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# Design Workbook Using **SOLIDWORKS® 2026**

Design, Detailing, Assembly & Analysis Basics



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



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# Design Workbook Lab 2

## Advanced 2D Sketching

### ADVANCED 2D SKETCHING

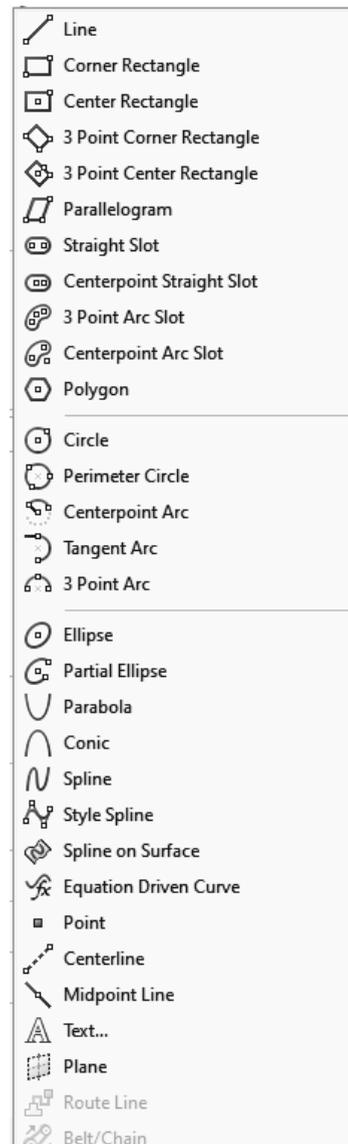
In the first chapter you learned some of the basic 2D sketch tools available in SOLIDWORKS. The first exercises reinforced how to draw a **Line**, **Circle**, **Rectangle**, **Arc**, **Polygon**, **Centerline**, and **Spline**. You also learned how to modify a 2D sketch using **Dimensions**, as well as 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 here, 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 the exercises in this book.



## SKETCH TOOLS MENU

All the 2D sketch editing tools are included in the menu **Tools, Sketch Tools**, and just like sketch entities, the most 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.

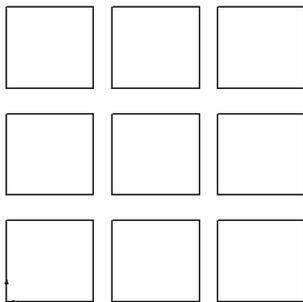
**Dynamic Mirror** dynamically mirrors new sketch geometry about a centerline. First select the entity about which to mirror, activate the Dynamic Mirror command, and draw the sketch entities to be mirrored.

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

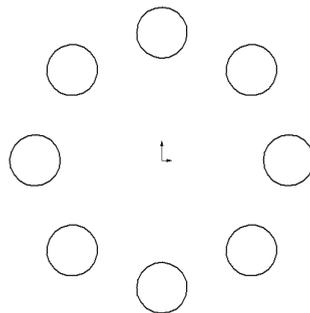
**Construction Geometry** converts sketch entities to construction geometry and vice versa.

**Linear Sketch Pattern** creates a linear or rectangular pattern of identical entities.

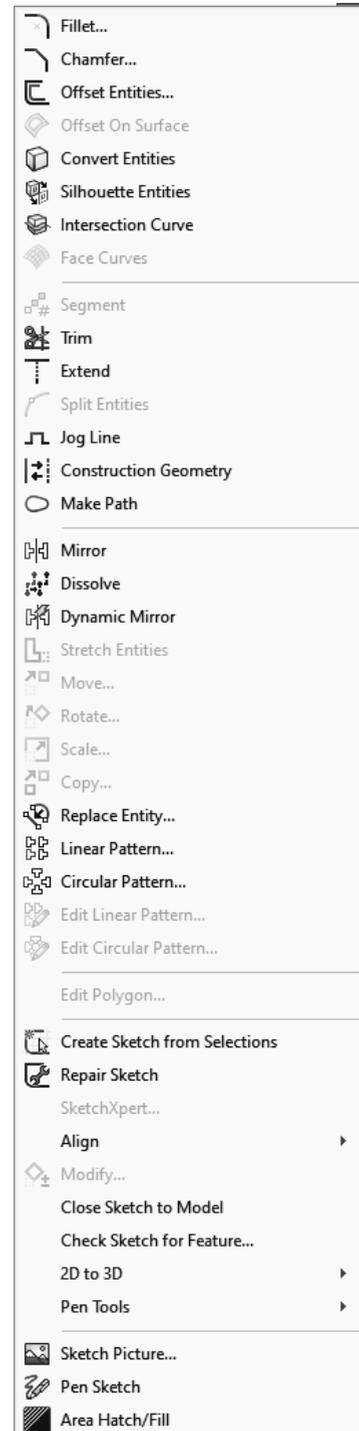
**Circular Sketch Pattern** creates a radial (or polar) pattern of identical entities around a center point.



Linear Sketch Pattern



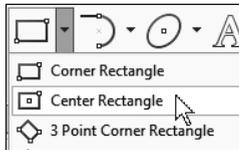
Circular Sketch Pattern



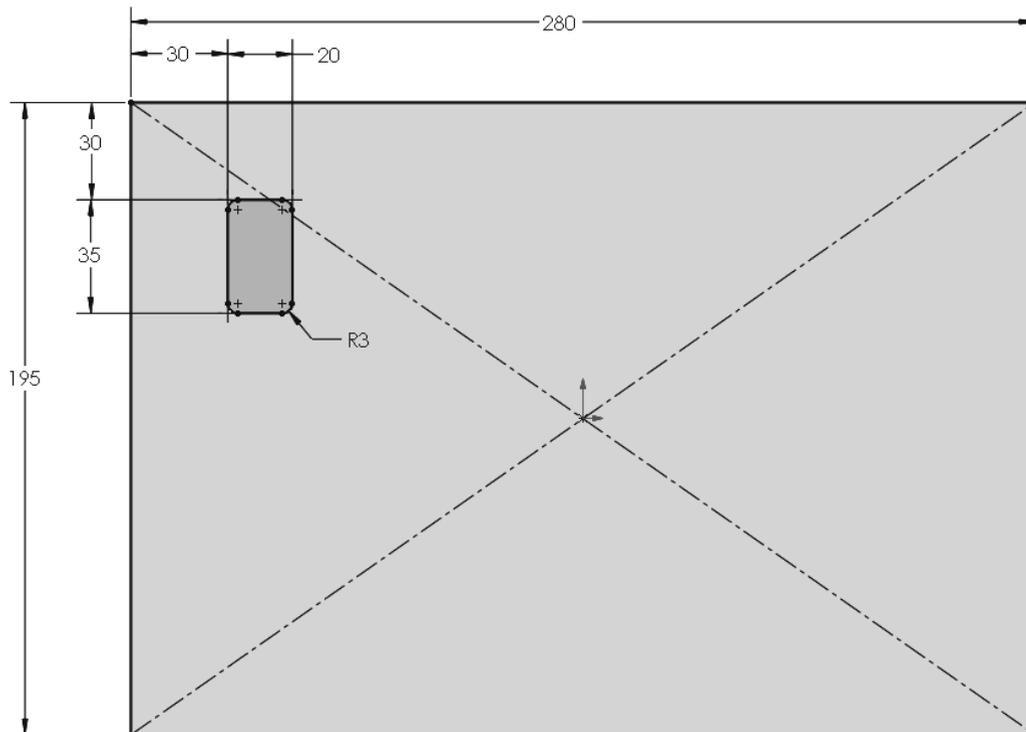
## Exercise 2.1: METAL GRATE

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**.



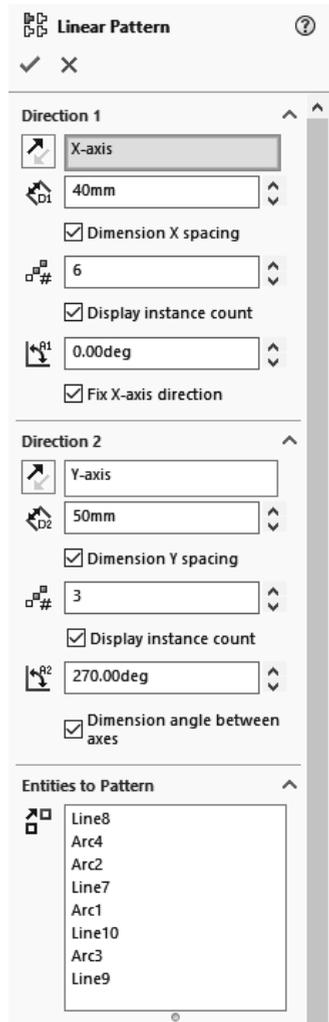
Draw the rectangle starting at the **origin** and dimension it **280 mm** wide by **195 mm** high. Next, select the **Rectangle** tool, draw the initial rectangular slot and dimension as shown next. Once the rectangles are dimensioned, your sketch will be fully defined (geometry will change from Blue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





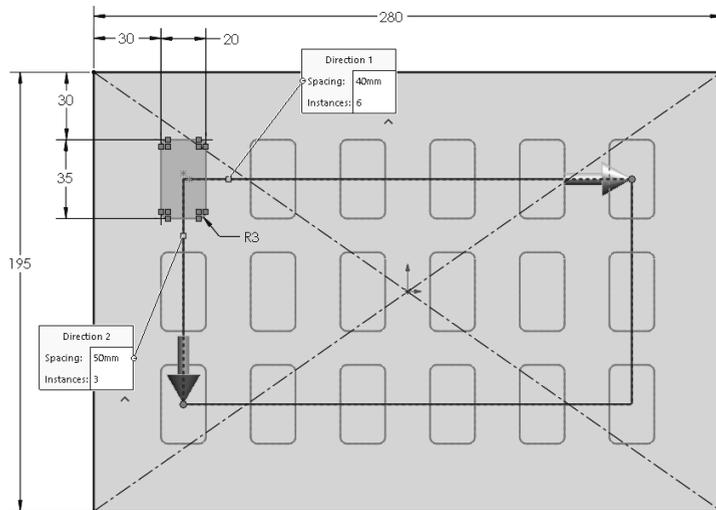
Linear Sketch Pattern

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.



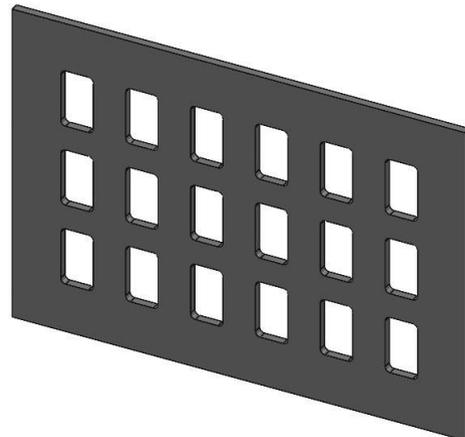
Change the settings for the grate's rectangular pattern as shown. The "Direction 1" is horizontal and will have **6** instances spaced **40 mm** at **0** degrees. To activate the "Direction 2" options, you have to change the number of instances to **3** and set the vertical spacing to **50 mm**. The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

As you change the pattern parameters, notice the pattern **Preview** is shown. For "Direction 2" you will have to select the Reverse Direction button for the pattern to go down. When finished, click the **OK** button to complete the pattern.



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.



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.

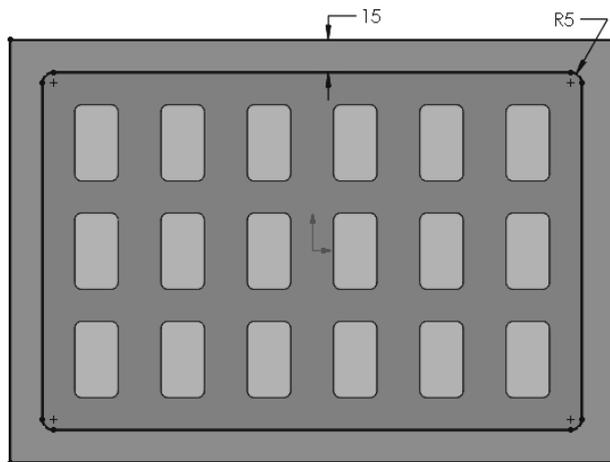
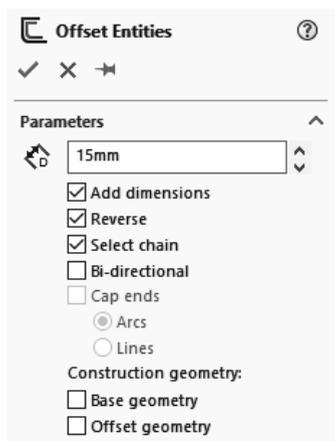
Notice the lines are all black; each projected entity is automatically added an **On Edge** geometric relation to the projected edges. This means that if the previous geometry is changed, the projected sketch entities will update accordingly.



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.

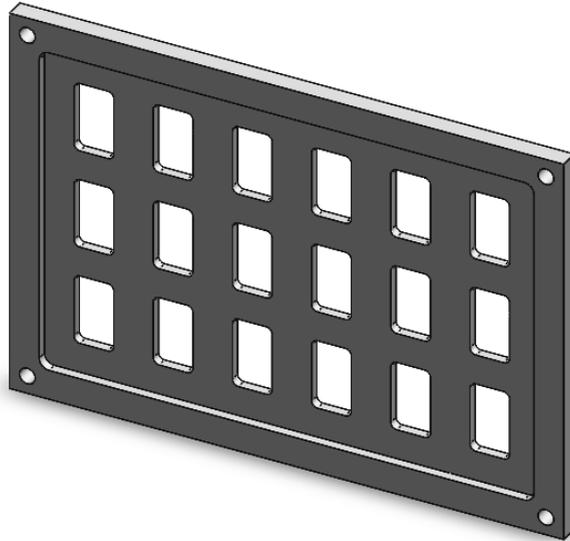
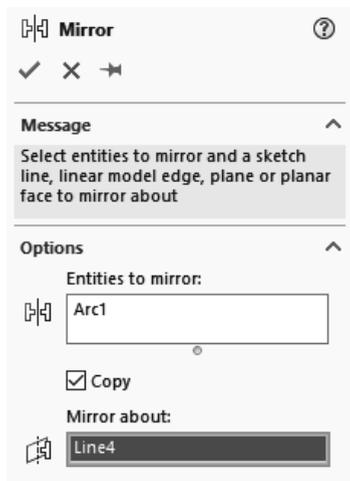
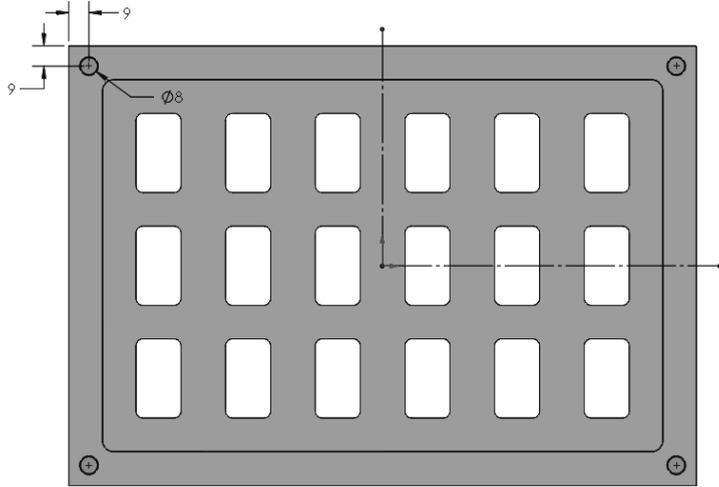


Before extruding the sketch, it is a good idea to change your view orientation to an **Isometric** view; this way it will be easier to see which direction the extrusion is going. Select the **Extruded Boss** command from the Features tab, set the end condition to **Blind** and change the extrusion depth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



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. Next use the **Mirror Entities** again and this time select the two circles and mirror them about the horizontal centerline.

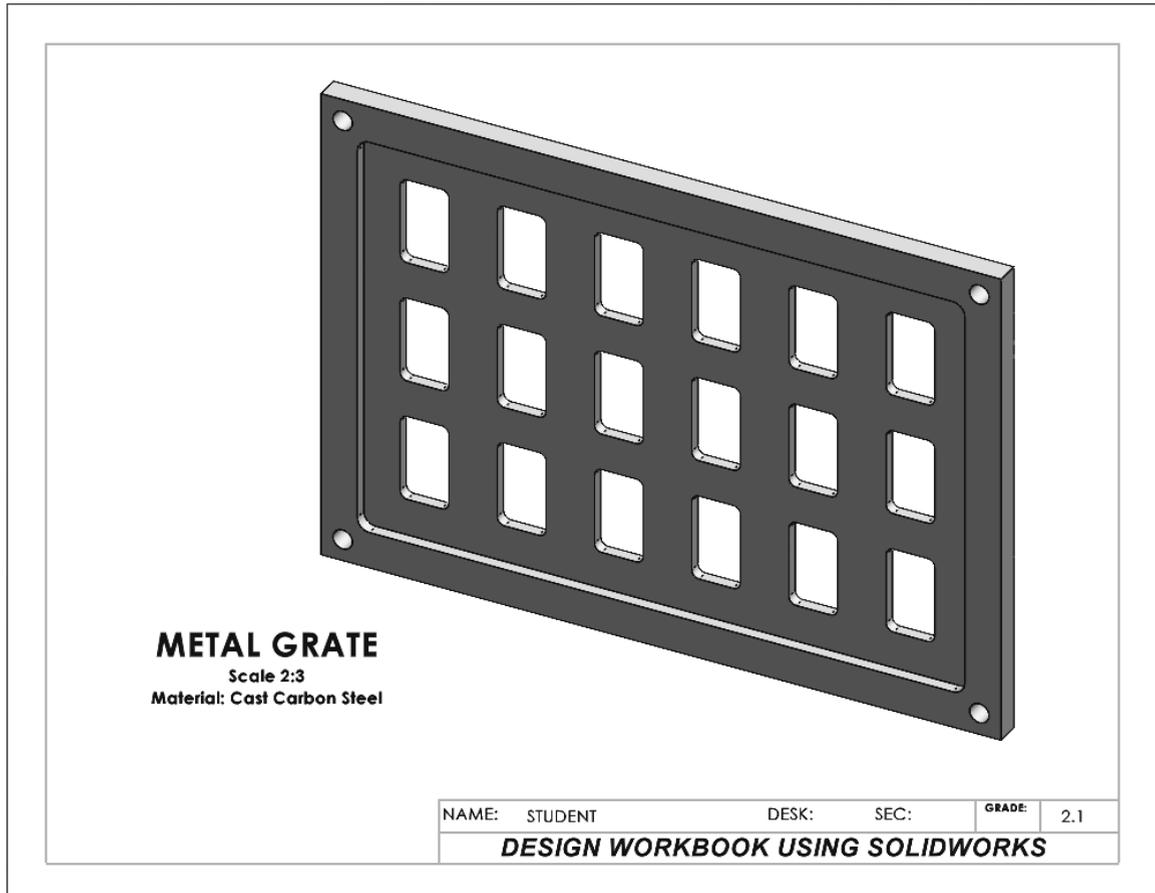


Change the model view's orientation to an Isometric. From the **Features** tab select **Extruded Cut**, set the End Condition to **Through all** and click OK to finish. The four corner attachment holes are now created on the grate.

The part is now complete, and you can view the lip feature more clearly by rotating the model.

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.

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



## 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.

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



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



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

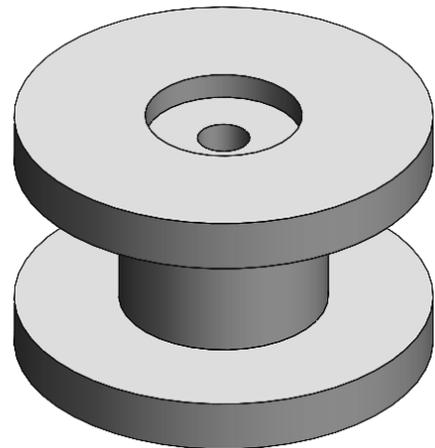
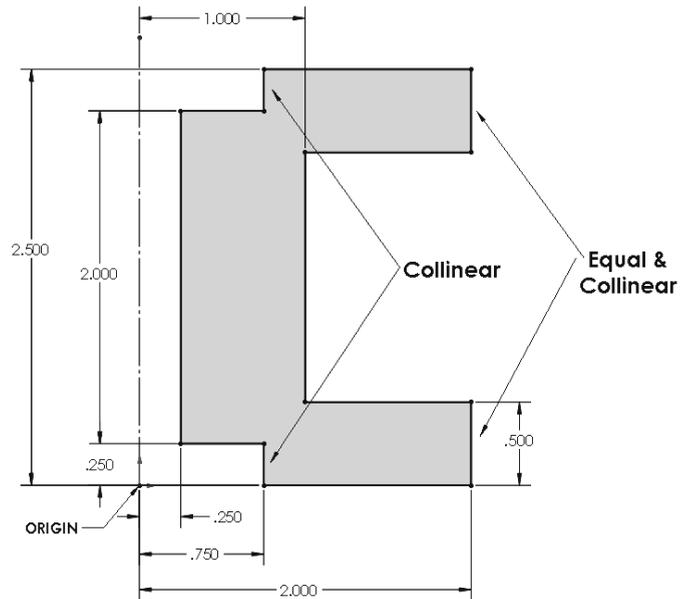


Select the two right short vertical lines and add a **Collinear** and **Equal** relation.



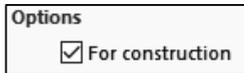
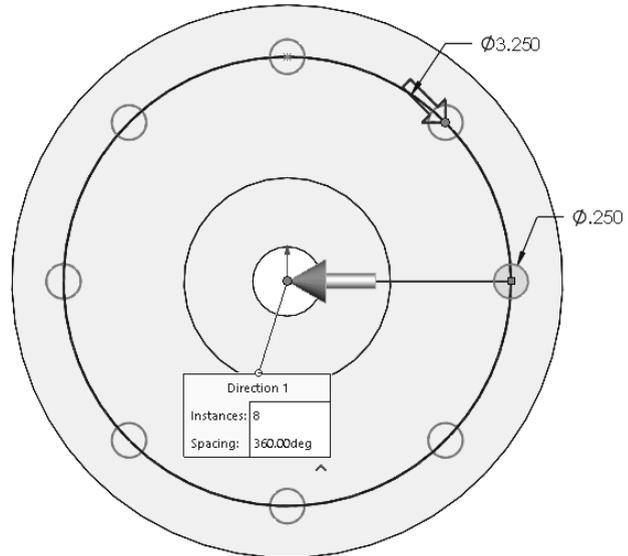
From the Features tab, select the **Revolved Boss** command. Make sure the centerline is selected as the axis of revolution, enter **360°** to make a full revolution and click OK to finish.

The resulting cylindrical part will have a **4.00"** outside diameter as shown.

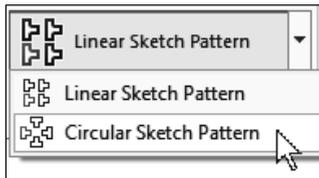


The next step is to add a circular pattern of holes at the top of the part. Change to a **Top** view orientation, select the top face of the part, and add a new sketch.

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



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 hole 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.



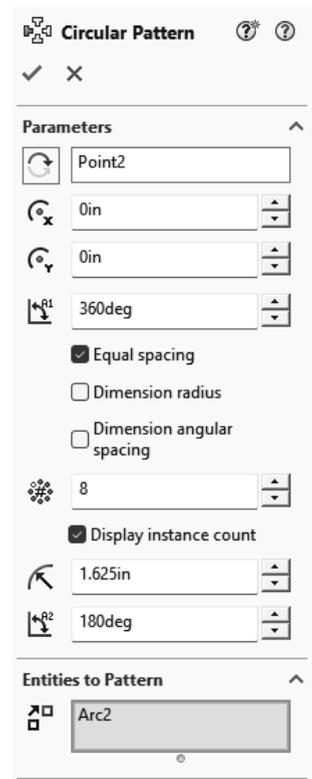
From the drop-down menu in the **Linear Sketch Pattern** select the **Circular Sketch Pattern** command.

In the **Pattern Center** selection box select the sketch origin.

From the Circular Pattern options activate the **“Equal spacing”** option 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 options and selections click **OK** to continue.

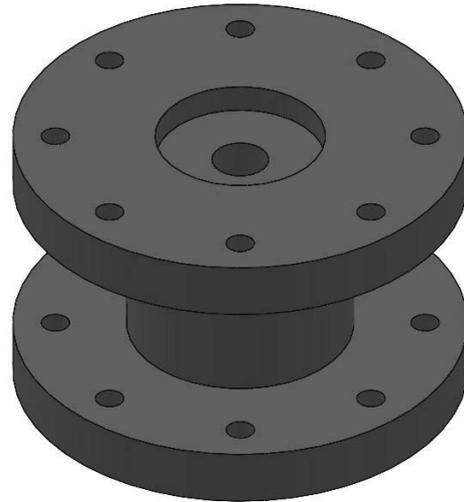
NOTE: In this case, activating the **“Dimension radius”** option would cause the sketch to become Over defined, because the location of the circle has already been defined by the larger construction circle.





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 complete model is shown here.



When finished, make sure your part is saved as **TORQUE-SENSOR.sldprt**. Follow the instructions on Page 1-9 to make a new drawing using the **TITLEBLOCK-INCHES.drwdot** and save it as **TORQUE SENSOR.slddrw**.

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.



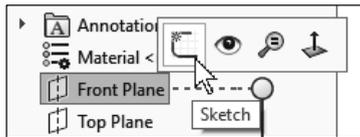
**TORQUE SENSOR**  
 Scale 1:1  
 Material: 1060 Aluminum Alloy

NAME: STUDENT	DESK:	SEC:	GRADE: 2.2
<b>DESIGN WORKBOOK USING SOLIDWORKS</b>			

## Exercise 2.3: SCALLOPED KNOB

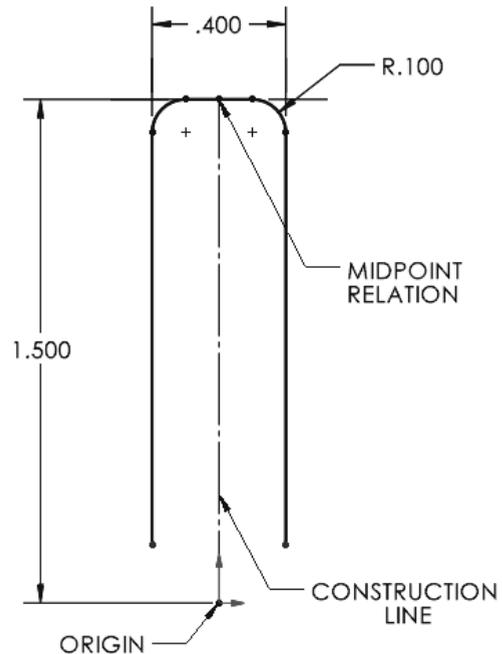
In Exercise 2.3, you will design a Scalloped Knob that has some complicated geometry around its edges. This particular knob design will be a hexagon type. Since the hexagonal features are equally spaced around the center of the knob, you will use a circular pattern to maintain the design intent.

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

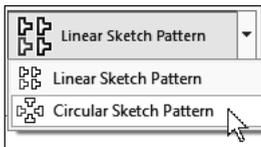


Select the **Front** plane in the FeatureManager and add a new sketch.

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. Make both vertical lines equal and add a **Centerline** from the **Origin** to the **Midpoint** of the top horizontal line. If the Centerline is not vertical, add a **Vertical** geometric relation to it.



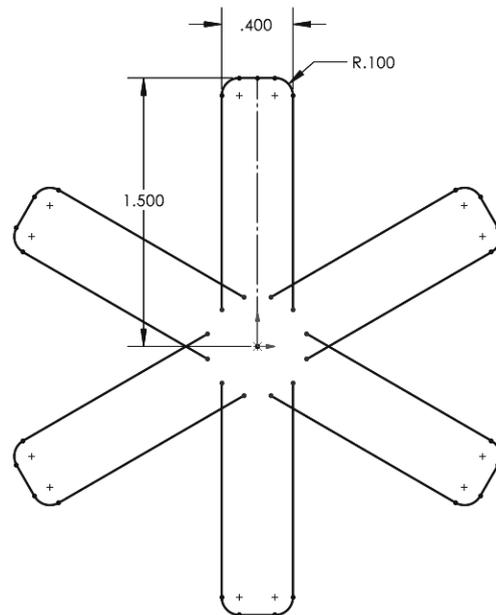
From the **Linear Sketch Pattern** drop-down menu select the **Circular Sketch Pattern** command.



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

In the **“Entities to Pattern”** selection box add the three lines and the two fillets and click **OK** to continue. Adding the Centerline to the circular pattern is not necessary.

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.

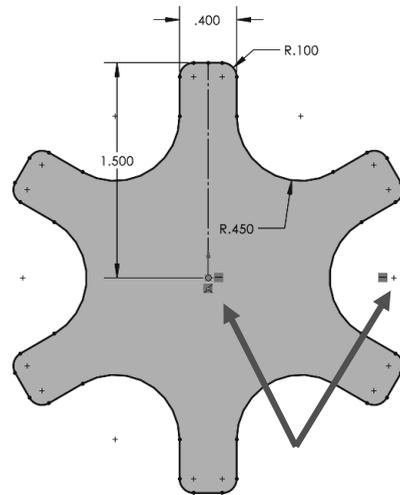




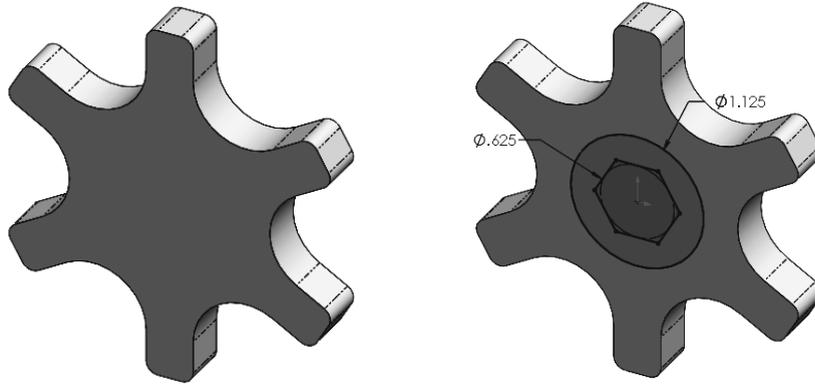
Select the **Sketch Fillet** command and set the radius to **0.45"** in the PropertyManager. Now select two intersecting lines to preview the fillet. Select the rest of the intersecting pairs of lines and click OK to finish.

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 this.

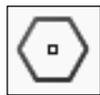
If needed, to fully define your sketch, add a **Horizontal** relation between the **Origin** and the center of the **Sketch Fillet** on either side.



From the Features tab select **Extruded Boss**. Extrude the sketch **0.375"** using the **Blind** end condition and click **OK** to finish.



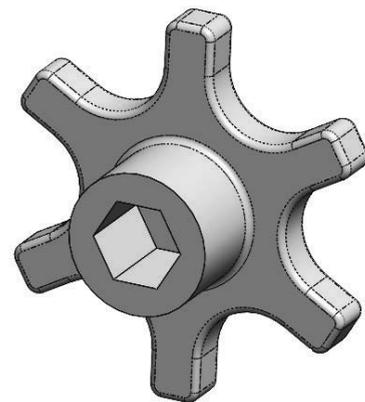
To finish the part, you will now add the attachment base. Change to a **Front** view orientation, select the front face, and add a new sketch. Draw a **Circle** centered at the origin and dimension it **1.125"** in diameter.



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



To fully define the sketch, select the top (or bottom) 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.

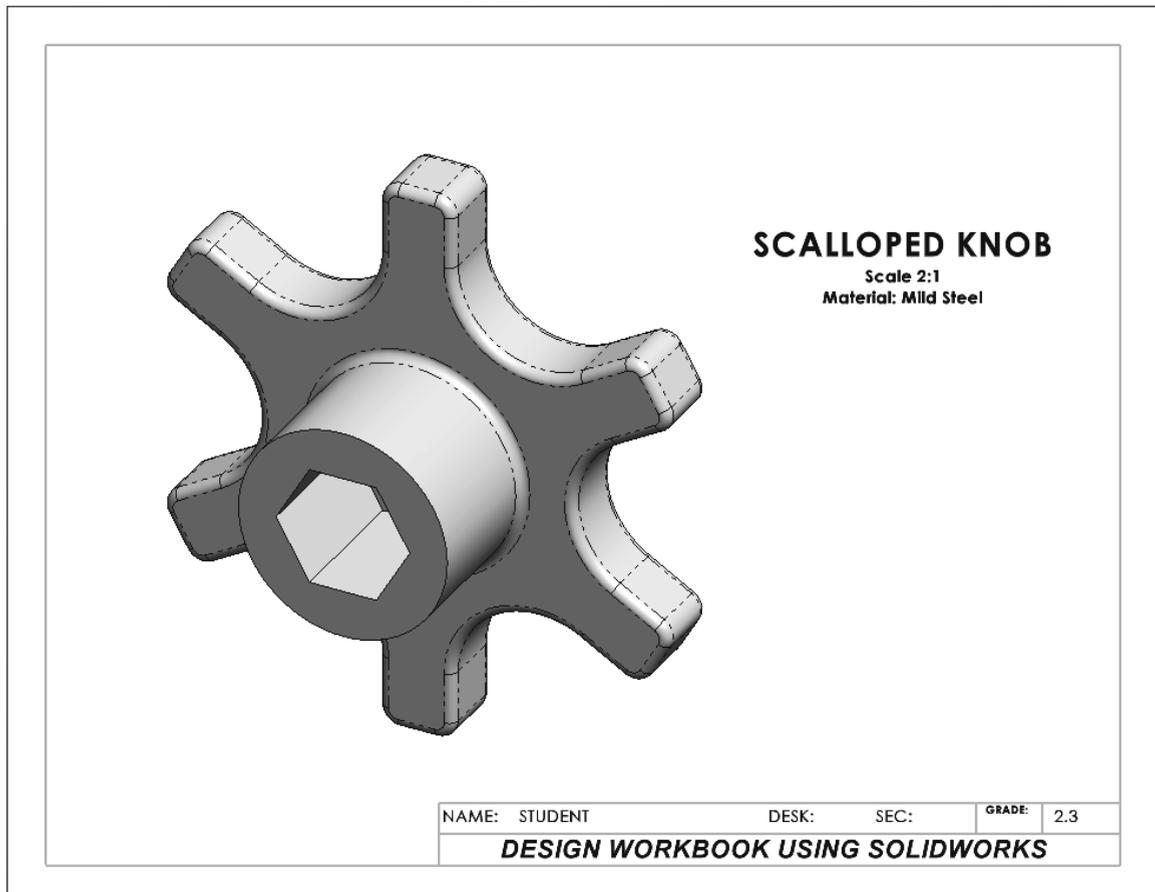
Or

- Select the front and back faces of the first extrusion and click **OK** to finish.

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 next.



## 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**.

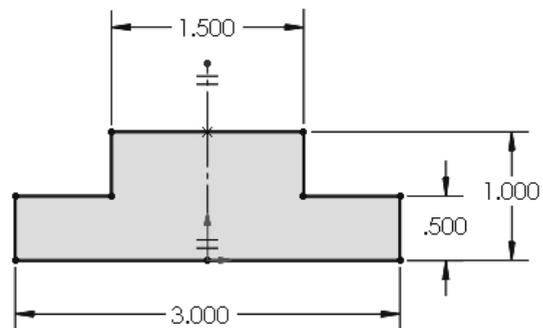
Select the **Right Plane** and add a new **Sketch**. Your part will be automatically rotated to a Right view. Draw a vertical centerline through the Origin, and with the centerline selected, go to the menu **Tools, Sketch Tools**, and select **Dynamic Mirror**. Draw the right half of the profile, and the sketch on one side of the centerline will be automatically duplicated on the other side. Use the **Smart Dimension** command to fully define the sketch.

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

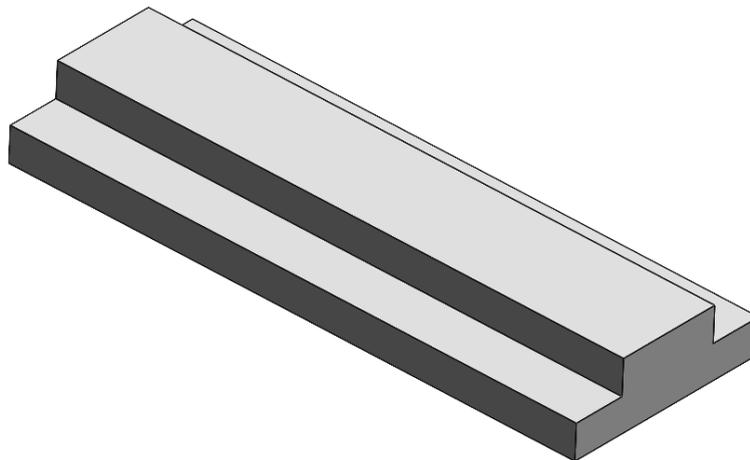
- Extrude the sketch **4.20"** in **Direction 1** using a **Blind** end condition, and **4.20"** in **Direction 2** using a **Blind** end condition.

or

- Extrude the sketch **8.4"** using the **Mid Plane** end condition.



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 as follows.



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.

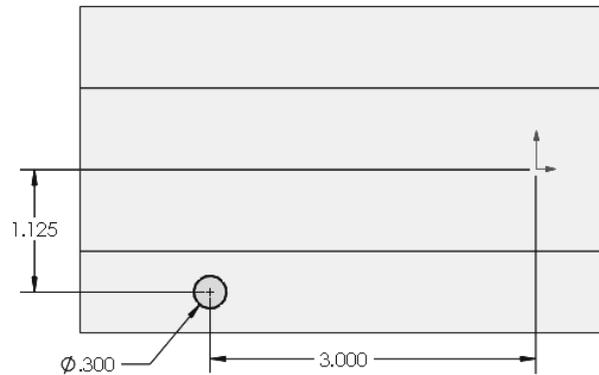


Linear Sketch Pattern

Now create a Linear Pattern. Select the **Linear**

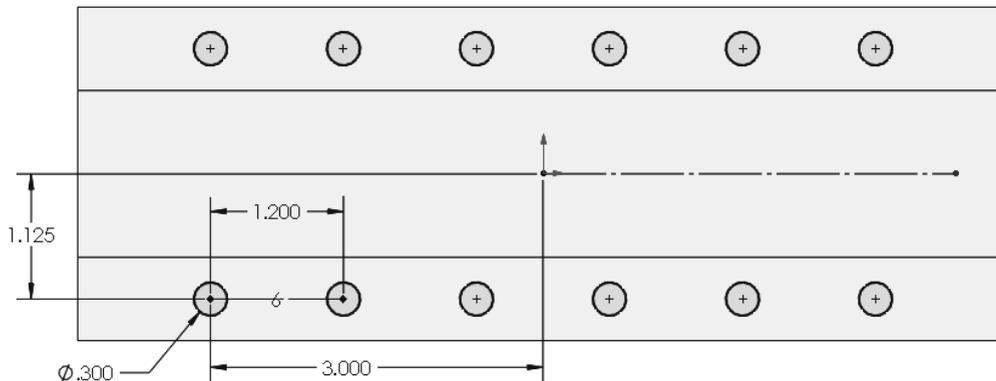
**Sketch Pattern** from the Sketch tab. In the **Direction 1** options set the number of copies to **6** with a spacing of **1.200**

and activate the “**Dimension x spacing**” and “**Fix X-axis direction**” options. The last options are used to fully define the sketch and prevent it from moving.



Mirror Entities

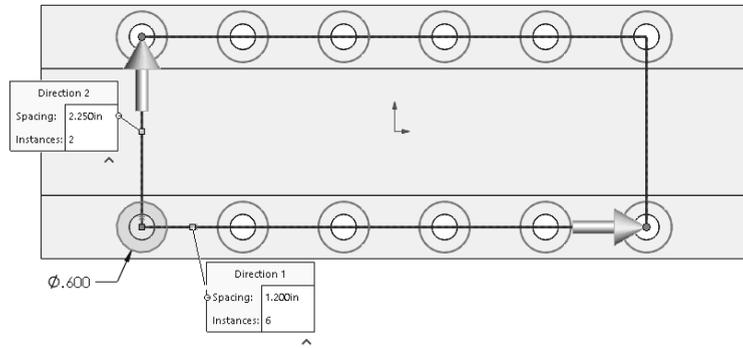
Now that the circles on one side are complete, you need to add the six circles on the other side. To copy the linear pattern, you will use the **Mirror** command. First draw a **Horizontal** centerline starting at the Origin and then select the **Mirror Entities** command. In the Mirror options add the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



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.

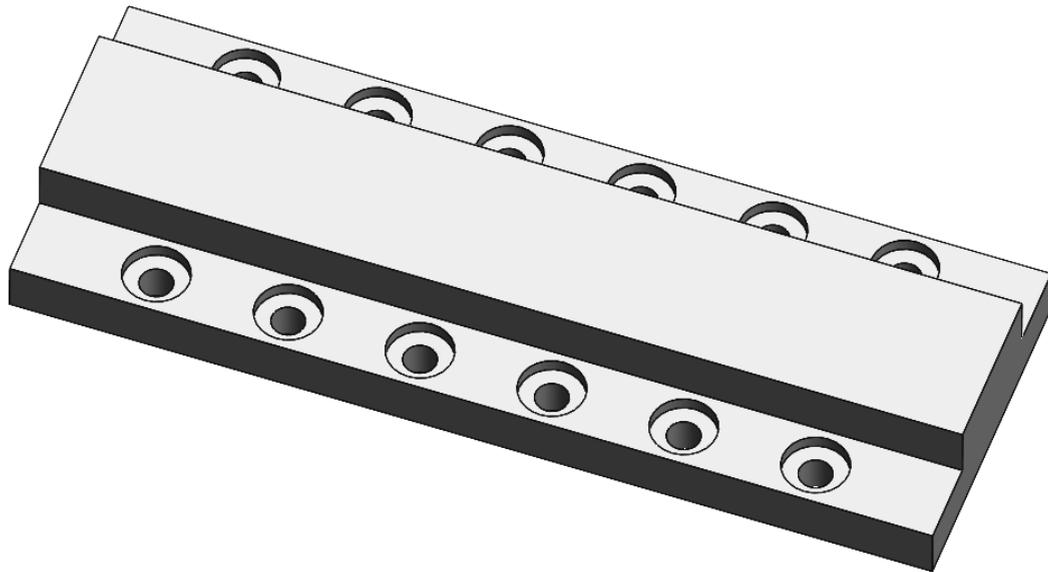
**NOTE:** SOLIDWORKS has a **Hole Wizard** command that can easily create many different types of holes for your designs, including counterbore holes, which can then be patterned and mirrored, simplifying the process significantly. The **Hole Wizard** will be covered in a later lesson; in this lesson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

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.

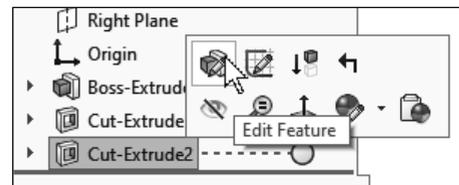


Now add a **Linear Pattern** as before, but this time you will make the pattern in two directions. In **Direction 1** make six (**6**) copies spaced **1.200**". In **Direction 2** make two (**2**) copies spaced **2.250**", turn on the "**Dimension Y spacing**" and "**Dimension angle between axes**" to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

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 like the following image.



**NOTE:** If you make a mistake or need to modify a 3D 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. If you need to modify the 2D sketch, select Edit Sketch.

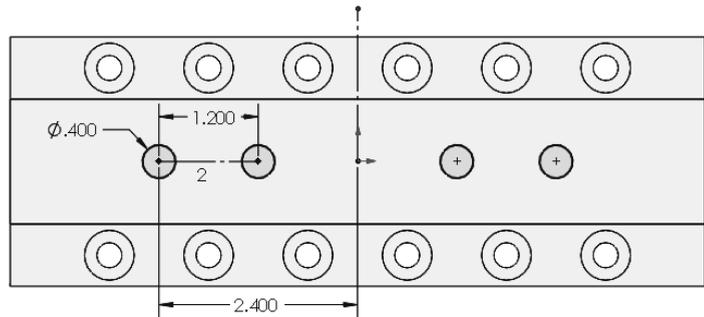


The next step for your design is to create four holes on the top of the plate. Change to a **Top** view orientation and add a new Sketch on the top face. Draw the first **Circle** on the left, and dimension it as indicated.

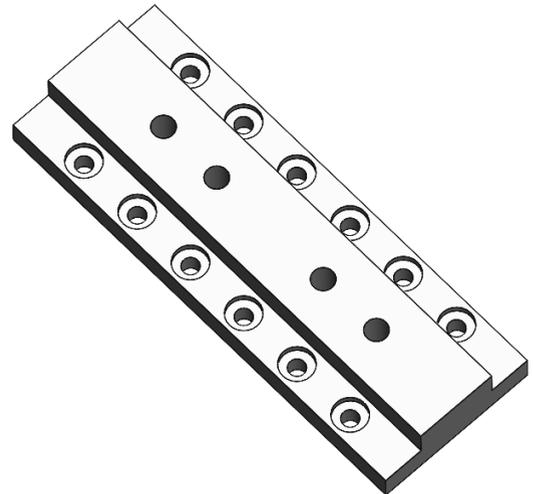


To maintain the design intent, select the center of the circle and the origin, and add a **Horizontal** relation to fully define the circle's location.

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.



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.



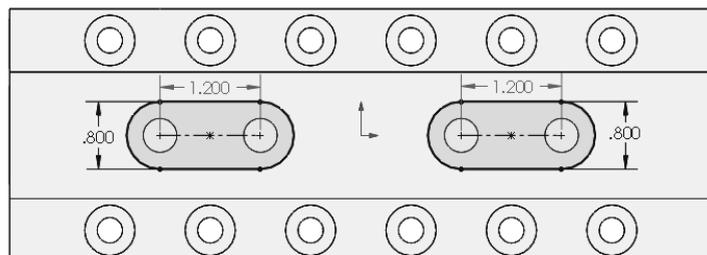
Now you need to add two counterbore slots. Change to a **Top** view and add a new sketch on the top face of the part. Select the **Slot** command from the sketch tab and use the **Straight Slot** option. Start the slot concentric to the left circle of the top face and end it concentric to the hole immediately to the right. This will



make the slot concentric with the two circles. Dimension the slot's height to **0.80"**. Using the same approach, draw a second slot on the holes to the right of the origin, or use the **Sketch Mirror** command.

**NOTE:** Using the slot's "**Add dimensions**" option will add width and height dimensions when drawing the slot.

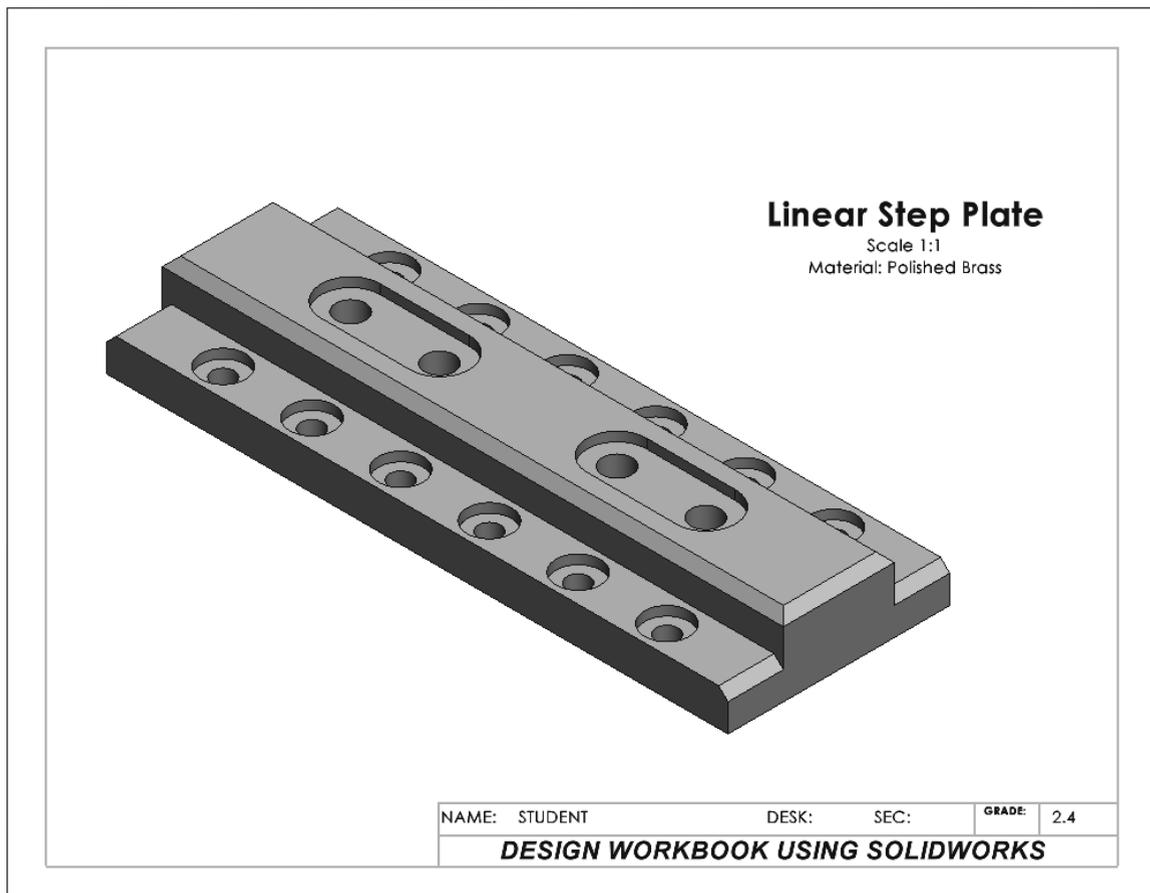
Since we defined the slot's width using the holes' centers, the width dimension is redundant and it will be colored grey, meaning it is a reference dimension. In other words, it cannot control the slot's size.



To make the counterbores select the **Cut Extrude** command using a **Blind** end condition of **0.250"**. The final step is to add a chamfer to 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" x 45 deg**, 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 part.

To finish the exercise, right click on **Material** in the FeatureManager and select **Edit Material**. Expand the **Copper Alloy** materials category, assign **Brass** to your part and save it as **LINEAR STEP PLATE.sldprt**.

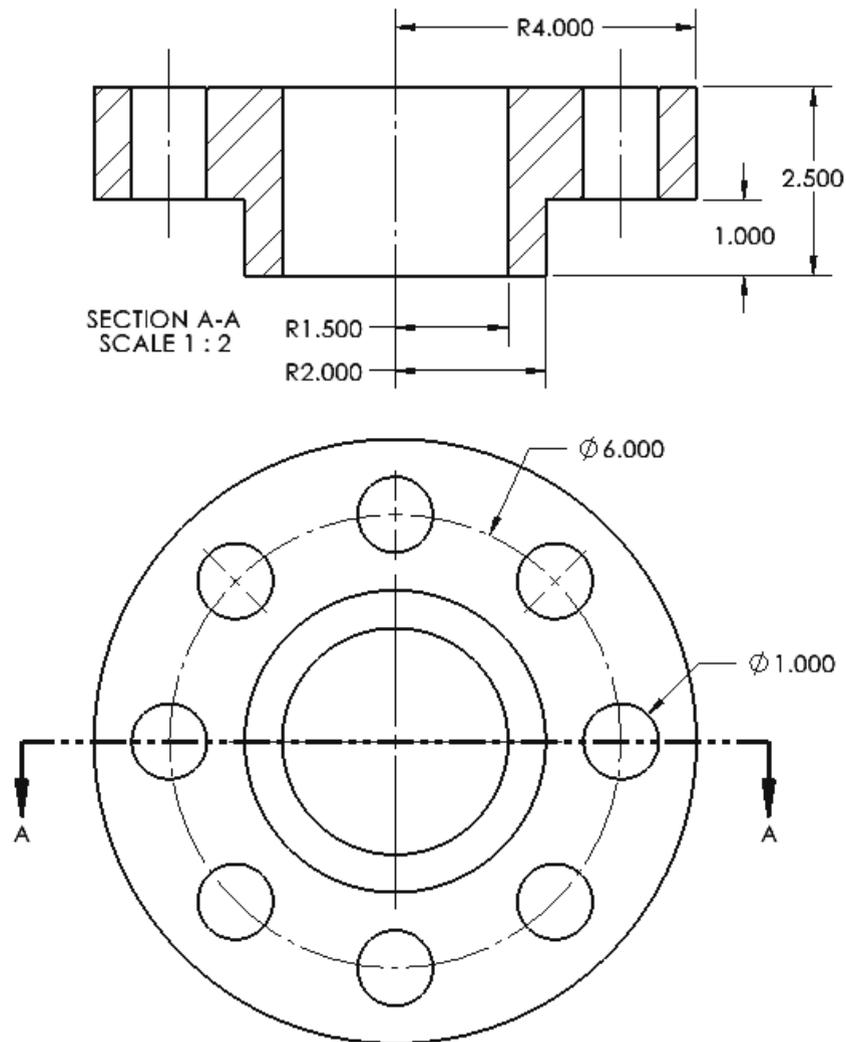
Make a new drawing using the **TITLEBLOCK-INCHES.drwdot** template, add a shaded with edges **Isometric** view with a 1:1 view scale, and add the necessary annotations. Save your drawing as **LINEAR STEP PLATE.slddrw** and print a copy for your lab instructor.



## 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.

