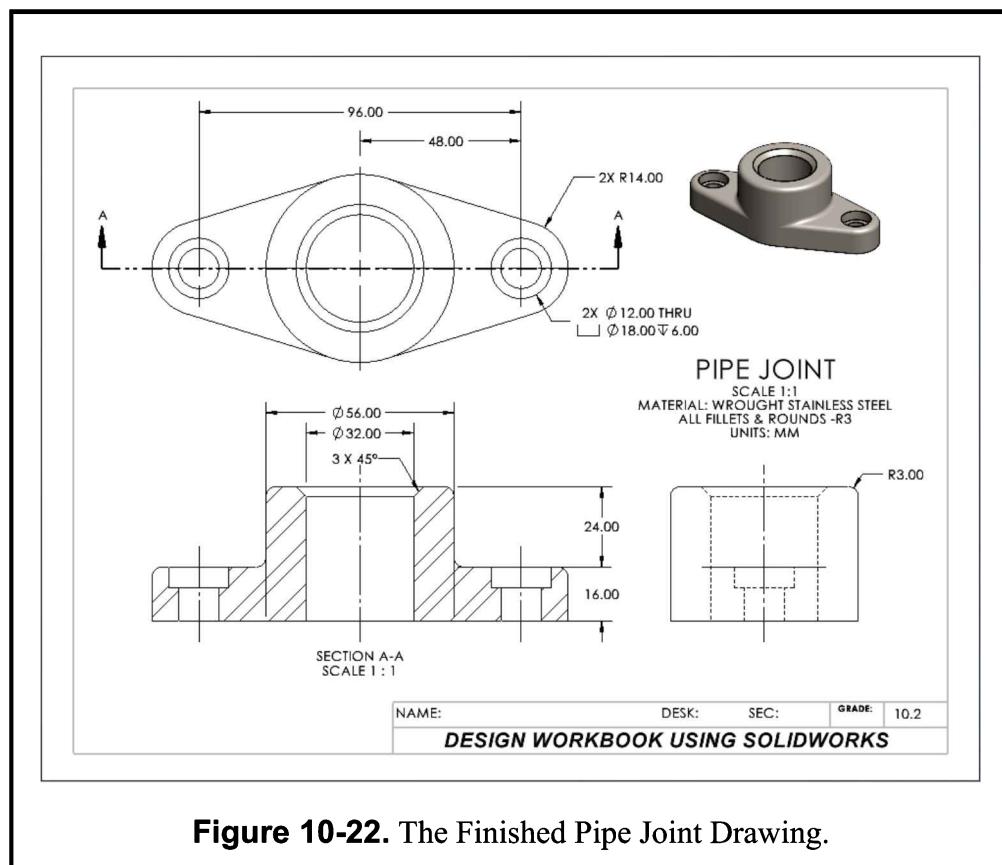


Figure 10-22. The Finished Pipe Joint Drawing.

Exercise 10.3: PEDESTAL BASE Drawing

Quite often in mechanical design, features such as holes are repeated in a regular circular pattern. Examples include flanged pipe joints, inspection cover plates, motor housings, and many others. Circular and rectangular patterns have been introduced in earlier exercises; here, you will create a part with multiple bolt circle patterns and discuss dimensioning practices for the part in a multi-view drawing.

BUILDING THE PEDESTAL BASE

Start a new part using the **METRIC** template and add a new sketch in the **Front Plane** and draw the sketch profile shown in **Figure 10-23**.

Since this sketch will be used to make a **Revolved Boss**, it is helpful to add diameter dimensions to the sketch. To add a diameter dimension, select the centerline and the entity to dimension, and then cross the centerline **BEFORE** locating the dimension; it will be automatically doubled.

While you add the **doubled dimensions**, the mouse cursor will be shown as:



When you finish adding the doubled dimensions, press the **ESC** key to return to the **Smart Dimension** tool and finish adding the height dimensions.

When the sketch is complete, add a **360° Revolved Base** and click **OK**. The resulting part will look like **Figure 10-24**.

Next, create the circular hole patterns in the base. The holes needed can be made using the **Hole Wizard**, which will be covered in the next lab. For this exercise we'll use a regular **Cut Extrude** feature.

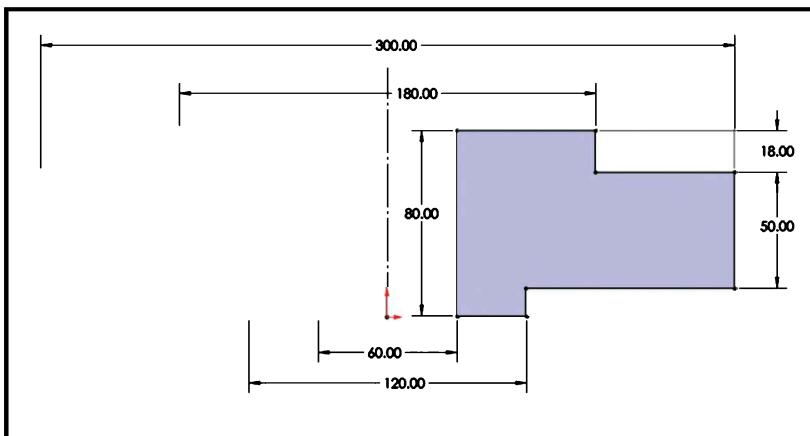


Figure 10-23. Sketch for the Base Revolve.

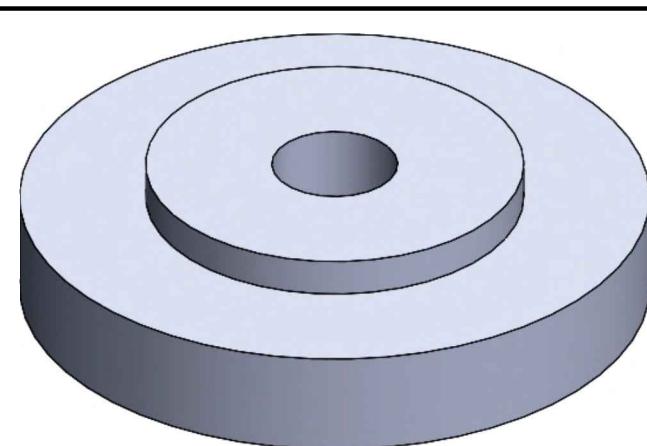
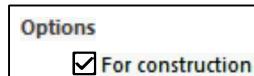


Figure 10-24. Base Revolve.

Change to a **Top** view and add a new **Sketch** in the outer top face of the base feature. Draw two construction lines and a construction circle.



After adding the **240 mm** circle select it and activate the checkbox **For Construction**. Finally dimension it as shown in **Figure 10-25**.

When the sketch is complete select the **Cut Extrude** command and use the **Through All** end condition.

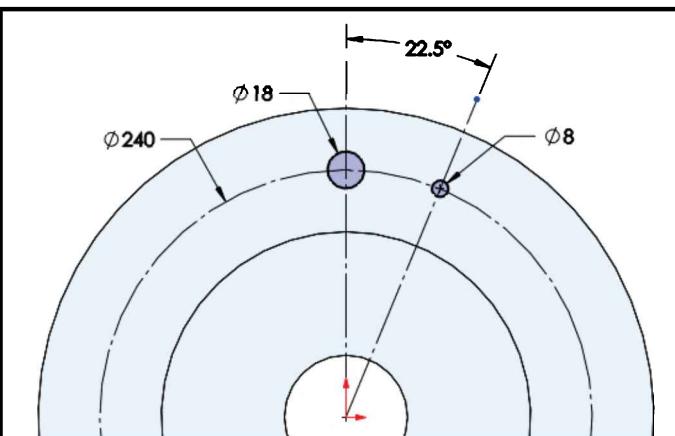


Figure 10-25. Dimensioned Construction Lines.

 **Circular Pattern** From the Linear Pattern command's drop-down list, select the **Circular Pattern** command, select the circular outer edge for the direction, add the **Cut Extrude** feature in the **Features to pattern** selection box, set the number of copies to **8** and use the equal spacing option in **360°**.

To finish the part, add a **5mm Fillet** to the outer edges, except for the holes, as shown in **Figure 10-26**.

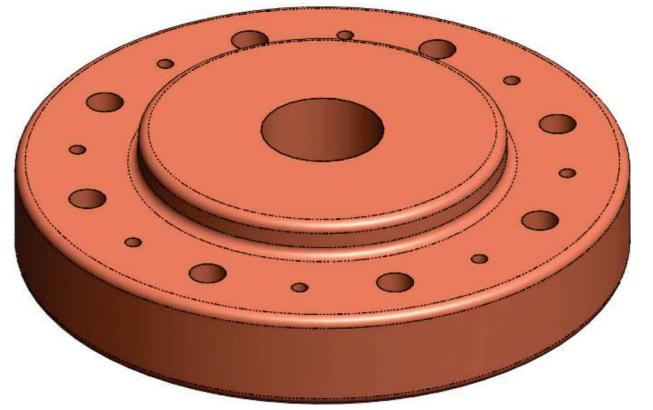


Figure 10-26. Completed Pedestal Base Model.

Set the part's material to **Copper** and save your part as **PEDESTAL BASE.sldprt**.

MAKE A MULTI-VIEW DRAWING

Using the **METRIC** drawing template start a new drawing. Open the **View Palette**, make sure the **Auto-Start Projected view** option is turned **off** and the **Pedestal Base** is selected in the drop-down list.

From the **View Palette** drag a **Top** view and then an **Isometric** view. Select the **Isometric** view and change it to **Shaded with Edges** mode. If needed, right click on the **Top** view, and select **Tangent Edge**, **Tangent Edges Removed**, and change the Top view's scale to 1:4.

From the **View Layout** tab select the **Section View** command, select the **Horizontal** section view option, click in the **Top** view when the section line snaps in the middle of the part, and locate the section view in place of the **Front** view.

Now select the **Section** view, click on the **Projected View** command from the View Layout tab, and locate the projected **Right** on the right side of the sheet. Select the **Right** view and set the Display Style to **Hidden Lines Visible**. Add the **Centerlines** to continue.



Now you need to import the part's dimensions into the drawing. From the Annotations tab, select the **Model Items** command. In the Source section select **Entire Model**, turn on the option **Import items into all views** and click **OK**.

When you import dimensions, they are first added to the **Detail** views, then **Section** views, and finally to main views.

Arrange the dimensions to match the drawing in **Figure 10-27**. Remember to hold down the **Shift key** to move dimensions to a different view. Delete the diameter dimensions of the two holes and change the **60 mm** dimension to display as diameter.

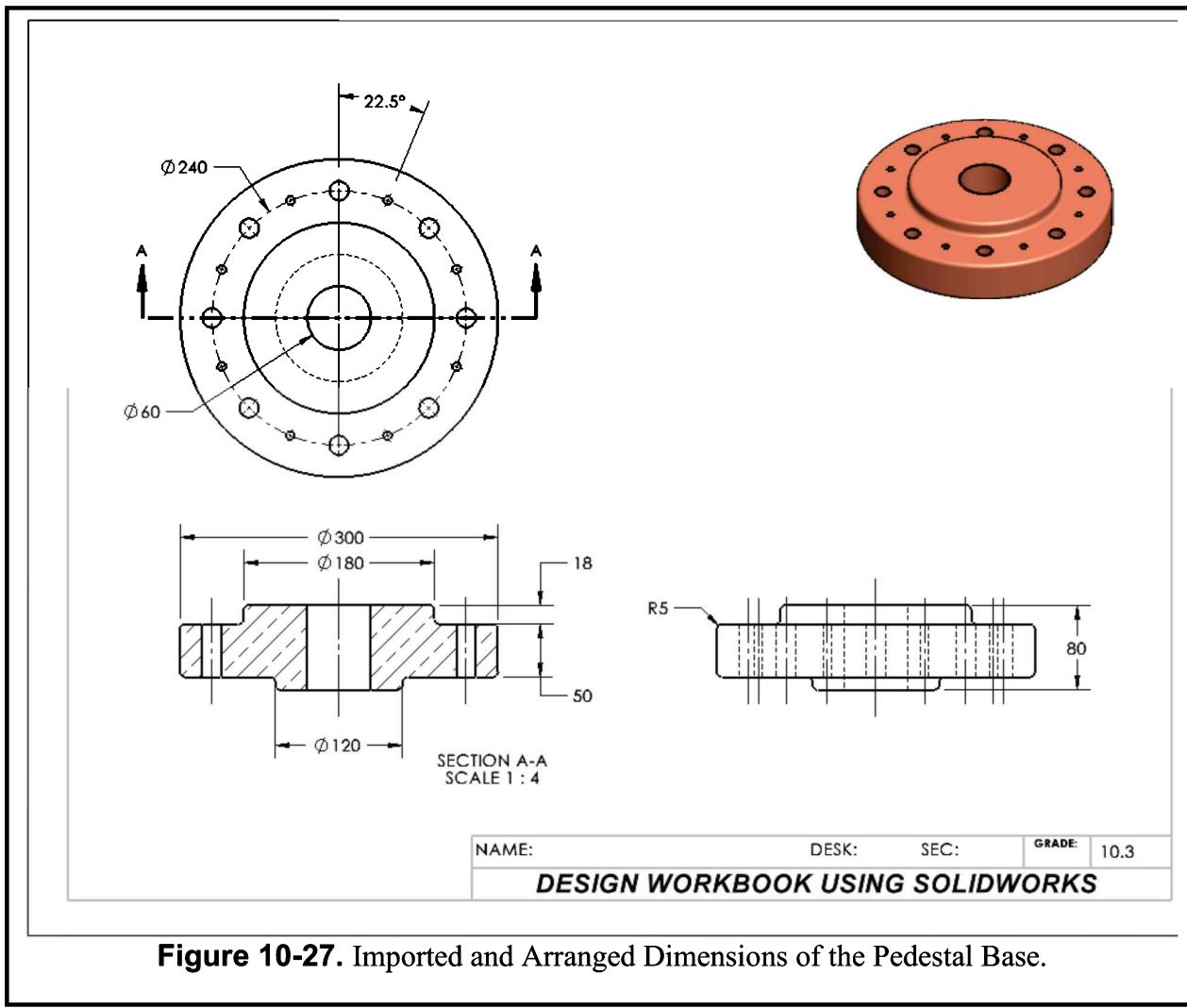
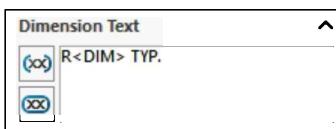


Figure 10-27. Imported and Arranged Dimensions of the Pedestal Base.

Now, add a **centerline** in the **Top** view from the origin to the first small hole to reference the imported **22.5°** dimension.



The **Fillet** dimension is added to a single fillet. To communicate the fillet's dimension is the same for all rounded edges—select the dimension, and in the **Dimension Text** box type “**TYP.**” at the end of the dimension text.

Hole Callout To complete the drawing, select the **Hole Callout** tool and click in both patterned holes; the correct annotation will be added. In the **Dimension Text** of both hole annotations type “**8X**” before the dimension text code.

Next use the **Note** command to add a note with title, scale, material, and “**UNITS MM**”, as shown in **Figure 10-28**. Change the font and size as needed.

Save your drawing as **PEDESTAL BASE.slddrw** and print a copy to submit to your lab instructor.

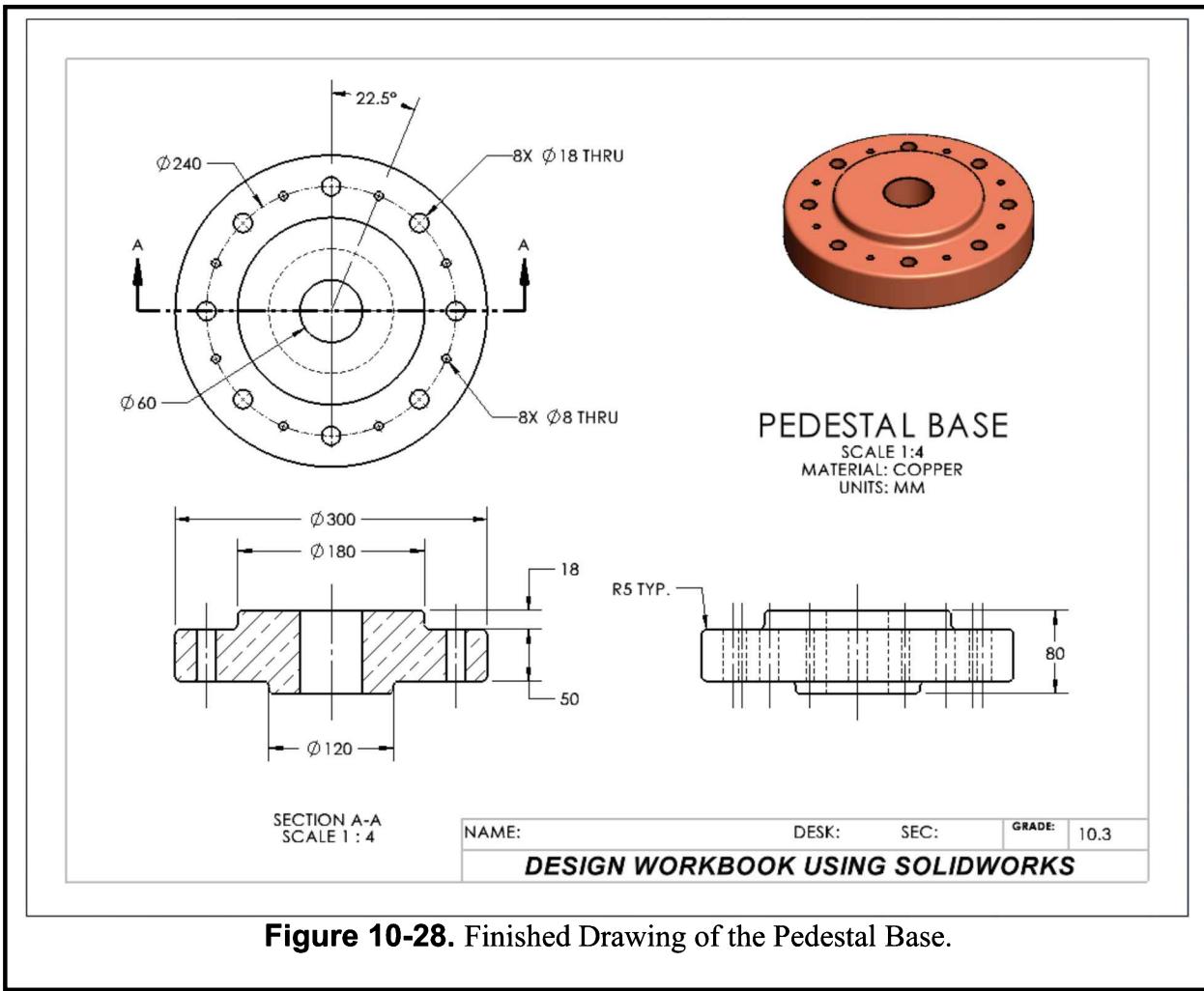


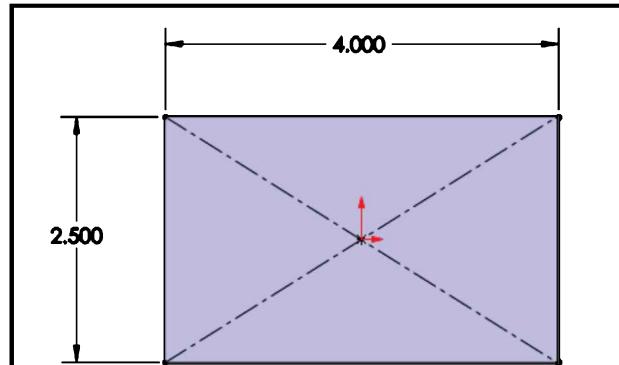
Figure 10-28. Finished Drawing of the Pedestal Base.

Exercise 10.4: TOOLING PAD Drawing

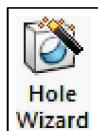
In Exercise 10.4, you will build a solid model of the Tooling Pad, make a drawing with three orthographic views, import the model dimensions and add the necessary annotations to complete the drawing.

BUILDING THE TOOLING PAD MODEL

Start a new part using the INCH template, add a new sketch in the **Top** plane, draw the profile in **Figure 10-29** using the **Center Rectangle** tool starting at the origin, and dimension as shown. Then **Extrude** the profile **0.725"** up.



Using the Hole Wizard



Now you will learn how to use the **Hole Wizard** to add holes on the Tooling Pad. Select the **Hole Wizard** command from the Features tab. In the

Figure 10-29. Base Feature Sketch.

Hole Specification select the Hole option, and set the following parameters, as indicated in **Figure 10-30**:

Standard: **ANSI Inch**
 Type: **Fractional Drill Sizes**
 End Condition: **Through All**
 Hole Diameter: **3/8 in**

After the hole parameters have been set, select the **Positions** tab; this is the second part of the Hole Wizard as shown in **Figure 10-31**.

Change to a **Top** view and select the top face of the base feature. In this step of the **Hole Wizard** we are editing a sketch, and you will use a sketch **Point** to define the location of each hole.

After selecting the face, the sketch **Point** tool is automatically selected. Click to add three points in the top face and dimension them as shown in **Figure 10-32**. At each **Point** location you will see a preview of the hole. When you finish dimensioning the locations click **OK** to finish.

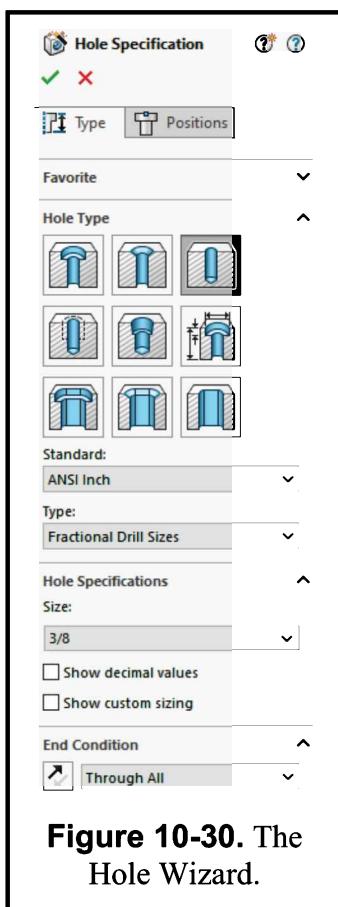


Figure 10-30. The Hole Wizard.

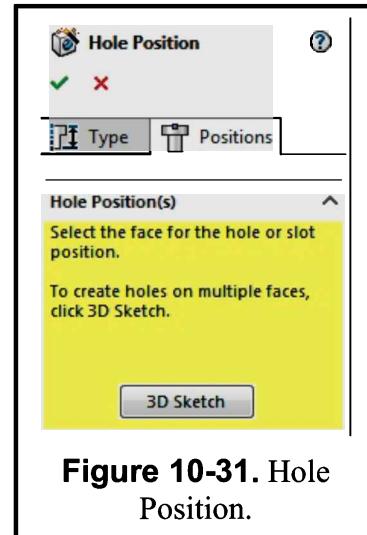


Figure 10-31. Hole Position.

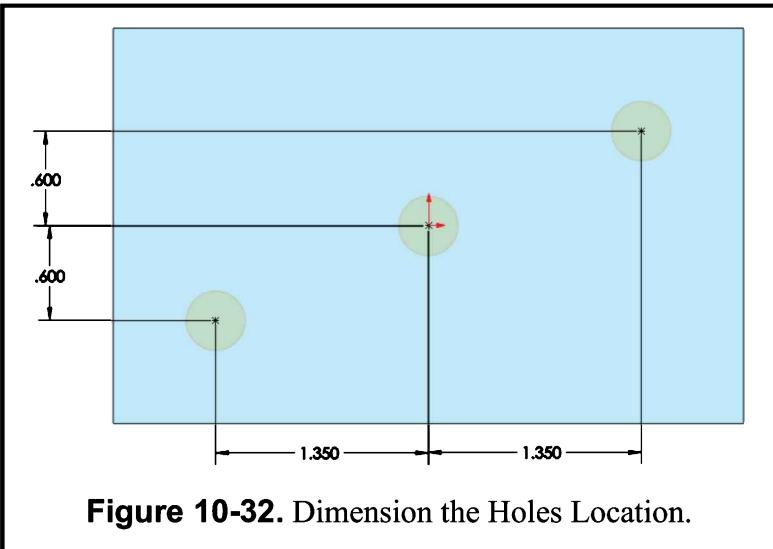


Figure 10-32. Dimension the Holes Location.

Add the Counterbore Holes



Now you will add counterbore holes using the **Hole Wizard**. In the **Hole Specification** select the **Counterbore** option, and set the parameters indicated in **Figure 10-33**.

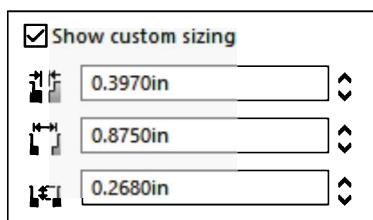
Standard = **ANSI Inch**

Type = **Hex Bolt**

Size = **3/8 in**

End Condition = **Through All**

Note: The option **Show Custom Sizing** allows you to customize the counterbore hole dimensions.



Now select the **Positions** tab to locate the counterbore holes. Change to a **Top** view and select the top face of the base feature as before.

After the top face is selected, the sketch **Point** tool is activated. Add two points in the top face and dimension them as shown in **Figure 10-34**. When dimensioned click **OK** to finish.

The part should look like **Figure 10-35**.

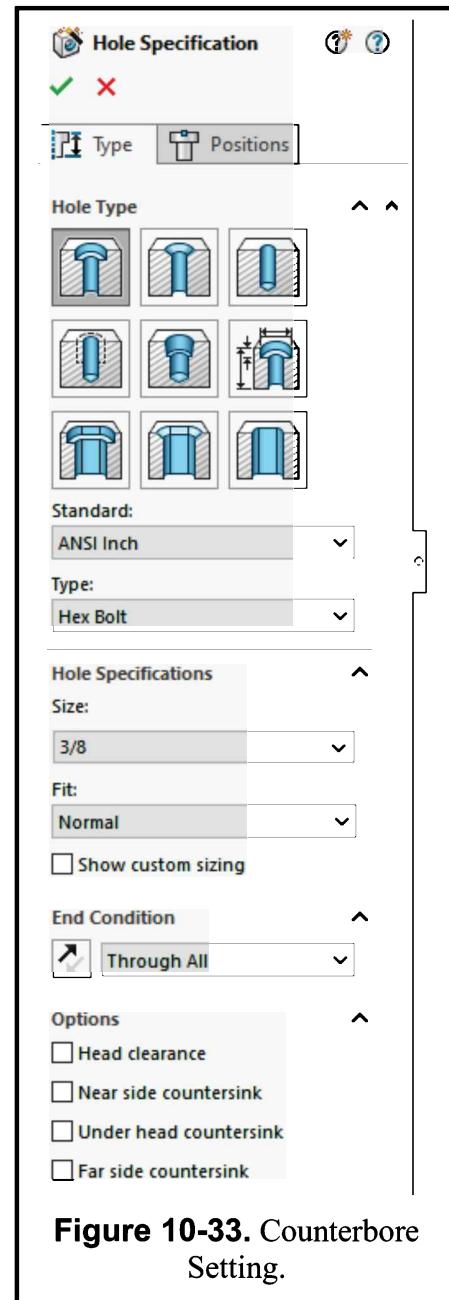


Figure 10-33. Counterbore Setting.

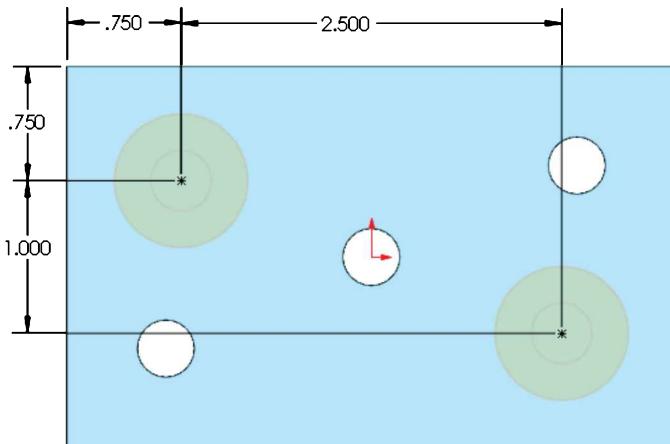


Figure 10-34. Locating the Counterbore Holes.

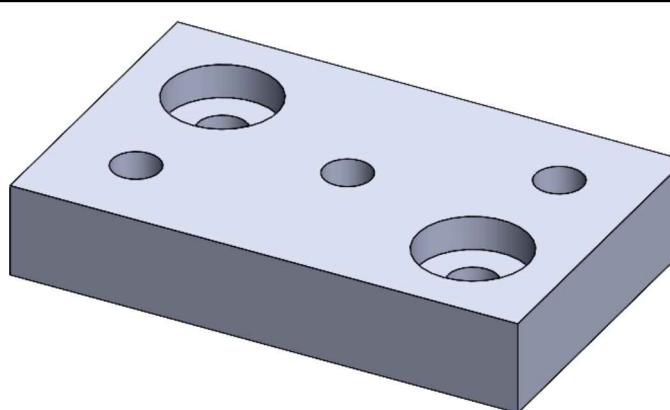


Figure 10-35. Counterbore Holes.

Adding a Countersink

Now you will add a countersink hole using the **Hole Wizard**. In the **Hole Specification** select the **Countersink** option, and set the options indicated in **Figure 10-36**.

Standard = **ANSI Inch**

Type = **Flat Head Screw**

Size = **1/4 in**

End Condition = **Up to Surface**

*In the **Up to Surface** end condition select the inner face of the middle hole.

Now select the **Positions** tab. Change to a **Front** view and select the front face of the base feature.

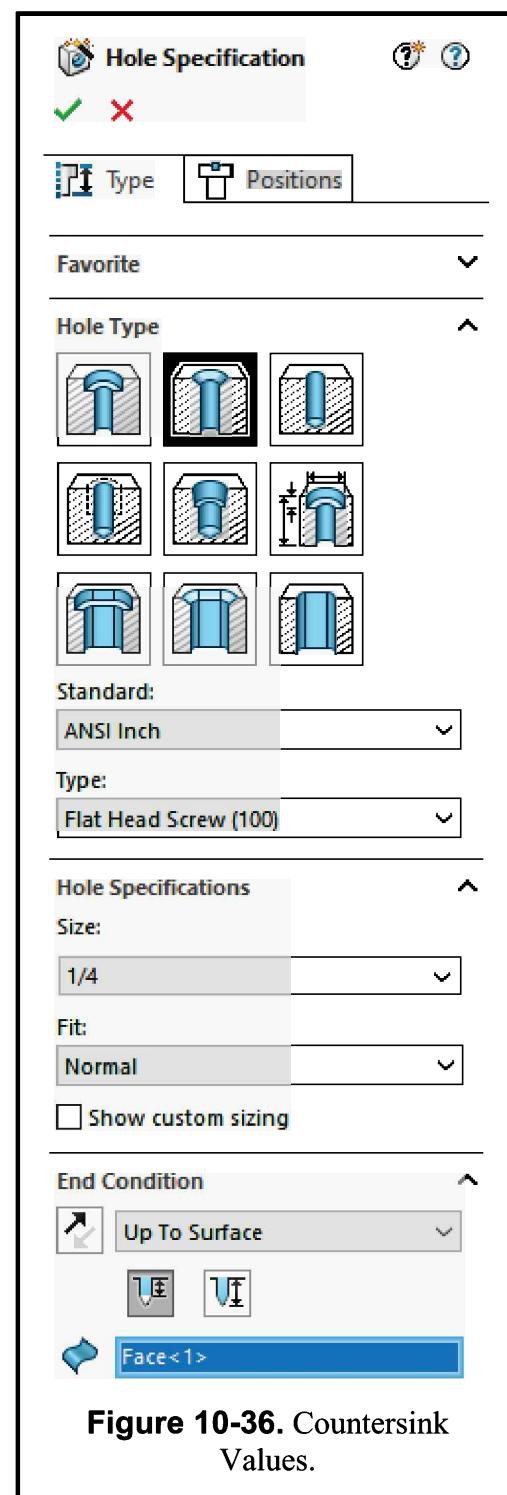


Figure 10-36. Countersink Values.

Click to add a **Point** in the **Front** face, dimension it and add a **Vertical** relation to the origin as shown in **Figure 10-37** to fully define the sketch. Click **OK** to complete the **Hole Wizard**.

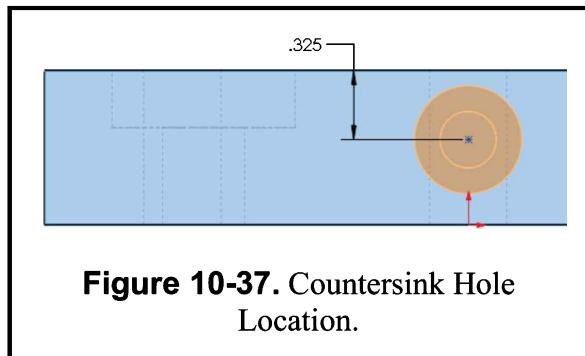


Figure 10-37. Countersink Hole Location.

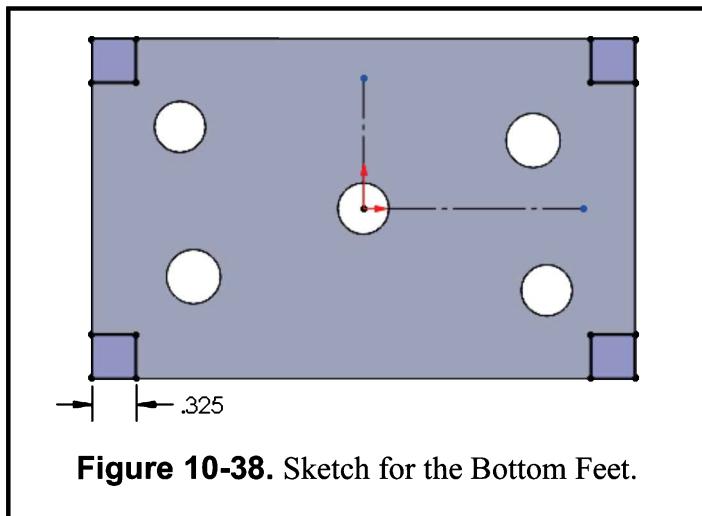


Figure 10-38. Sketch for the Bottom Feet.

Adding the Bottom Base Feet

Change to a **Bottom** view to add the four corner feet to the Tooling Pad. Add a sketch in the bottom face and draw the profile as shown in **Figure 10-38**.

Draw a rectangle in a corner and make two perpendicular lines **Equal** to make a square.

Use the **Mirror Entities** command about a Vertical and Horizontal centerline to add the other three corners.

Extrude the sketch **0.125"**, add a **0.0625"** **Fillets** to the outer top edges and the outer vertical edges as shown in **Figure 10-39**.

Set the material to **Titanium Ti-8Mn, Annealed** from the **Titanium Alloys** library, and save your part as **Tooling Pad.sldprt**.

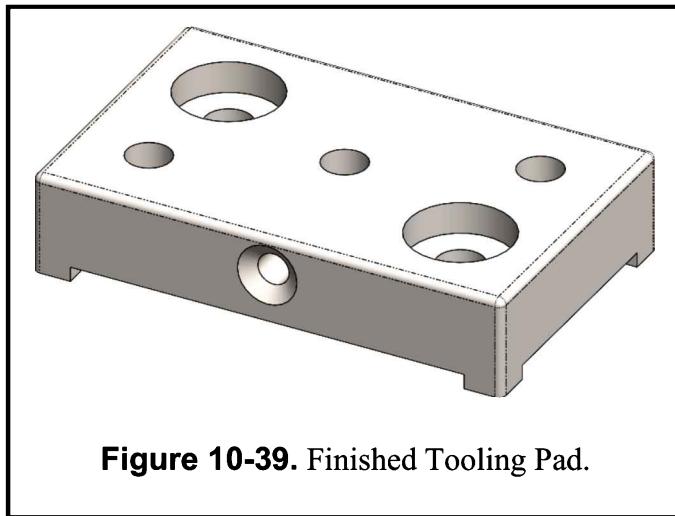


Figure 10-39. Finished Tooling Pad.

MAKE A MULTI-VIEW DRAWING

Start a new drawing using the **Inch** template and change the sheet's scale to **1:1**. Open the **View Palette**, make sure the **Auto-Start projected view** option is turned **on** and the **Tooling Pad** is selected in the drop-down list.

From the **View Palette** drag the **Front** view, and project a **Top** and an **Isometric** view. Select the **Isometric** view, change it to **Shaded with Edges** mode and change the scale to **1:2**. If needed, right click on the **Front** and **Top** view and select **Tangent Edge, Tangent Edges Removed**. Your drawing will look like **Figure 10-40**.

ADD A SECTION VIEW

From the **View Layout** tab select the **Section View** command. Activate the **Vertical** section view option, locate the section line in the middle of the **Front** view, and place the section view to the right as shown in **Figure 10-41**.

ADD CENTERLINES AND CENTER MARKS

From the Annotations tab, select the **Centerline** command and add the missing centerlines to the Top, Front, and Section views. If missing, also add the missing **Center Marks**.

DIMENSIONING THE DRAWING



Now you need to import the part's dimensions into the drawing.

From the **Annotations** tab, select the **Model Items** command. In the Source section select **Entire Model**, turn on the option **Import items into all views**.



Activate the **Hole Callout** option and click **OK** to add the dimensions.

The **Hole Callout** option adds the correct annotations to **Hole Wizard** features, including the number or instances.

In this case, the dimensions added fully annotate the drawing. There are only a few changes to make, like moving dimensions to a different view (hold **Shift** to move dimensions). In the bottom feet, since all sides are equal, only one dimension is needed; in this case add a **TYP** text after the **0.325"** dimension. Also, a dimension's arrows can be **reversed**. Select the dimension and click in the arrow to reverse it, as shown in **Figure 10-42**. Add any missing dimensions as needed to finish.

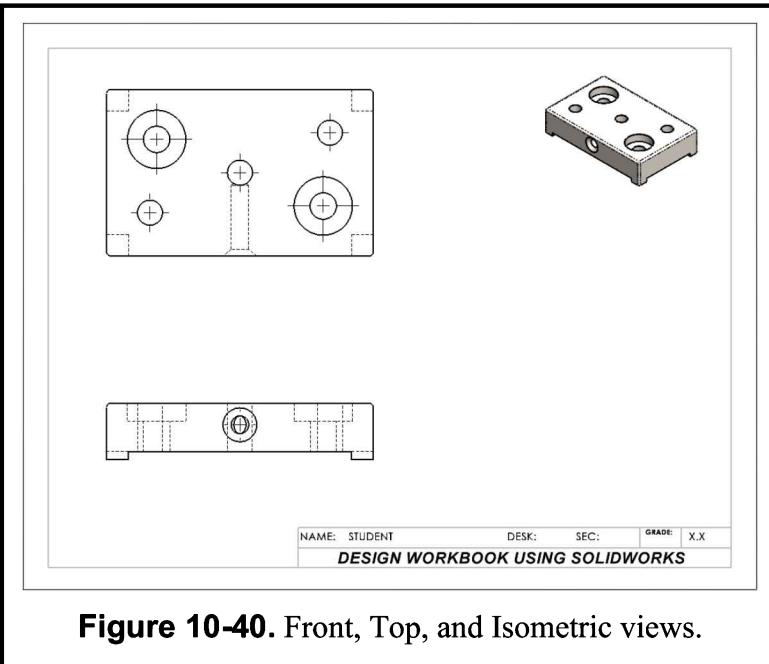


Figure 10-40. Front, Top, and Isometric views.

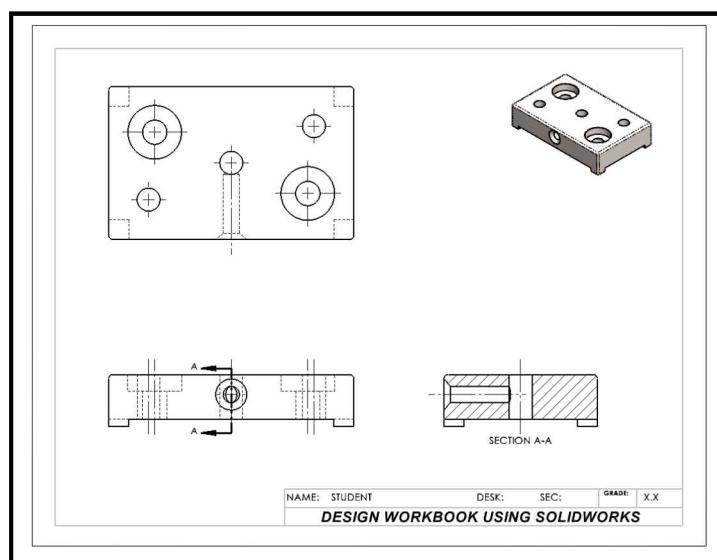


Figure 10-41. Section View of the Tooling Pad.

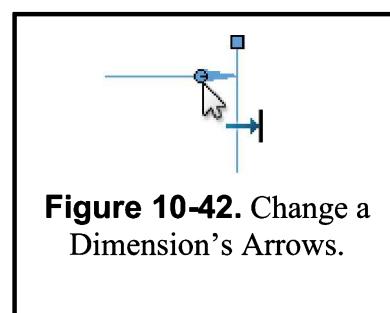


Figure 10-42. Change a Dimension's Arrows.

Finally add a note to the drawing including the title, scale, material, and “**ALL FILLETS AND ROUNDS R-0.0625”**”, as shown in **Figure 10-43**. Change the font and size as needed.

Save your drawing as **TOOLING PAD.slddrw** and print a copy to submit to your lab instructor.

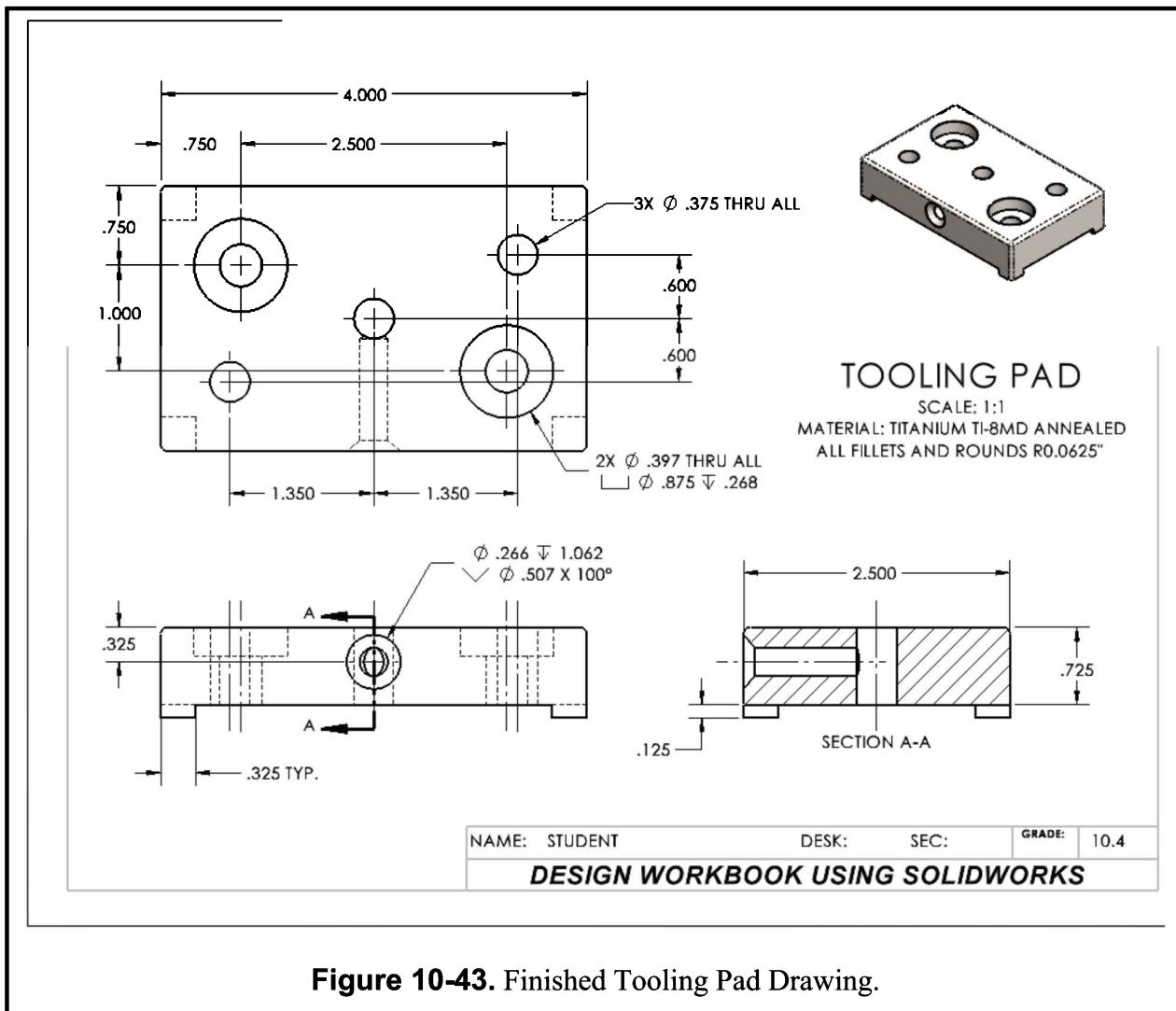
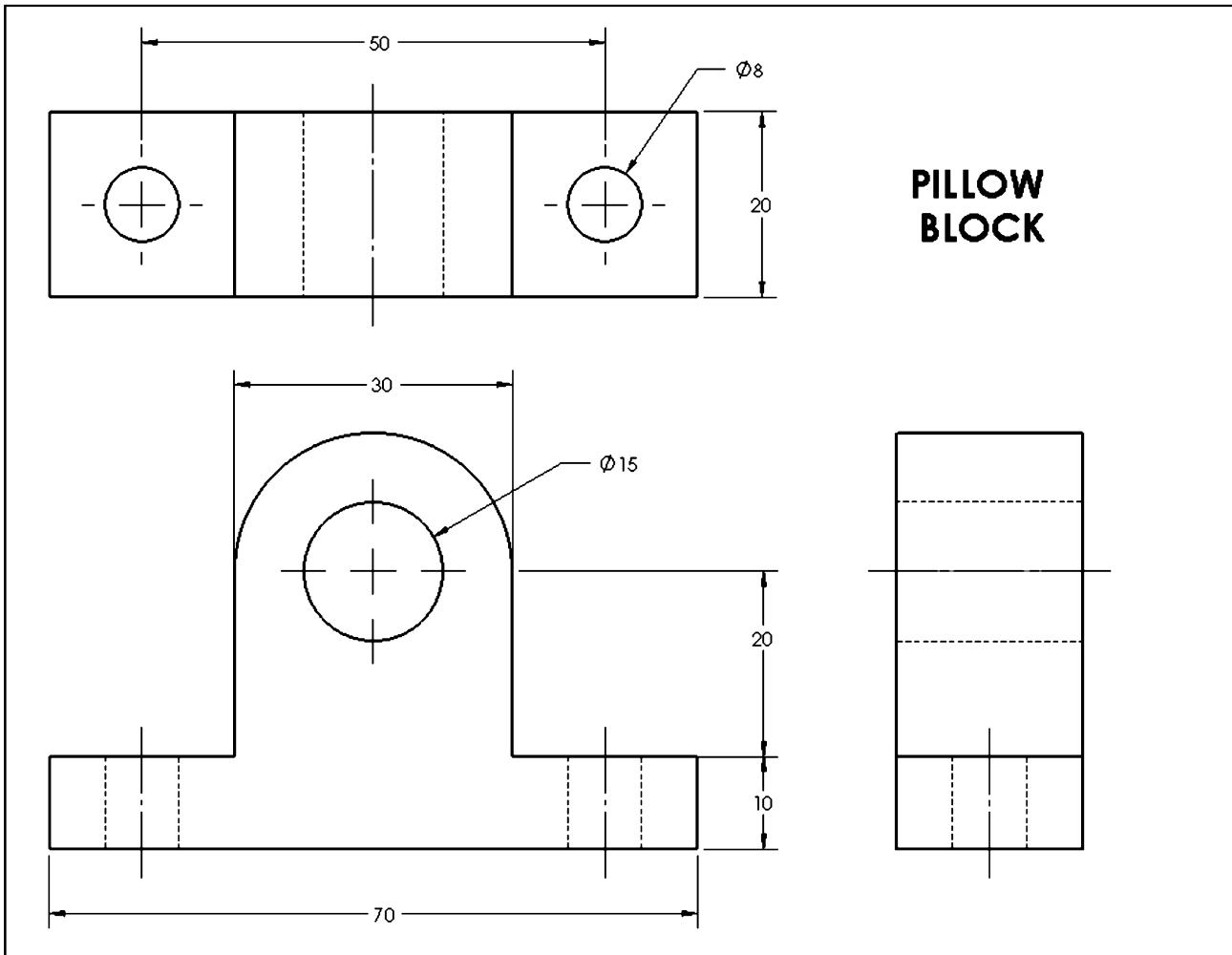


Figure 10-43. Finished Tooling Pad Drawing.

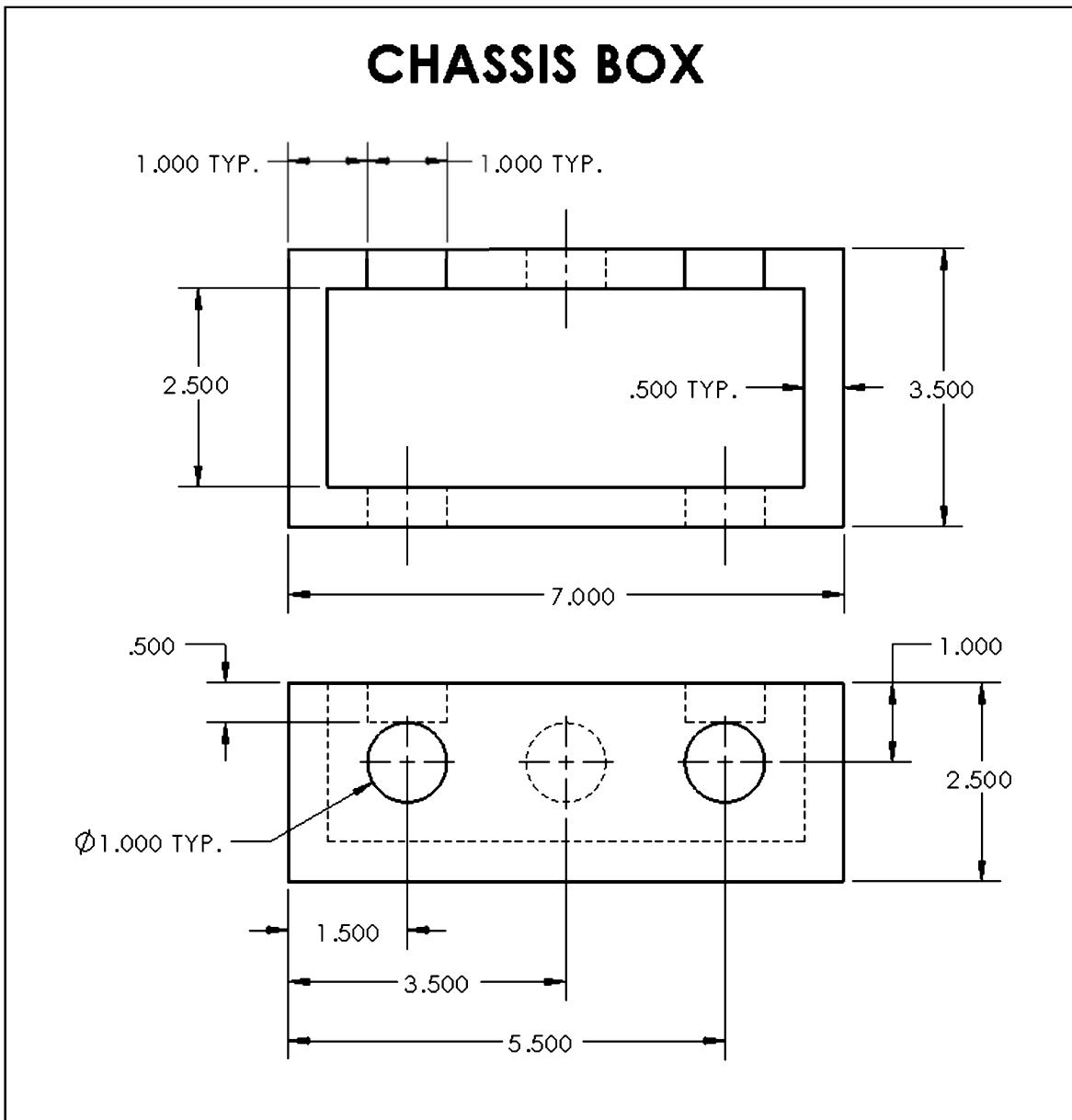
Supplementary Exercise 10-5: PILLOW BLOCK

Build a solid model of the figure below. Make a detail drawing and Dimension it. Insert a small Isometric of the part in the upper right-hand corner of the sheet. Provide the proper Titles, Scale and other pertinent notes. Drawing dimensions are in Millimeters.



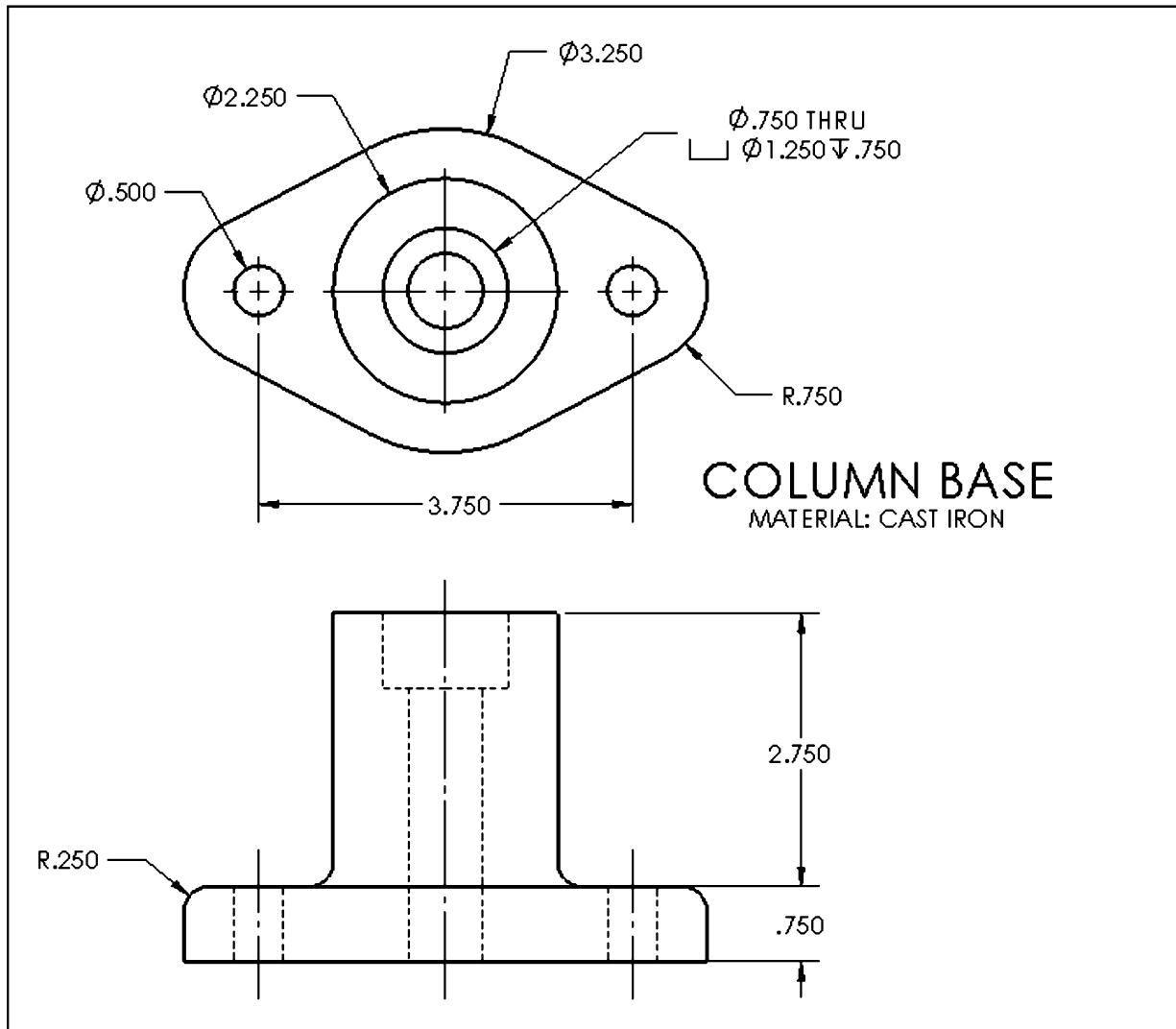
Supplementary Exercise 10-6: CHASSIS BOX

Build a solid model of the figure below. Make a detail drawing and Dimension it. Insert a small Isometric of the part in the upper right-hand corner of the sheet. Provide the proper Titles, Scale and other pertinent notes. Drawing dimensions are in Inches.



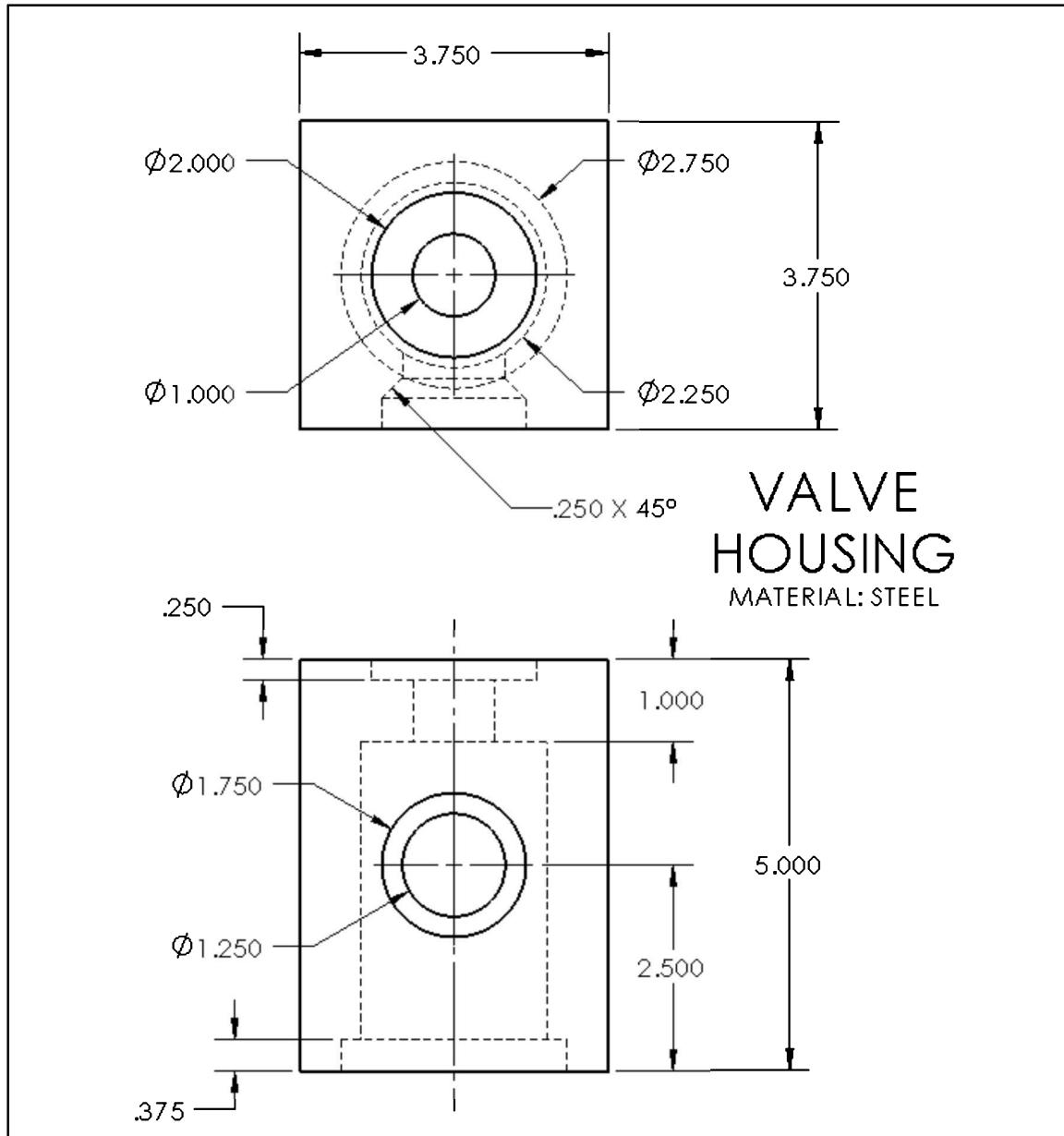
Supplementary Exercise 10-7: COLUMN BASE

Build a solid model of the figure below. Make a detail drawing and Dimension it. Instead of a Front view, add a Horizontal section view of the Top view. Add an Isometric of the part in the upper right-hand corner of the sheet. Provide the proper Titles, Scale, and other pertinent notes. Drawing dimensions are in Inches.



Supplementary Exercise 10-8: VALVE HOUSING

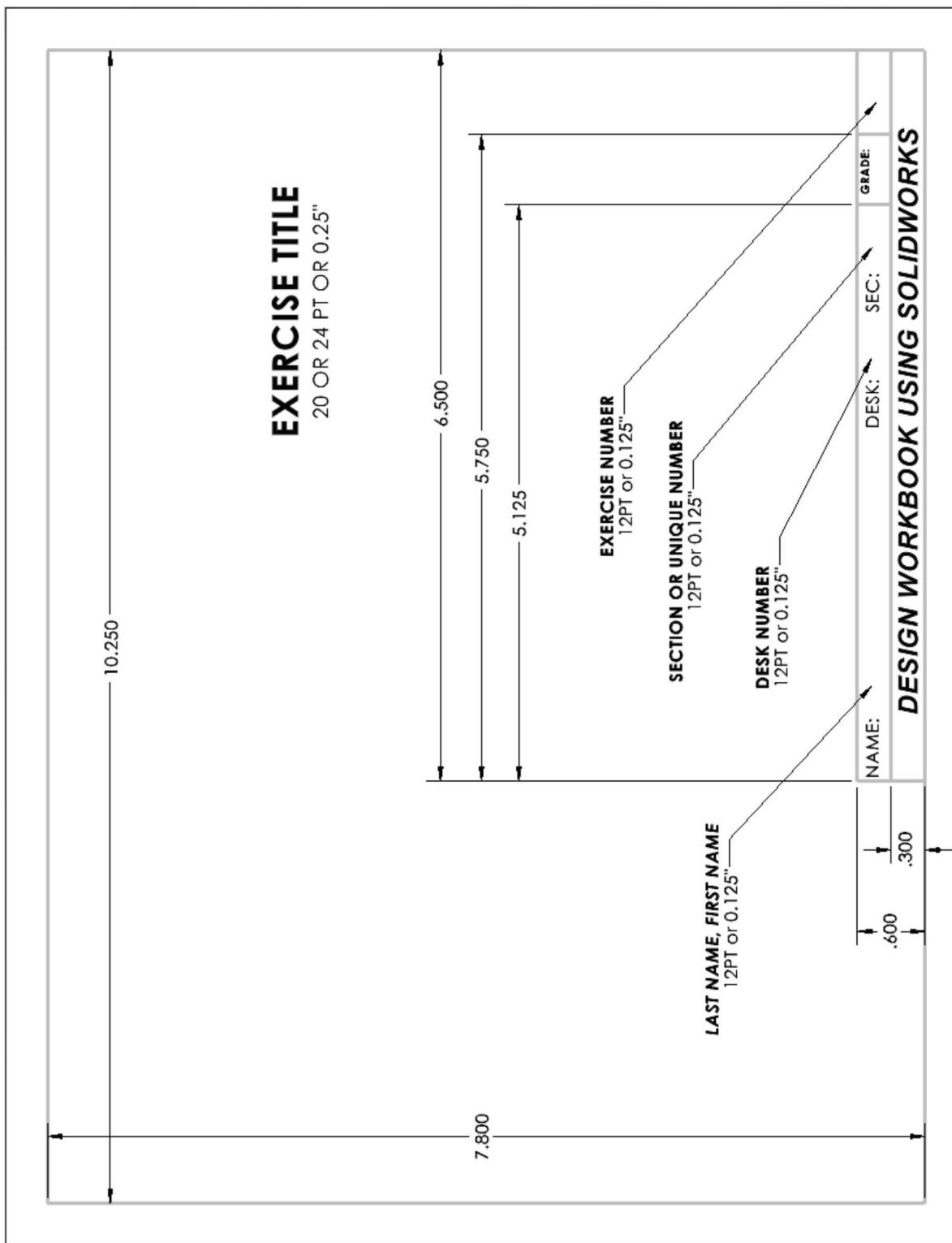
Build a solid model of the figure below. Make a detail drawing and Dimension it. Add a Section view in place of the Front view, and project the Right view from the Section view. Add an Isometric view in the upper right-hand corner of the sheet. Provide the proper Titles, Scale, and other pertinent notes. Drawing dimensions are in Inches.



NOTES:

APPENDIX A

DRAWING SHEET TEMPLATE



NOTES:

Design Workbook Using SOLIDWORKS® 2020

Design, Detailing, Assembly & Analysis Basics

- This edition features many new enhancements using SOLIDWORKS 2020
- An exercise-based workbook using step-by-step tutorials teaches you to use SOLIDWORKS 2020
- Designed for use in undergraduate engineering and pre-college courses
- Covers modeling, finite element analysis, assembly modeling, kinematic simulation, rapid prototyping and projecting engineering drawings
- Incorporates the principles of engineering graphics into lessons

Description

Fully revised and refreshed for SOLIDWORKS 2020, Design Workbook Using SOLIDWORKS 2020 is an exercise-based book that guides you through a series of easy to understand, step-by-step tutorials that cover basic SOLIDWORKS commands. The 2020 edition includes updated SOLIDWORKS processes and methods to create models more efficiently than ever before. The intended audience is undergraduate engineering majors, but it can also be used in pre-college engineering courses. The engaging and straightforward lab exercises in this workbook are also ideal for self-learners.

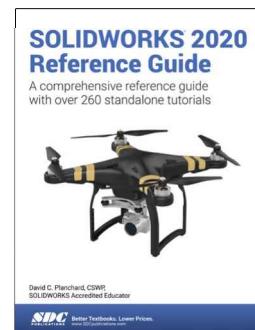
The text takes an educational approach where you learn through repetition, starting with simple models, and introducing more complex models and commands as the book progresses, leading you to create assemblies, make Finite Element Analyses, detail manufacturing drawings, complete dynamic simulations, and learn the basics of rapid prototyping. The principles of engineering graphics are also incorporated into the lessons throughout the text.

The commands and functions learned throughout this book will help a new user understand their use, how to apply them in different situations, and design ever more complex components.

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