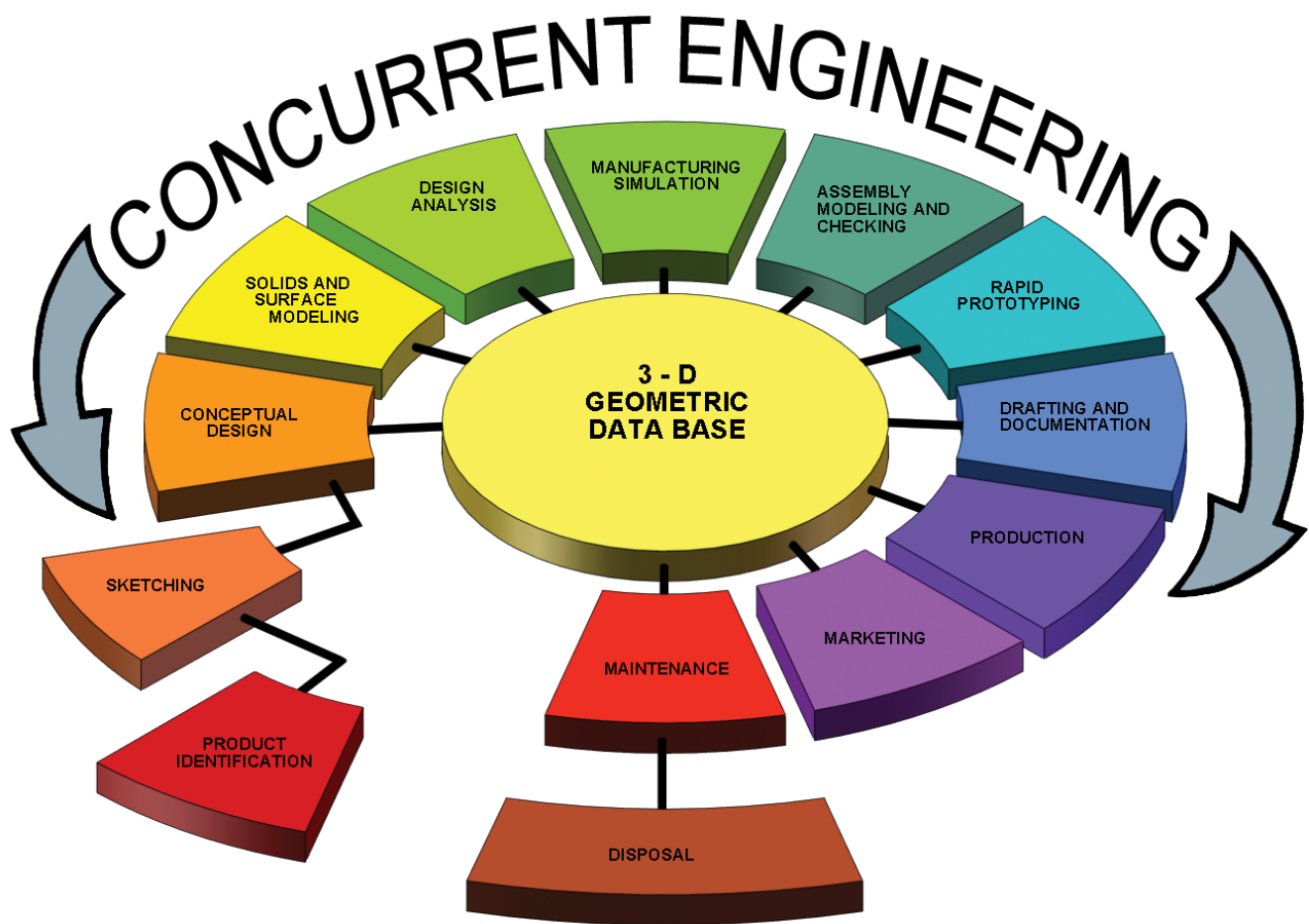


# Engineering & Computer Graphics Workbook

Using SOLIDWORKS® 2019



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# Computer Graphics Lab 2: 2-D Computer Sketching II

## ADVANCED 2-D SKETCHING

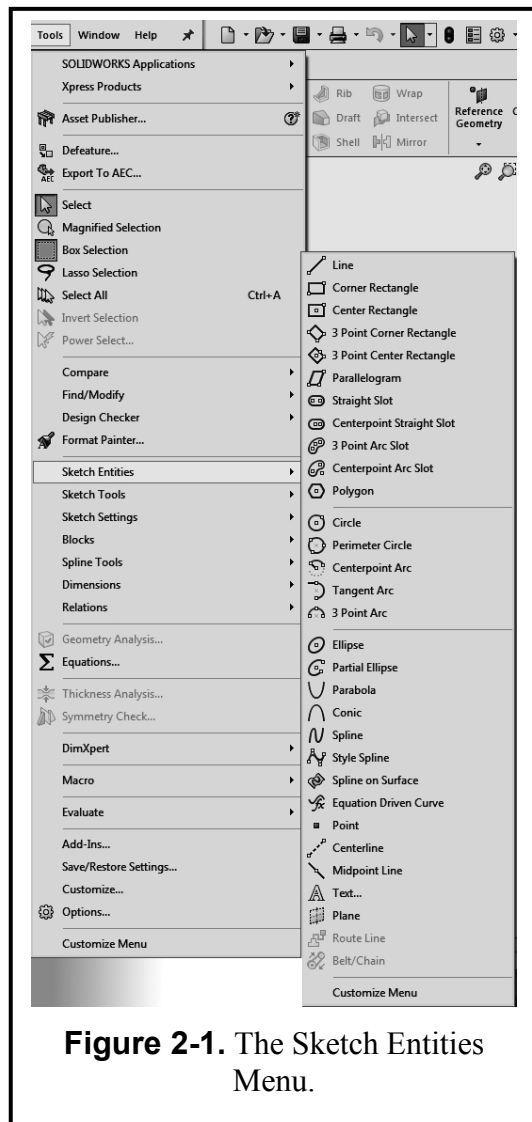
In the first Computer Graphics Lab, you used some of the basic 2-D sketching capabilities of SOLIDWORKS. These first exercises concentrated on using items that were available on the sketch toolbar. You learned how to draw a Line, Circle, Rectangle, Arc, Polygon, Centerline, and Spline. You also learned how to edit the 2-D sketch using Dimensions, Trim, Mirror, Fillet, and Chamfer functions. In this Computer Graphics Lab 2, you will learn some more advanced 2-D sketching and editing features available in SOLIDWORKS.

## SKETCH ENTITIES MENU

The sketch entities shown under the sketch tab are not the only ones available. Many of the icons have a small down arrow next to them. Each of these icons have additional options available for your use. These entities are also accessible under the **Tools – Sketch Entities** and are shown in **Figure 2-1**. Here you can find the following 2-D sketch entities:

- Line
- Rectangle (several options)
- Parallelogram
- Slot (several options)
- Polygon
- Circle
- Perimeter Circle
- Centerpoint Arc
- Tangent Arc
- 3 Point Arc
- Ellipse (several options)
- Partial Ellipse
- Parabola
- Spline
- Spline on Surface
- Point
- Centerline
- Text

Some of these 2-D entities are more commonly used in engineering design than others, but hopefully you will have a chance to use each of them somewhere in one of your exercises.



**Figure 2-1.** The Sketch Entities Menu.

## SKETCH TOOLS MENU

All of the 2-D sketch editing commands are found under the **Sketch Tab**. In this toolbar you will find the following commands:

**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 the entity.

**Extend** extends an entity to meet another entity.

**Mirror** copies a sketch element about a centerline.

**Dynamic Mirror** dynamically mirrors new sketch elements about a centerline. First select the entity about which to mirror and then draw the sketch entities to mirror.

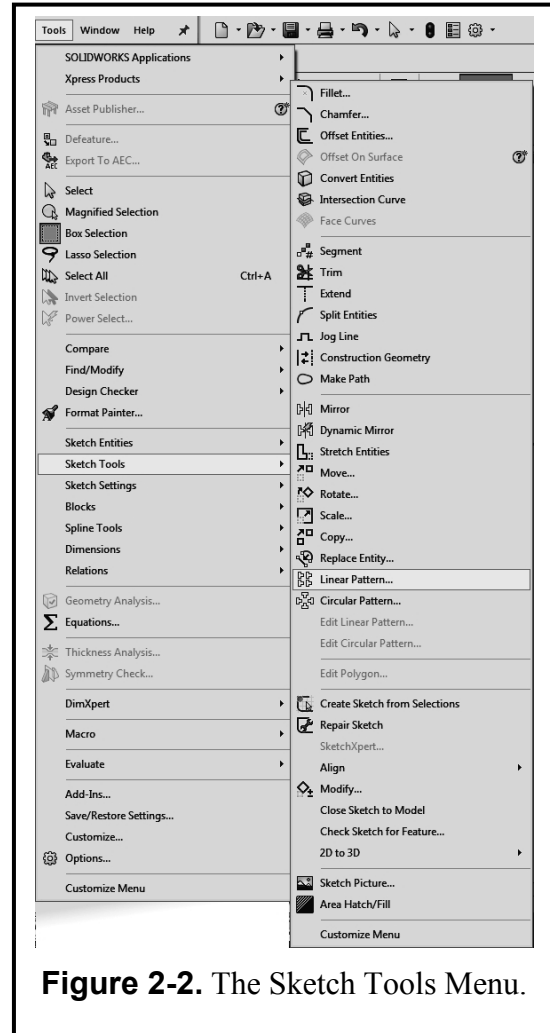
**Jog Line** moves a piece of the line up or down in a rectangular shape.

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

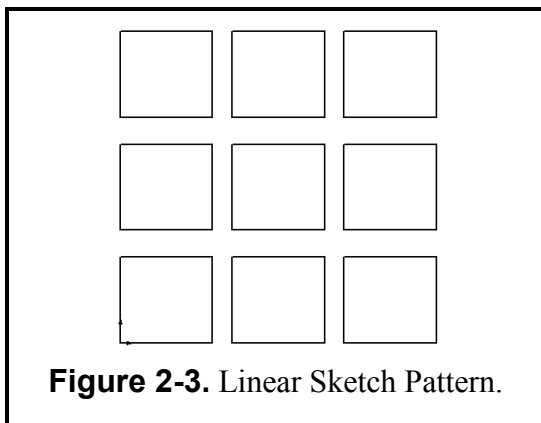
**Linear Sketch Pattern** creates a rectangular pattern (row X column) of identical entities (see **Figure 2-3**).

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

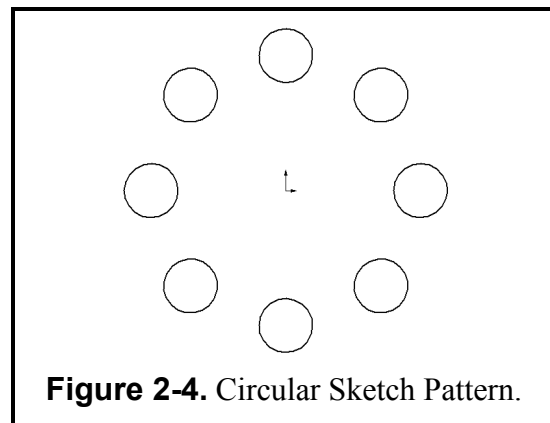
**Align** is used to align a sketch and a grid point.



**Figure 2-2.** The Sketch Tools Menu.



**Figure 2-3.** Linear Sketch Pattern.



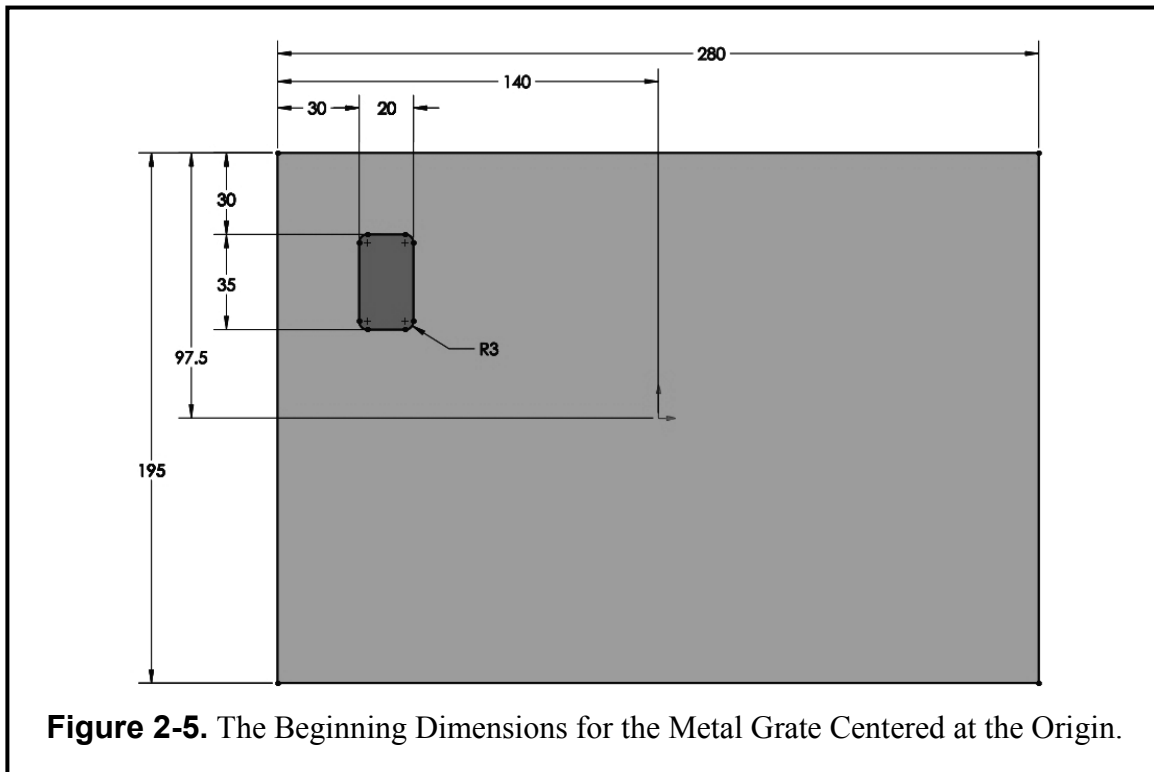
**Figure 2-4.** Circular Sketch Pattern.

## Exercise 2.1: METAL GRATE

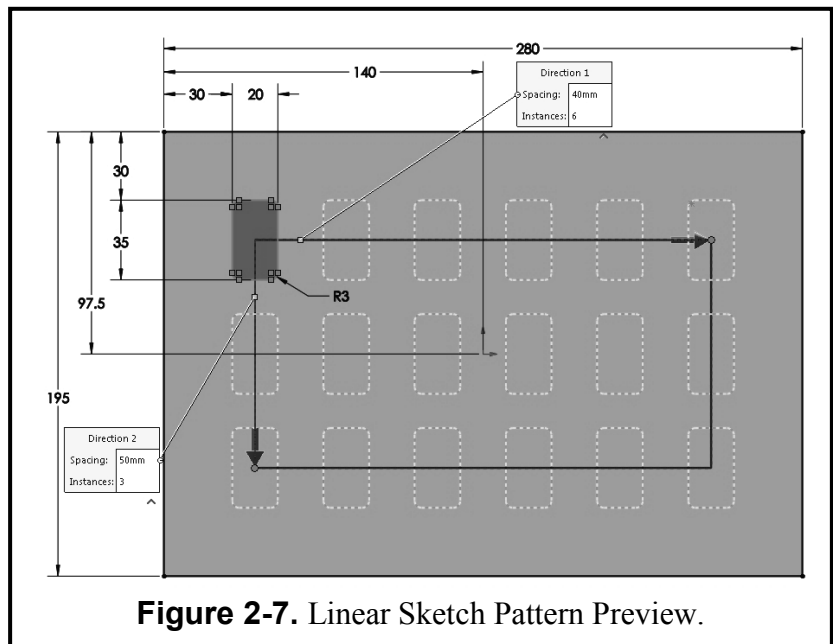
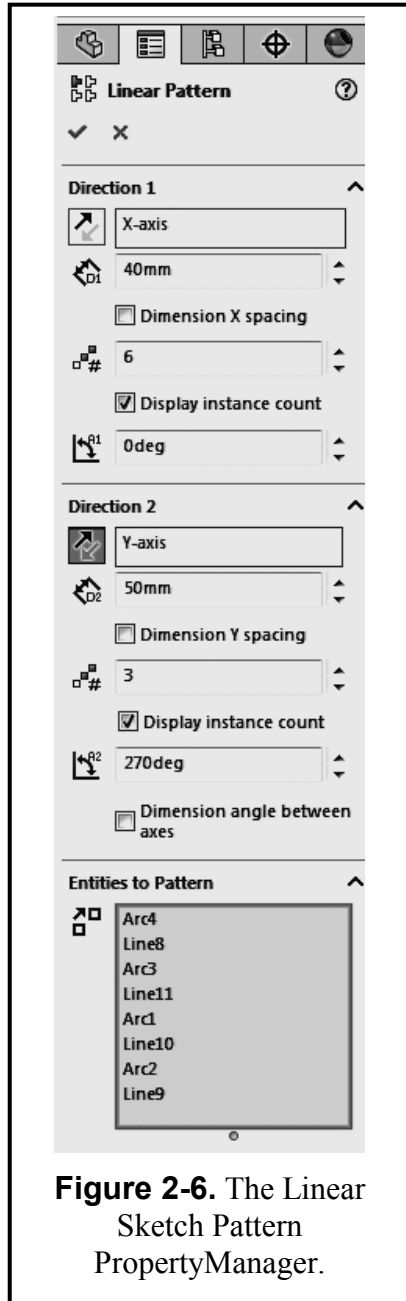
In Exercise 2.1, you will design a Metal Grate. The function of a metal grate is such that many identical slots are cut through it. Instead of drawing each slot separately, you will use an advanced sketching command SOLIDWORKS to create a rectangular pattern of slots. Then you can simply extrude the sketch to create the base feature of the grate.

Start by going to your folder and **Open** the file **ANSI-METRIC.prt** because the dimensions of the Metal Grate are in Metric units. Immediately **SAVE AS – METAL GRATE.sldprt**. You will not need a grid for this exercise. Go to **Tools – Options - Document Properties** and click the **Grid/Snap** tab and make sure the “Display Grid” function is **not** checked (✓) on, then click the **OK** button. Select the **Front** plane in the FeatureManager for the sketch plane.

From the **Sketch Tab** select the **Sketch** command to start. You will first draw two **Rectangles**. The first one is the large outline of the grate and the second one is the initial small rectangular slot that eventually will be patterned. Refer to **Figure 2-5** and use the **Dimension** command to add the sketch dimensions needed. The overall size of the grate is **280 mm** by **195 mm**, and the (**140** and **97.5**) dimensions are used to center it about the origin. The small slot is **20 mm** by **35 mm** and is **30 mm** below the top and **30 mm** to the right of the upper left corner. *Note:* After all the dimensions are applied, the lines turn black. This means that the geometry is fully defined and constrained. Using the fillet command, add **3mm fillets** to the four corners of the small rectangle.



Now select the **Linear Sketch Pattern** command in the Sketch toolbar or the menu **Tools**; select **Sketch Tools** and then click on **Linear Pattern**. The “Linear Pattern Repeat” PropertyManager is automatically displayed. In the Entities to Pattern selection box select the lines and fillets of the small rectangle. The settings for this rectangular Pattern operation are shown in **Figure 2-6** below. “Direction 1” is horizontal and will have **6** instances. The horizontal spacing is **40** mm and the angle is **0** degrees. To activate “Direction 2” change the number of instances to **3**. You will then be able to change the vertical spacing to **50** mm and the angle to **270** degrees. Notice that as you change the options a **Preview** of the pattern is shown. You should now have a 6 x 3 pattern of slots. You will have to select the Reverse Direction option in the Direction 2 section to the left of the Y-axis button to reverse the pattern’s direction. Your pattern preview should look like the image in **Figure 2-7**. When finished, click the **OK** button to complete the pattern.



From the **Features Tab** select **Extruded Boss/Base**, enter the following parameters and click OK to finish:

Type of Extrusion = **Blind**

Distance 1 = **5** mm

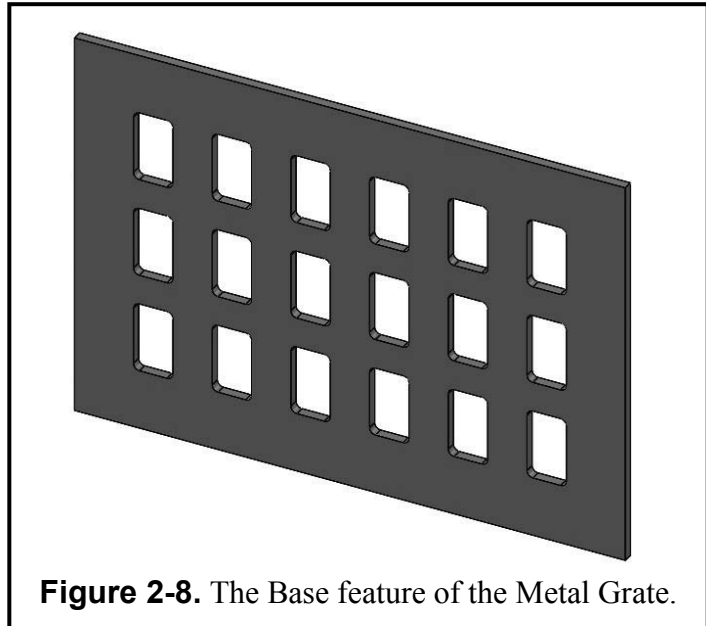
You will now have the base solid model of the grate, as shown in **Figure 2-8** in a **Trimetric** view.

The next step for the Metal Grate is to add a lip to the metal grate in order to provide a support when attached to the wall air duct. Click on the front surface of the Metal Grate. It should highlight *blue*. Then select the **Sketch** tab and click on the **Sketch Command** to add a new sketch. You have already drawn the outer rectangular profile, so you will borrow from it for the outer edge of the lip. From the Sketch tab select the **Convert Entities**



command. The outer lines are projected as sketch entities into your active sketch. Notice that they are all black lines since the geometry is already defined.

Now select the top converted line (it turns *cyan*) and then select the **Offset Entities** command. Enter the offset dimension of **15 mm** and make sure the **Select Chain** option box is selected. If the 15 mm offset is previewed on the outside, select the **Reverse** option box so the offset is *inside* of the original lines and then click **OK** to create the offset, as shown in **Figure 2-10**. Also, notice that the offset command places a small 15 mm dimension on your sketch to indicate the offset value. You could now double-click on the dimension, enter in a new dimension value, and change the offset to a new value. But for now leave it at 15 mm. Add a **5mm Fillet** to the inside corners of the offset pattern as shown in **Figure 2-10**.



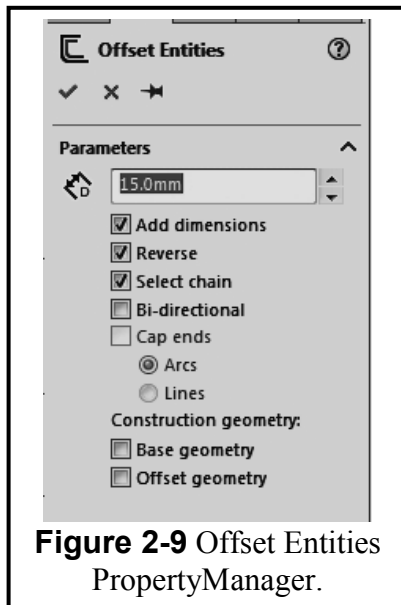
**Figure 2-8.** The Base feature of the Metal Grate.

Before you Extrude the sketch, you may want to change to an Isometric view in order to see which direction you are extruding. From the **Features** tab select **Extruded Boss**. In the **Extrude** PropertyManager enter the following parameters:

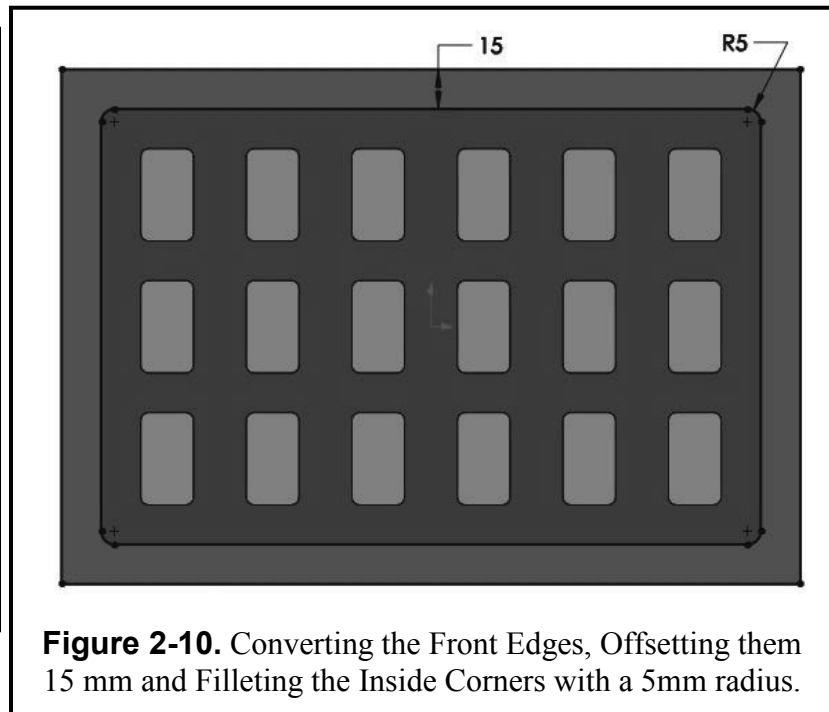
End Condition = **Blind**

Distance 1 = **5 mm**

Click OK to complete the boss.



**Figure 2-9** Offset Entities PropertyManager.



**Figure 2-10.** Converting the Front Edges, Offsetting them 15 mm and Filletting the Inside Corners with a 5mm radius.

Now you need to add four attachment holes to the corners of the grate. **Select** the face of the last extrusion (it will turn *blue*). From the **Sketch Tab** select the **Sketch** command and draw a **Circle** in the upper left corner. Use the **Dimension** values supplied in **Figure 2-11** for the circle diameter (**8 mm**) and position from the corner (**9mm x 9mm**).

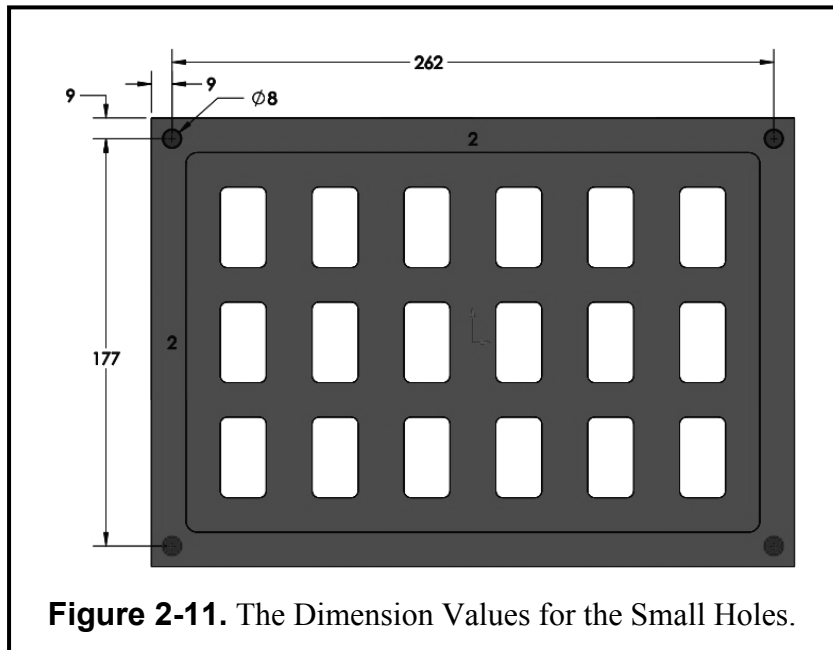
Now draw three more **Circles** in the other three corners. **Dimension** them to have the same diameter (**8**) and same relative position (**9 x 9**) from each corner. *Or*, now that you are an expert with a rectangular array, use the **Linear Sketch Pattern** command instead. If you use this command, then the horizontal distance of the **2** items is **262 mm** and the angle is **0** degrees. The vertical distance of the **2** items is **177mm** and the angle is **270** degrees. *Either way*, when you are finished you should have circles at the four corners and click OK to finish the Linear Pattern.

Change the model orientation to a Trimetric view. From the **Features** tab select **Extruded Cut**. Set the end condition type to **Through all** and click OK to finish. The four corner attachment holes are now created on the grate.

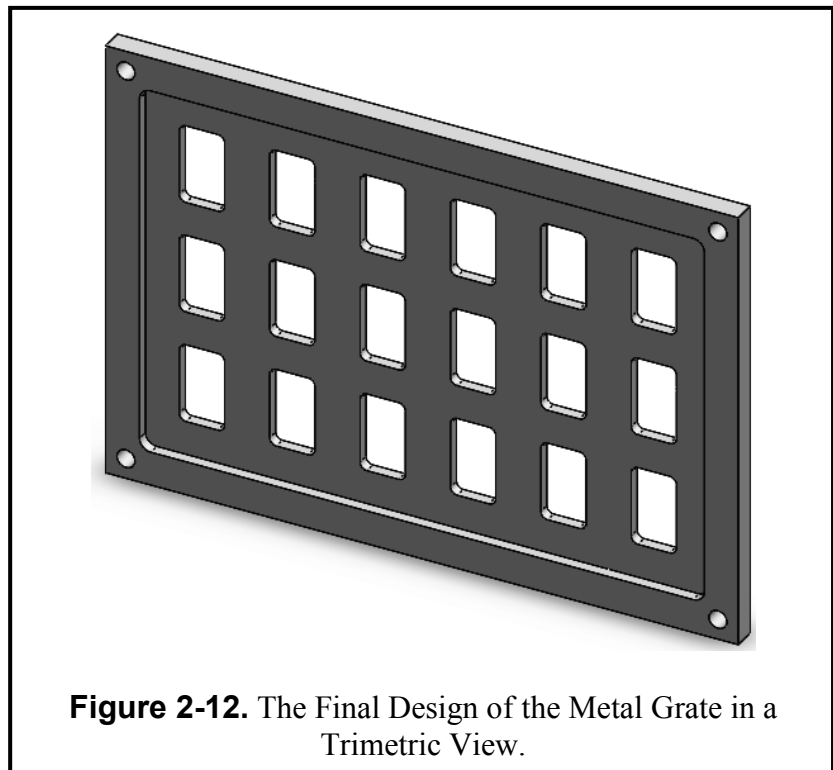
The part is now complete, and you can view the lip feature more clearly by using the **Rotate View** command as shown in **Figure 2-12**.

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 in the context toolbar. You can then assign any color you wish to the model.

Return to a **Trimetric** View of your part as shown in **Figure 2-12**. You should now save your model. Pull down **File**, select **Save As**,



**Figure 2-11.** The Dimension Values for the Small Holes.

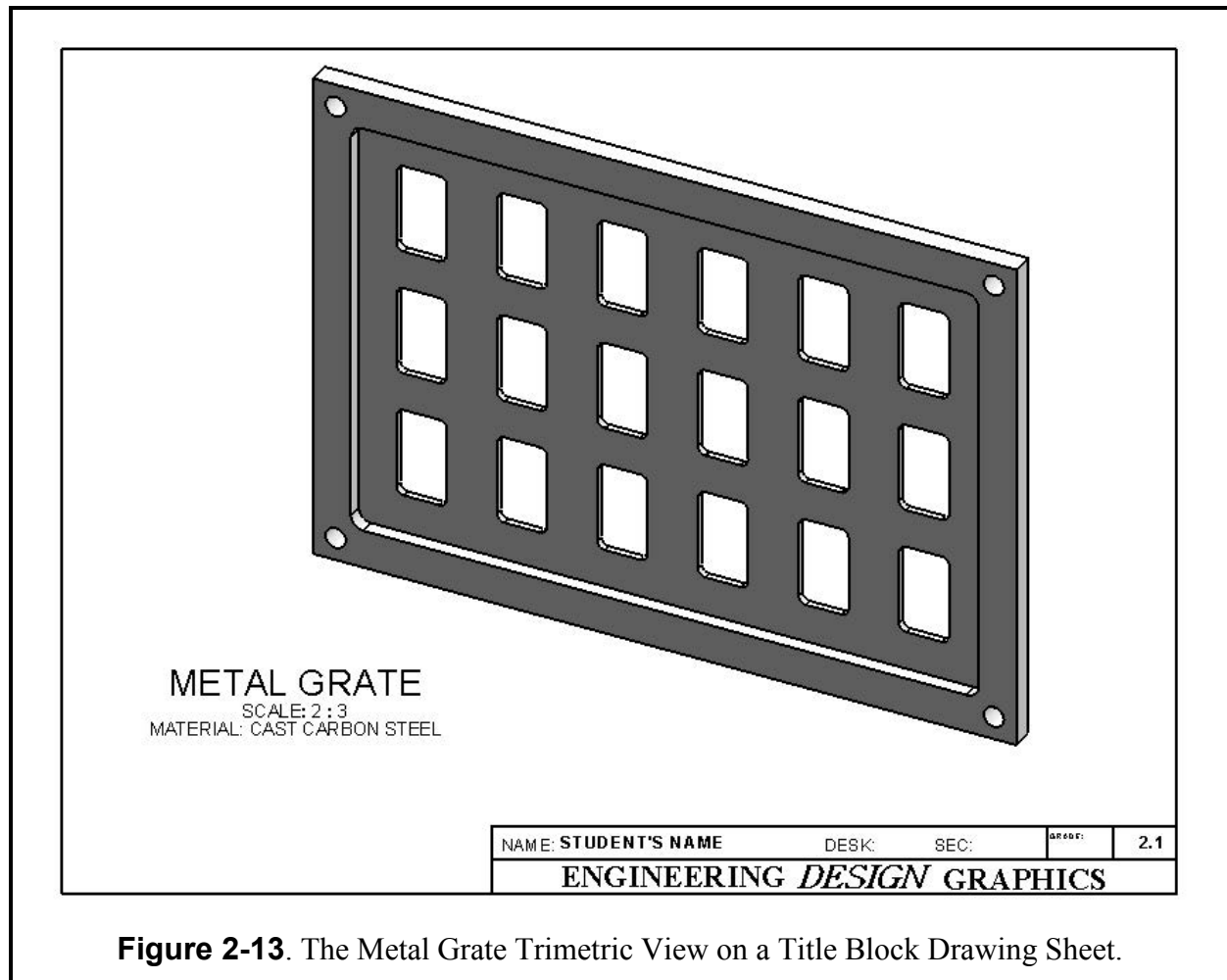


**Figure 2-12.** The Final Design of the Metal Grate in a Trimetric View.



type in the part name **METAL GRATE.sldprt**, and then click **Save**. Open your copy of **TITLE BLOCK – METRIC.drwdot** and immediately **SAVE AS – METAL GRATE.slddrw**. Now insert the Trimetric Metal Grate view into your **Title Block** template and **Print** it (see **Figure 2-13**).

**Print** a hard copy to submit to your lab instructor.



## Exercise 2.2: TORQUE SENSOR

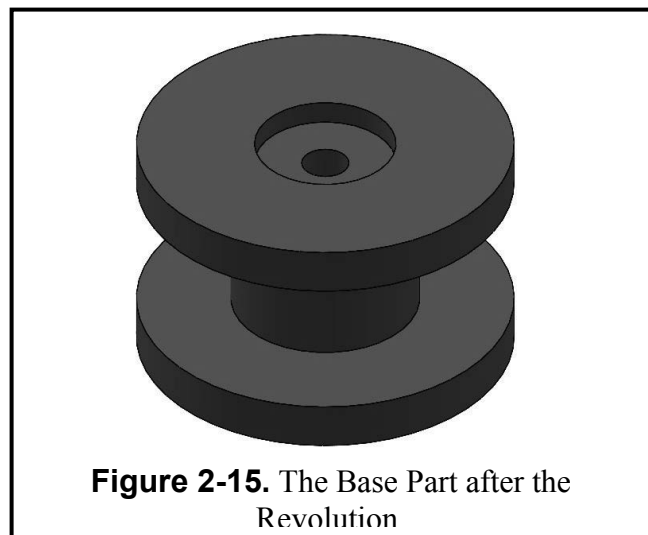
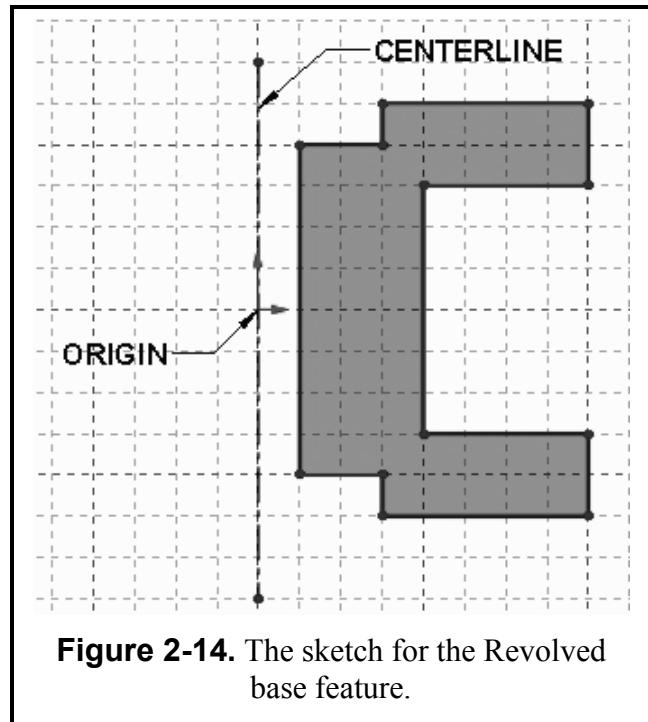
In Exercise 2.2 you will design a Torque Sensor casing. Since it is a cylindrical (axis-symmetrical) object, you will use some of the advanced editing features like circular pattern. Go to your folder and **Open** the file **ANSI-INCHES.prtdot**, and immediately **SAVE AS – TORQUE SENSOR.sldprt**. Select the **Tools, Options, Document Properties** menus and Select “**Grid/Snap.**” Make the following settings on this menu: “**Major grid spacing**” = **1.00**, “**Minor lines per major**” = **4**. Also go to System Snaps and make sure “**Display Grid**” and the “**Snap**” functions are selected, then click the **OK** button. Make sure the **Units** are in **Inches**. Then click **OK**.

The circular features of the Torque Sensor are on the top and bottom surfaces. But the main body is also round and can be created by a 360 degrees solid of revolution using a sketch drawn in the Front plane. Select the **Front** plane in the FeatureManager tree. From the **Sketch Tab** select the **Sketch** command and the sketching grid appears with minor grids spaced every 0.25 inches. Also make sure you are viewing the model from the **Front** view orientation.

First draw a **Centerline** vertically through the origin. Next, use the **Line** tool to sketch the closed sketch shown in **Figure 2-14**. This sketch will generate a part that is 2.50 inches tall and 4.00 inches in diameter at the top and bottom of the part.

From the **Features** tab **Select** the **Revolved Boss/Base** command. Make sure that the centerline is selected for the Axis of Revolution. Enter **360°** to make a full revolution and click OK to complete the revolved boss. The cylindrical base is shown in **Figure 2-15** in an **Isometric** view.

The next step is to create a circular pattern with eight holes around the edge in a bolt circle on the top surface of the part.



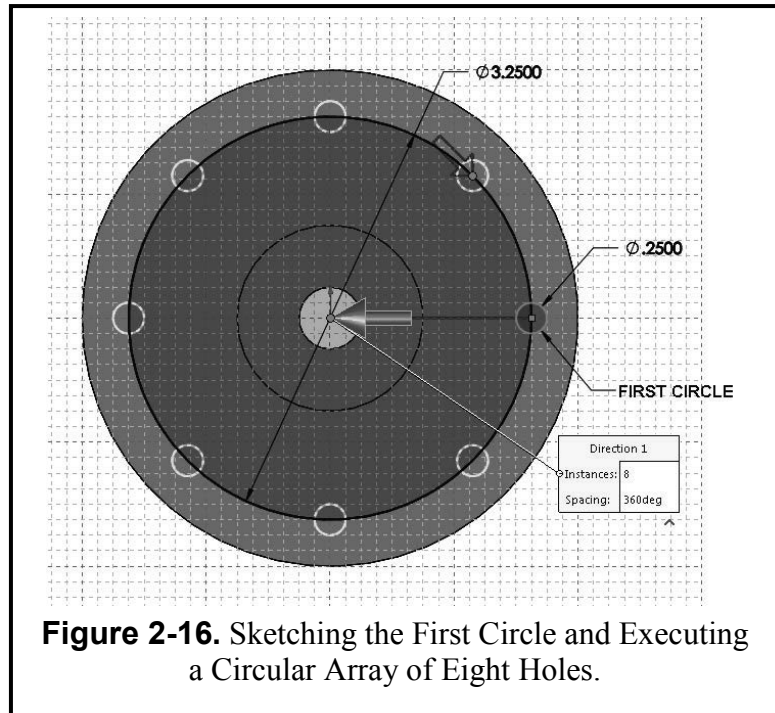
Select the **top face** of the part and it should highlight **blue**.

Also change to a **Top** view orientation. Select the **Sketch** command. Draw a **Circle** that is **3.25** inches in diameter, and activate the “**for construction**” option in the PropertyManager. Then draw a horizontal construction line from the origin to the right side of the construction circle. The intersection of these two entities defines the center of the first of eight holes, thus resulting in a radius of 1.625. Or you can go to the **Document Properties** menu and on the **Grid/Snap** tab change the “Minor lines per major” value to **8**, thus resulting in a one-eighth inch grid.

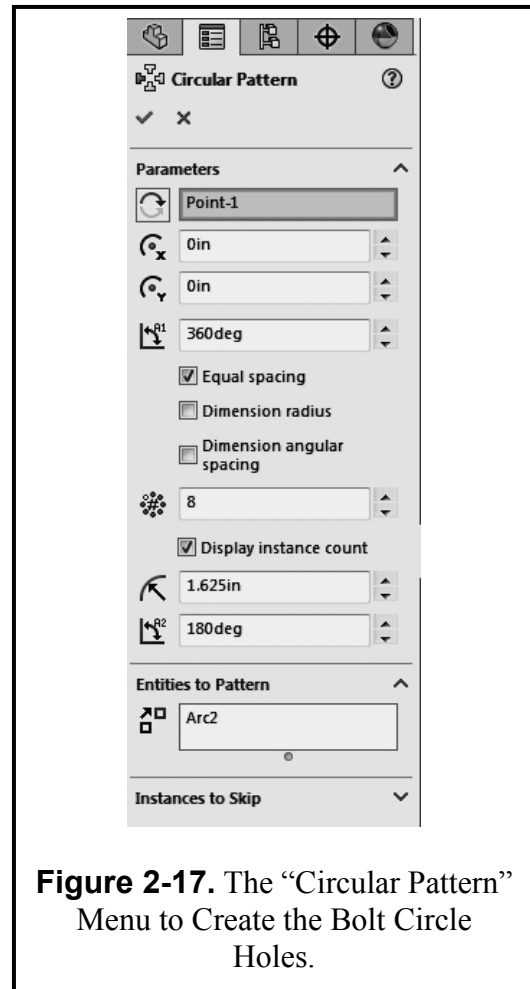
Also on the **Units** tab change the decimal places to **3**. Click **OK** and the grid should now be updated to the new values. Now locate the center of the first **Circle** on the grid and draw it with a diameter of **0.25**. Use **Figure 2-16** to aid you.

Select the circle (it should highlight **blue**). From the drop-down menu in the **Linear Sketch Pattern** select the **Circular Sketch Pattern** command. The “Circular Pattern” PropertyManager is displayed on the left. Referring to **Figure 2-17**, set the parameters for the circular pattern. The “Radius” is **1.625** from the center **(0,0)**. The “Step Number” is **8** for a “Total angle” of **360°**. Make sure the “Equal spacing” option is selected, and click **OK** to finish. You now have a bolt circle of 8 holes as shown in **Figure 2-16**. You are now ready to cut these holes through the entire model.

Switch to the **Shaded** model mode and change to an **Isometric** view orientation to better visualize the next operation. From the **Features** tab select the **Extruded Cut** command. In the “Cut Extrude” PropertyManager select the **Through all** end condition and click **OK** to make the cut extrusion all the way through the model. Use the **Rotate View**



**Figure 2-16.** Sketching the First Circle and Executing a Circular Array of Eight Holes.



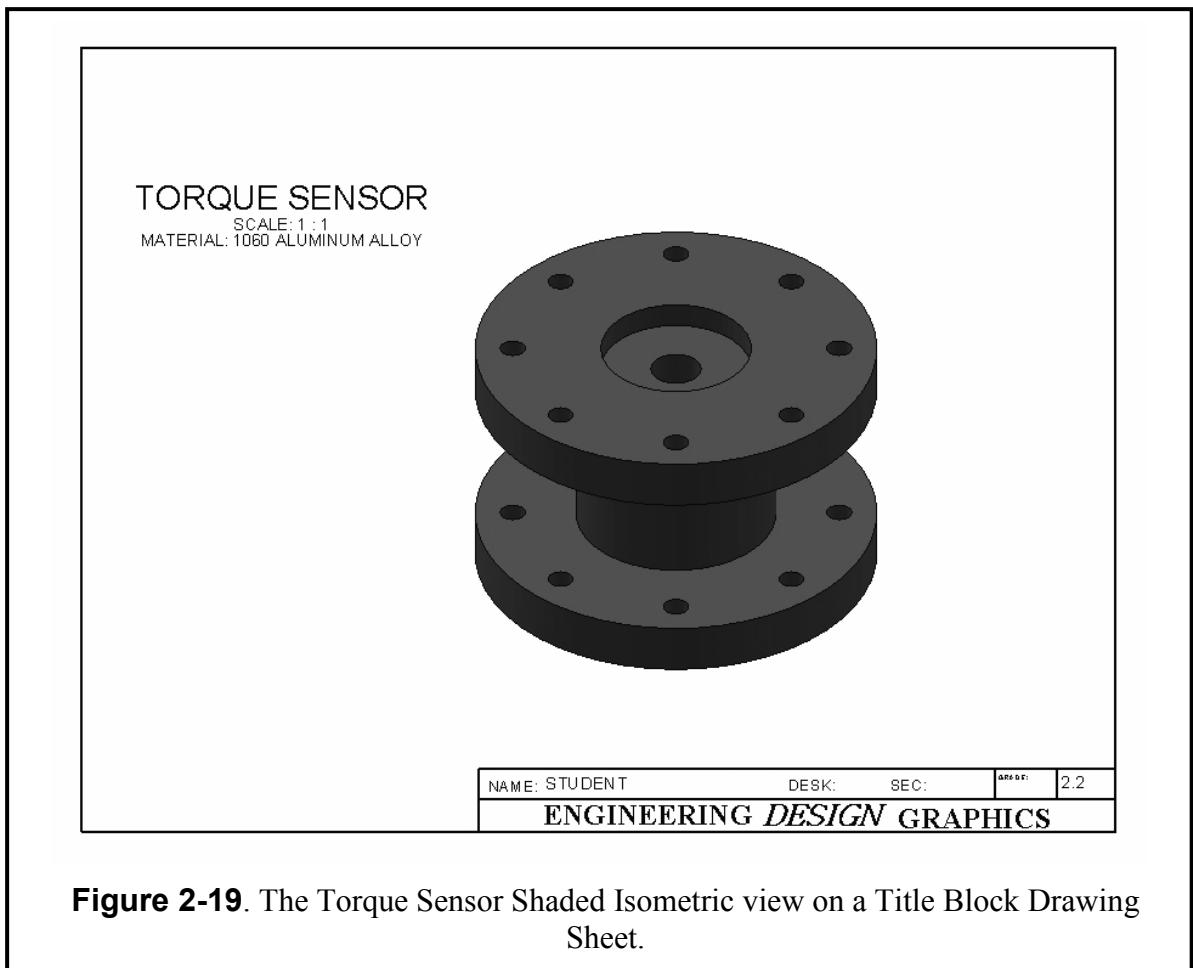
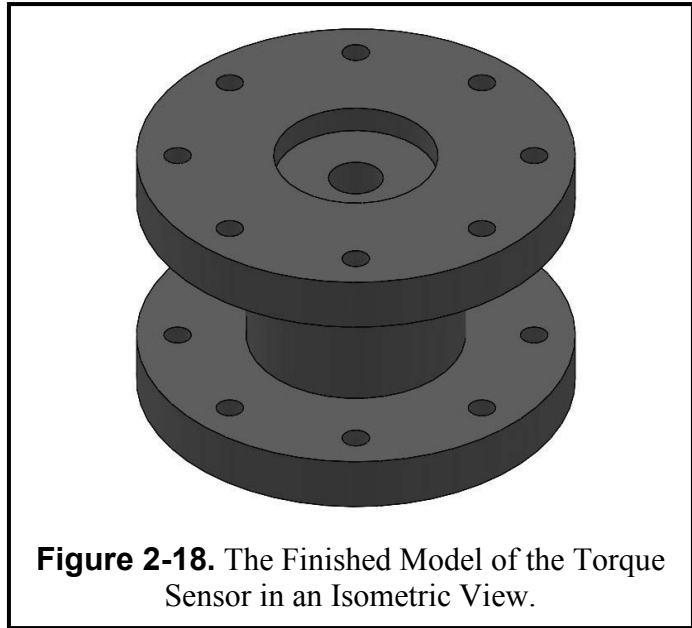
**Figure 2-17.** The “Circular Pattern” Menu to Create the Bolt Circle Holes.

command to verify the holes were cut all the way through the bottom of the model. The model is complete as shown in **Figure 2-18**.

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 any color you wish to the model.

You should now save your model. Pull down **File**, select **Save As**, type in the part name **TORQUE-SENSOR.sldprt**, and then click **Save**. Open your **TITLE BLOCK – INCHES.drwdot** and **SAVE AS: TORQUE SENSOR.slddrw**. Now

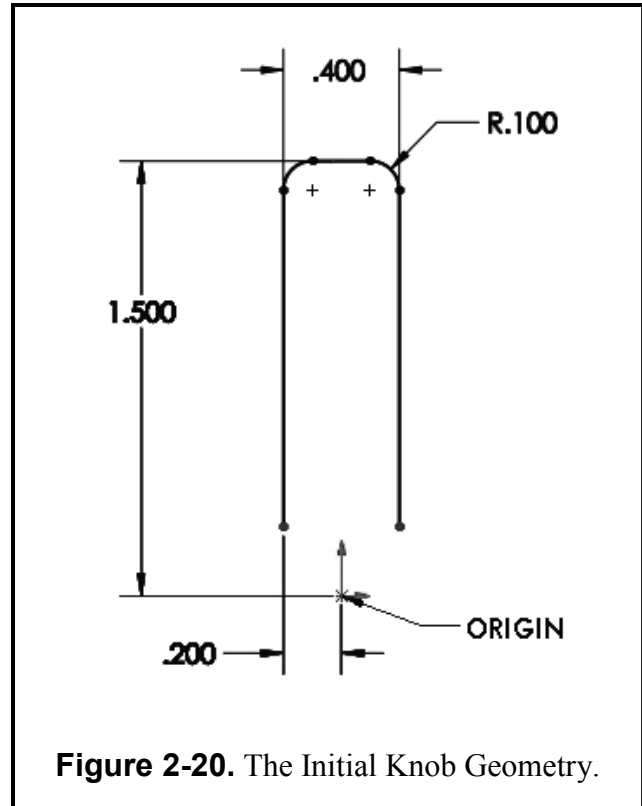
insert the shaded Isometric view of the Torque Sensor into your **Title Block** drawing sheet that was created in Chapter 1 and **Print** it for your instructor (see **Figure 2-19**).



## Exercise 2.3: SCALLOPED KNOB

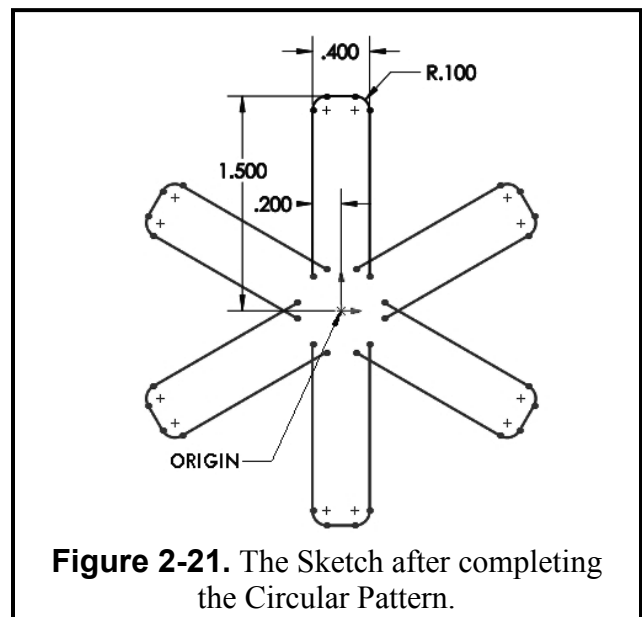
In Exercise 2.3, you will design a Scalloped Knob that has some complicated geometry around its edges. This particular knob design will be a hexagonal type. Since the hexagonal features are equally spaced around the center of the knob, you can use a circular pattern.

Start by going to your folder and **Open** the file **ANSI-INCHES.prtdot** and immediately **SAVE AS – SCALLOPED KNOB.sldprt**. Go to **TOOLS – OPTIONS – DOCUMENT PROPERTIES** and change the **UNITS** to three decimals. Select the **Front** plane for the sketch. Then start a new **Sketch**. Complete the initial geometry of the sketch according to **Figure 2-20**. Using the **Line** tool, draw two vertical lines and cap them off with a horizontal line that touches their top ends. **Fillet** the top two corners with a **0.10** radius. Use the **Dimension** tool to define the geometry by adding the dimensions shown in **Figure 2-20**. Add the dimensions referenced to the origin. When the geometry is defined, all lines turn *black*.



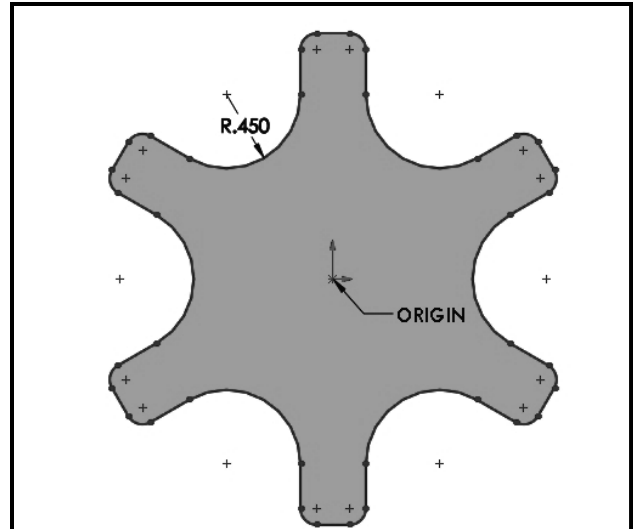
**Figure 2-20.** The Initial Knob Geometry.

Now make a circular pattern to form a hexagonal layout. From the **Linear Sketch Pattern** drop-down menu select the **Circular Sketch Pattern** command. When selected you will see the **Circular Sketch Pattern** PropertyManager. Set the “Step Number” to **6** with a “Total angle” of **360°**. Make sure the “Equal spacing” option is selected. Activate the “Entities to Pattern” box and **select the three straight lines and the two fillets** and click **OK** to finish. You now have a pattern that is the first step of the sketch for the knob outline. Notice that some of the lines may overlap as can be seen in **Figure 2-21**. You may want to trim the intersecting lines; however, that is not necessary to complete the remainder of the exercise.



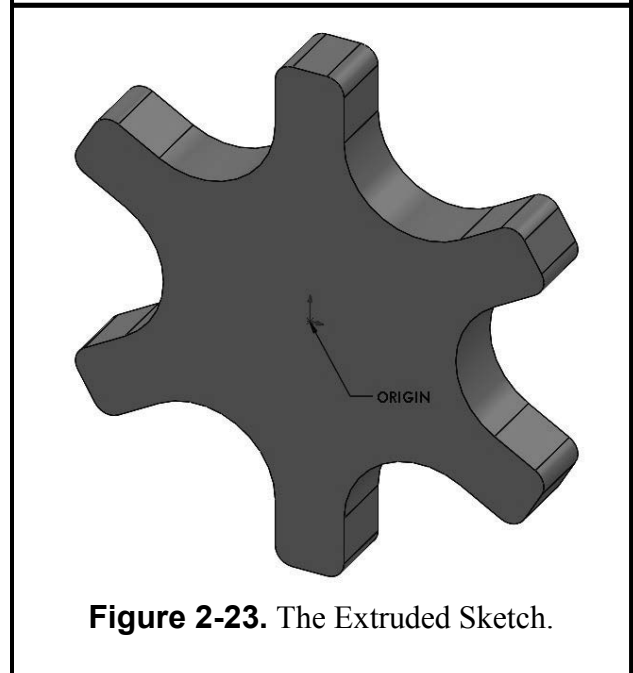
**Figure 2-21.** The Sketch after completing the Circular Pattern.

Next, you will add a fillet to the six sharp inner corners to create the scallop effect. Select the **Sketch Fillet** command and enter the fillet radius as **0.45** in the PropertyManager. Now select two intersecting lines. A large 0.45 radius is added and a dimension is attached to show the fillet value. Repeat this filleting process on the remaining five sharp inner corners. When you are finished, your sketch should look like **Figure 2-22**.



**Figure 2-22.** The Finished Sketch after Filleting Six Sharp Inner Corners.

From the **Features** tab select **Extruded Boss/Base**. Extrude the sketch using the **Blind** end condition to a depth of **0.375** inches. Click OK to finish the operation. When finished, change the to a **Trimetric** view orientation as shown in **Figure 2-23**.



**Figure 2-23.** The Extruded Sketch.

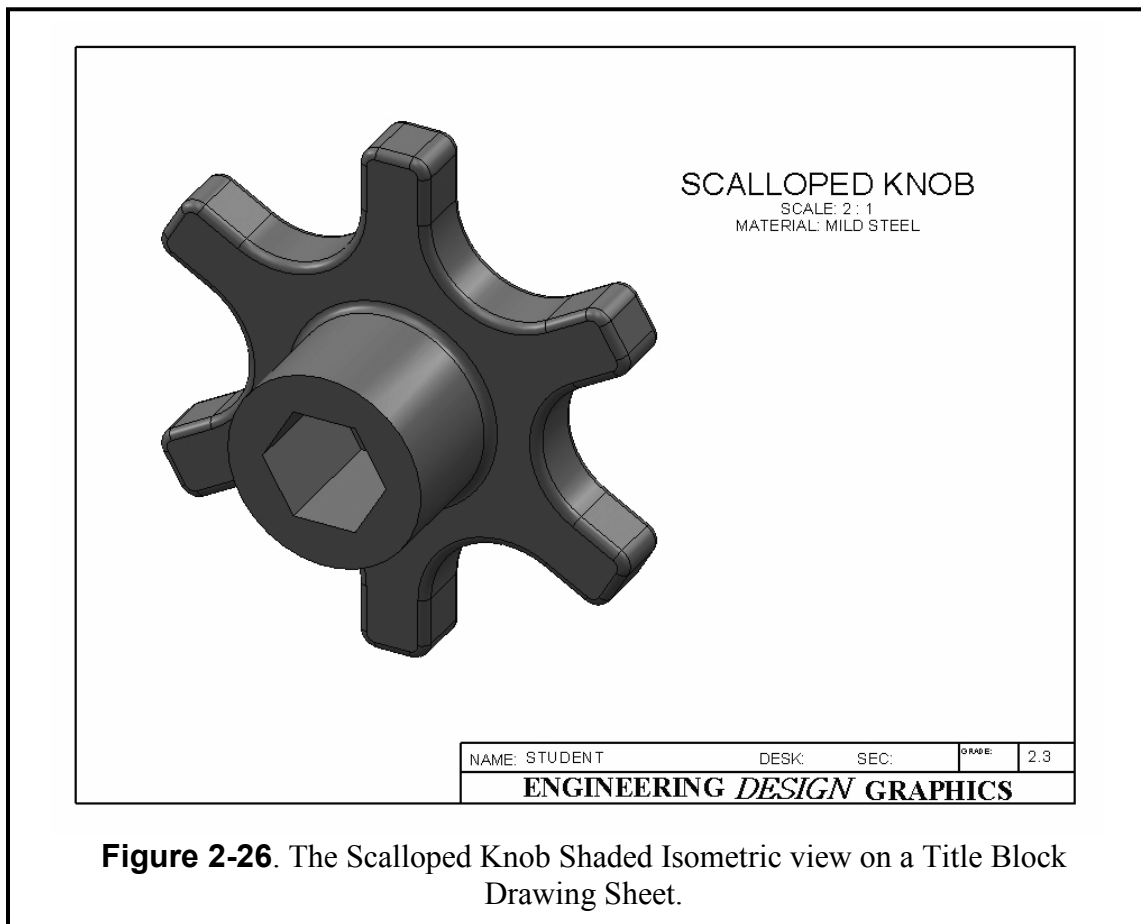
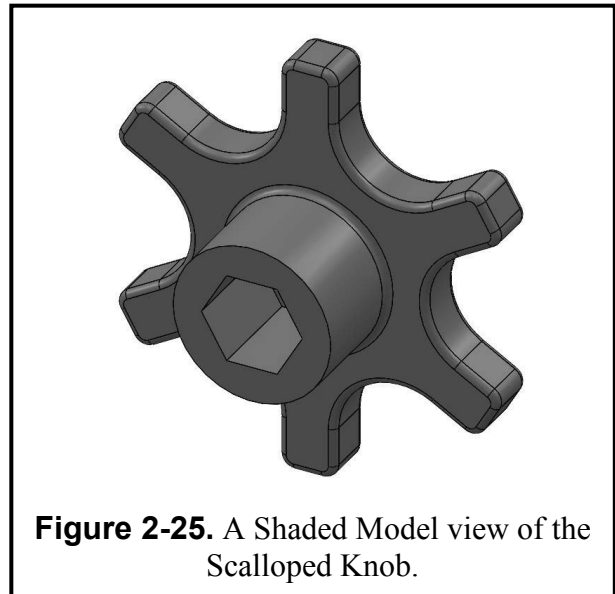
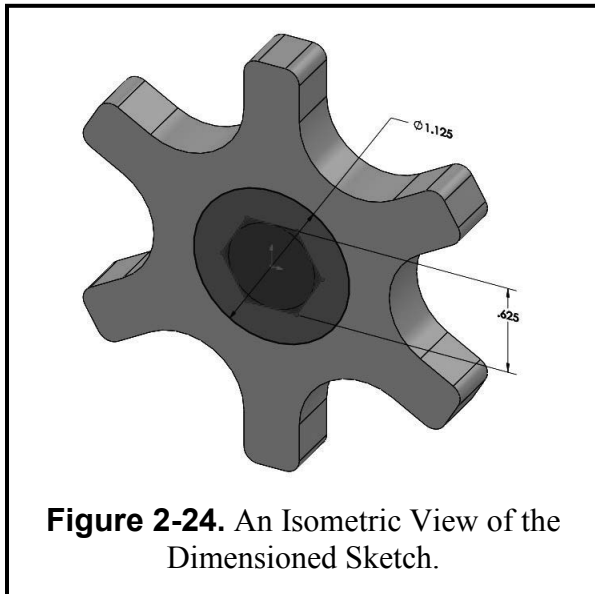
You now can finish the part by adding the attachment base. **Select the front face** and change to a **Front** view orientation. Select the **Sketch** command and draw a **Circle**, centered at the origin. **Dimension** the circle to be **1.125** inch in diameter. Now draw a **Hexagon** at the origin. Check **Inscribed Circle** and set the diameter to **0.625** inches. From the **Features** tab select the **Extrude** command. Change to an **Isometric view** for better visibility when making an extrusion of any kind. Extrude the sketch to a **Blind** depth of **.75** inches away from the front face.

Change to a **Dimetric** view orientation to see the inside of the hexagonal hole. From the Features tab select the **Fillet** command and set the radius to 0.05". Select the edges of the first extrusion with the 0.45" radius and the edge connecting the first extrusion with the second one to remove the sharp edges of the knob and click OK to finish.

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 any color you wish to the model.

Now save your model to your designated folder. Pull down **File**, select **Save As**, type in the part name **SCALLOPED KNOB.sldprt**, and then click **Save**. Open your **TITLE BLOCK – INCHES.drwdot** and immediately **SAVE AS – SCALLOPED KNOB.slddrw**. Now add a shaded Trimetric view of the Scalloped Knob onto your **Title Block** drawing sheet created in Chapter 1 (see **Figure 2-26**).

**Print** a hard copy to submit to your lab instructor.



## 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 plate, and you will find the linear pattern and mirror commands to be quite helpful. Start by going to your folder and **Open** the file **ANSI-INCHES.prt**; immediately **SAVE AS – LINEAR STEP PLATE.sldprt**. Select the **Right Plane**, change to a **Right** view orientation for better visibility and add a new **Sketch**. Draw a vertical centerline through the origin. Go to **TOOLS – Sketch Tools**, and Select **Dynamic Mirror**. Now sketch the right half of the profile shown in **Figure 2-27**. Each line drawn on the right side of the centerline will be duplicated on the left. Use the **Dimension** command to define the geometry by adding the dimensions shown in the **Figure 2-27**.

From the **Features** tab select **Extruded Boss/Base**. On the “Base Extrude” PropertyManager set the extrude parameters as shown in **Figure 2-28**:

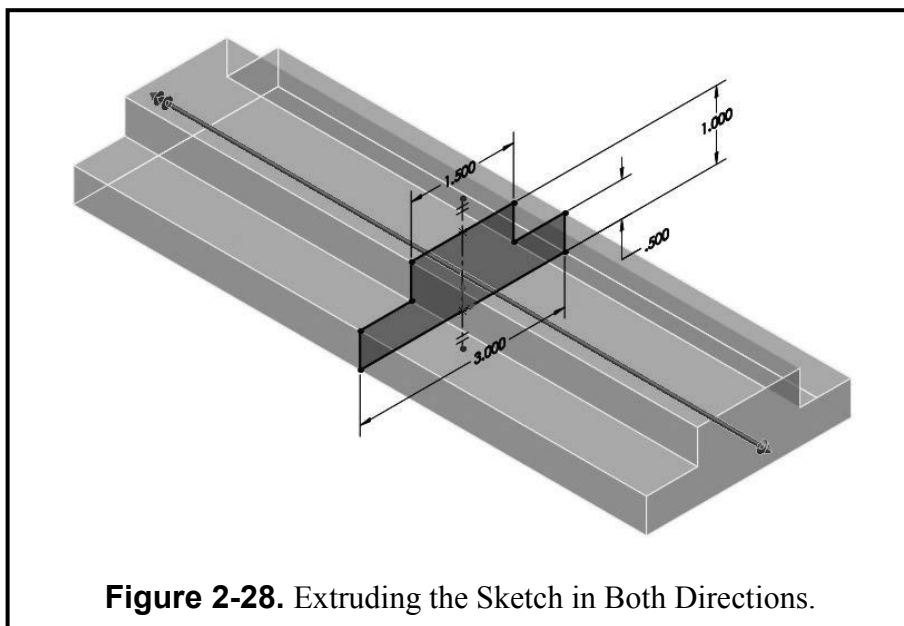
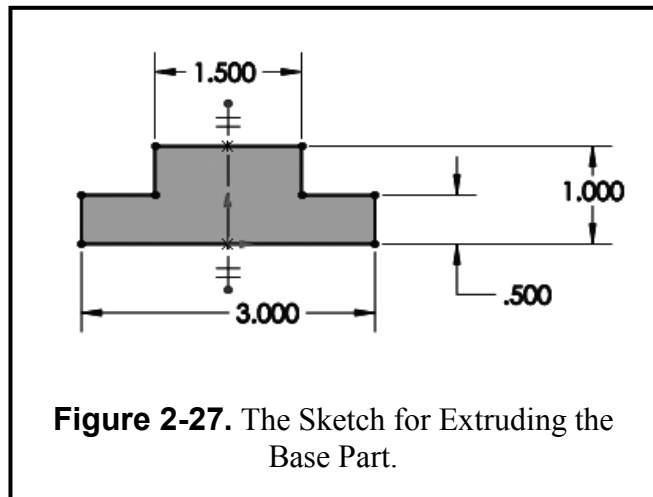
*Direction 1:* **Blind, 4.2000 in.**

*Direction 2:* **Blind, 4.2000 in.**

**OR**

**Extrude** the Sketch **8.4”** using the **MID-PLANE** end condition.

Notice that you can preview this operation in an **Isometric** view on the screen. Click **OK** to complete the extrusion in two directions. The base part looks like **Figure 2-29**.





Now you will create some holes. **Select the top surface of the small step on the front side** (see **Figure 2-29**). It should highlight *blue*. In a **Top** view, select the **Sketch** command and draw a small **Circle** on the surface as shown in **Figure 2-30**. Use the **Dimension** tool to add the three dimensions given to define it:

Diameter = **0.300**  
 From center origin = **1.125**  
 From center origin = **3.000**

Now create a Linear Sketch Pattern. **Select** the circle (it should turn *cyan*). Select the **Linear Sketch Pattern** from the Sketch tab. The “Linear Pattern” PropertyManager is displayed. Enter the following parameters and click OK:

Direction 1:

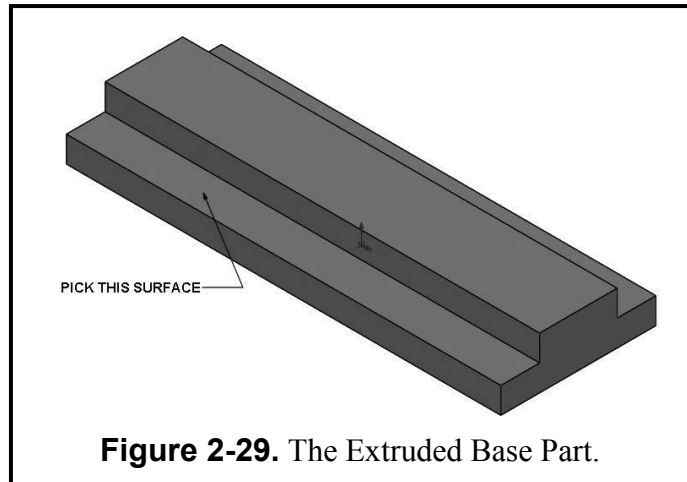
Number = **6**  
 Spacing = **1.2000**  
 Angle = repeat to right side

Direction 2:

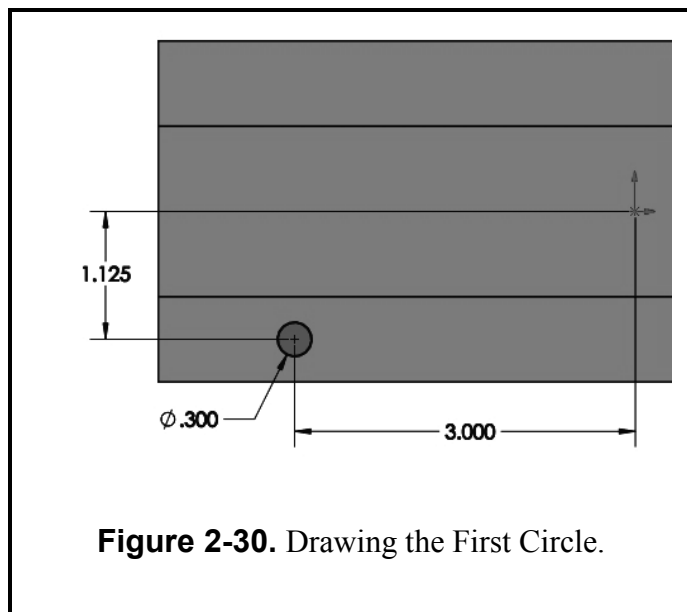
Number = 1

You should now have six circles on the front step surface. You need to add six more circles to the back step surface. You can mirror them.

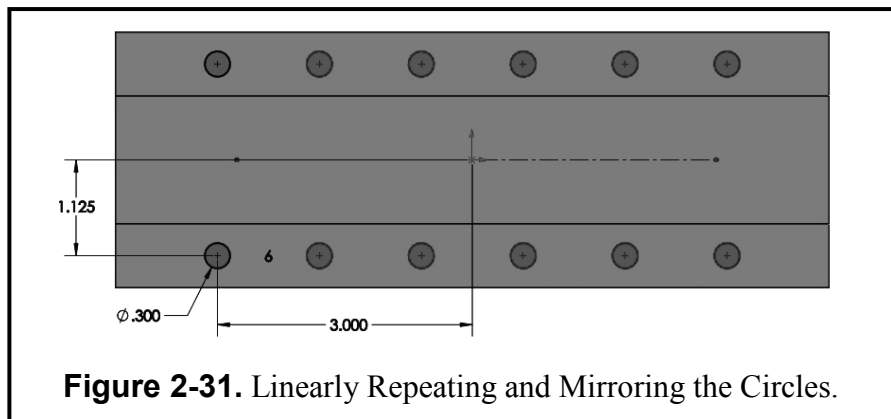
Draw a horizontal **Centerline** across the origin (Note the “—” symbol on your cursor means horizontal). Select the **Mirror** sketch command and the mirror PropertyManager is shown. For the “Entities to Mirror,” select the six circles just created with the Linear pattern and in the “Mirror About” box select the centerline drawn through the origin and click OK to finish. The selected items will be mirrored about the centerline, as shown in **Figure 2-31**.



**Figure 2-29.** The Extruded Base Part.

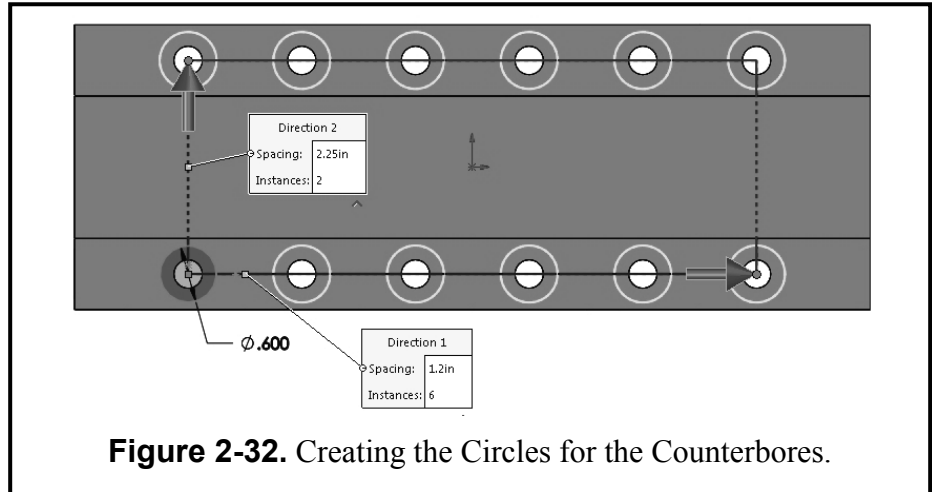


**Figure 2-30.** Drawing the First Circle.



**Figure 2-31.** Linearly Repeating and Mirroring the Circles.

From the **Features** tab select **Extruded Cut**. Use the **Through All** option and click OK to finish. You have now drilled the small holes all the way through the plate's steps. You now need to bore some counterbore holes a quarter of the way down the small through holes. Note: This design feature is called a "Counterbore" and SOLIDWORKS has a special "Wizard" that can create it. However, we will leave that "Wizard" for a later lab.



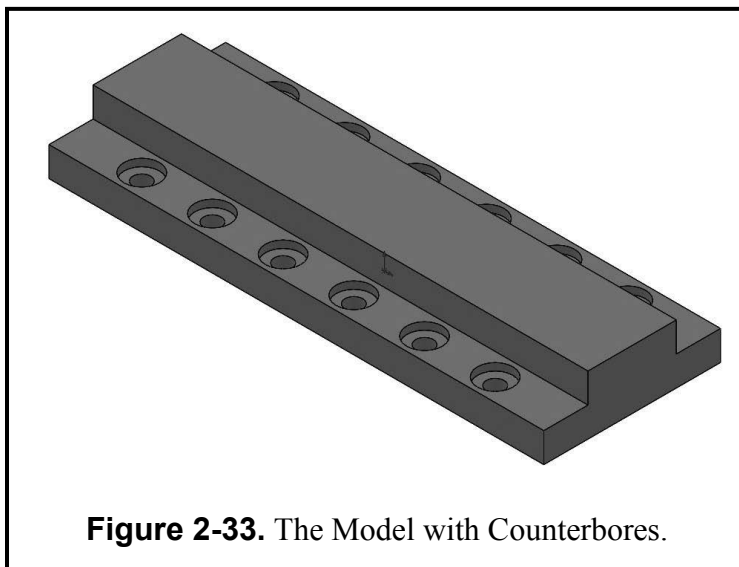
**Figure 2-32.** Creating the Circles for the Counterbores.

**Select** the top surface of the front step again. **Sketch** a **Circle** on that surface. Then add a relation to make the circle concentric with the hole beneath it (the diameter is **.60**). Now repeat the exact same process as before to get the twelve circles for the counterbore holes.

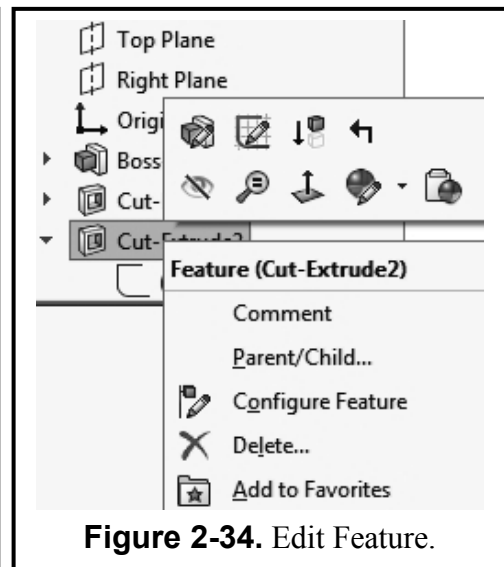
- **Select** the new circle.
- Add a **Linear Pattern** to get the front **6** circles at **1.20** inches apart in **Direction 1**.
- **In Direction 2** increase the instances to **2** at a distance of **2.25** inches.

From the **Features** tab select **Extruded Cut**. Use the **Blind** option to a depth of **0.125** inches into the material and click OK to finish. You now bored the counterbores into the plate's two steps, as shown in **Figure 2-33** in a **Rotated View**.

Note: Sometimes you might make a mistake with a **FEATURE** operation like this one. You can simply right mouse click on its name in the FeatureManager and select **Edit Feature** from the context menu. See **Figure 2-34**.

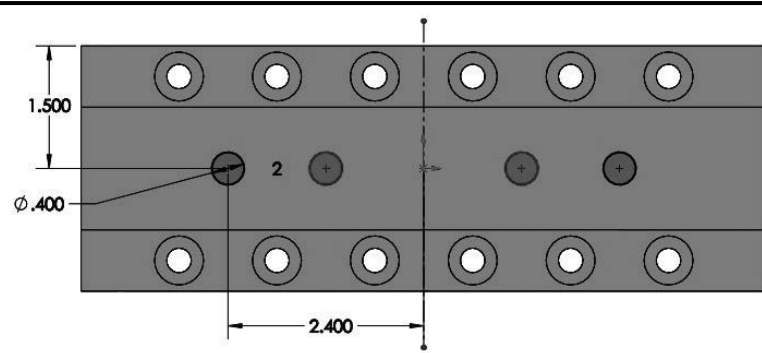


**Figure 2-33.** The Model with Counterbores.



**Figure 2-34.** Edit Feature.

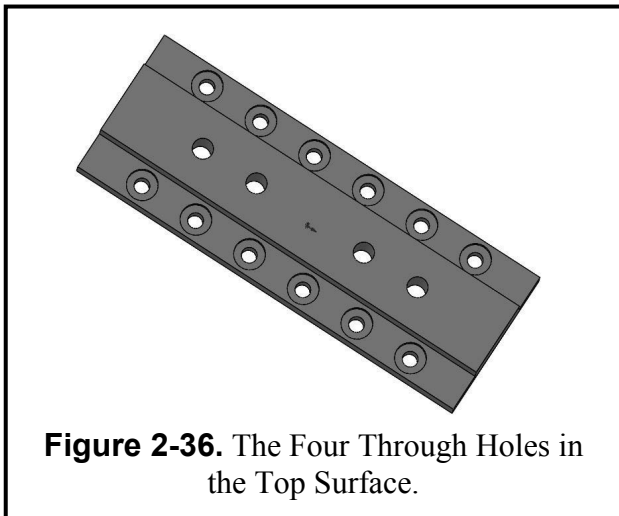
The next design requirement is to create four holes on the top of the plate. Change to a **Top** view and select the top surface to **Sketch** on. Draw the first **Circle** with the **Dimension** values given in **Figure 2-35**. Use a **Linear Pattern** operation to get a second circle **1.2000** inches from the first circle. Draw a vertical **Centerline** through the origin



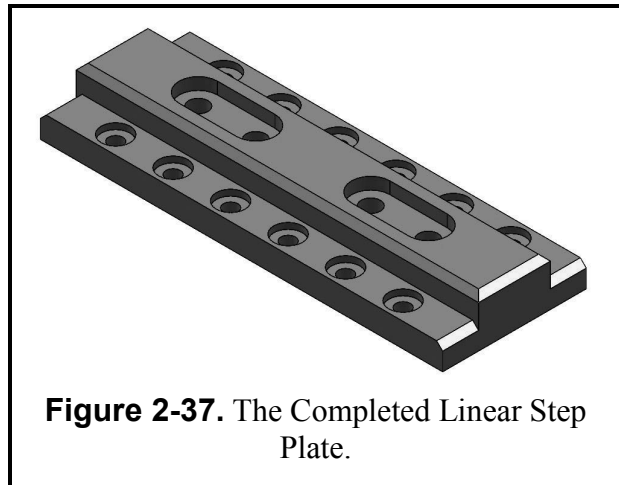
**Figure 2-35.** Creating the Holes in the Top Surface.

(a | appears on the cursor). Then **Mirror** the two circles. This results in four circles as shown in **Figure 2-35**.

Change to an **Isometric view** for better visibility. From the **Features** tab select **Extruded Cut**. Use the **Through All** option and OK to finish. You now drilled the small holes all the way through the thick part of the plate. **Select the top surface** again and begin a new sketch. You will now add two counterbore slots. Select the **Slot** command from the sketch tab and use the **straight slot** type. Select the center of the left circle on the top face and the one immediately to its right. This will make the slot concentric with the two circles to the left of the center. You can use an identical process to sketch the slot to the right of the center. Dimension the arcs of both slots to have a **Radius of .40**. Then add a **Cut Extrude** feature with a **Blind** depth of **0.250**. These counterbore slots are shown in **Figure 2-37**. To finish the step plate add a chamfer to the three horizontal edges on both ends of the model. From the **Features** tab and under the pull-down **Fillet** command, **Select Chamfer**. Set the chamfer value to **.125**, then select the three top horizontal edges of the ends of the step plate and the two long edges of the top surface. Click OK to complete the exercise.



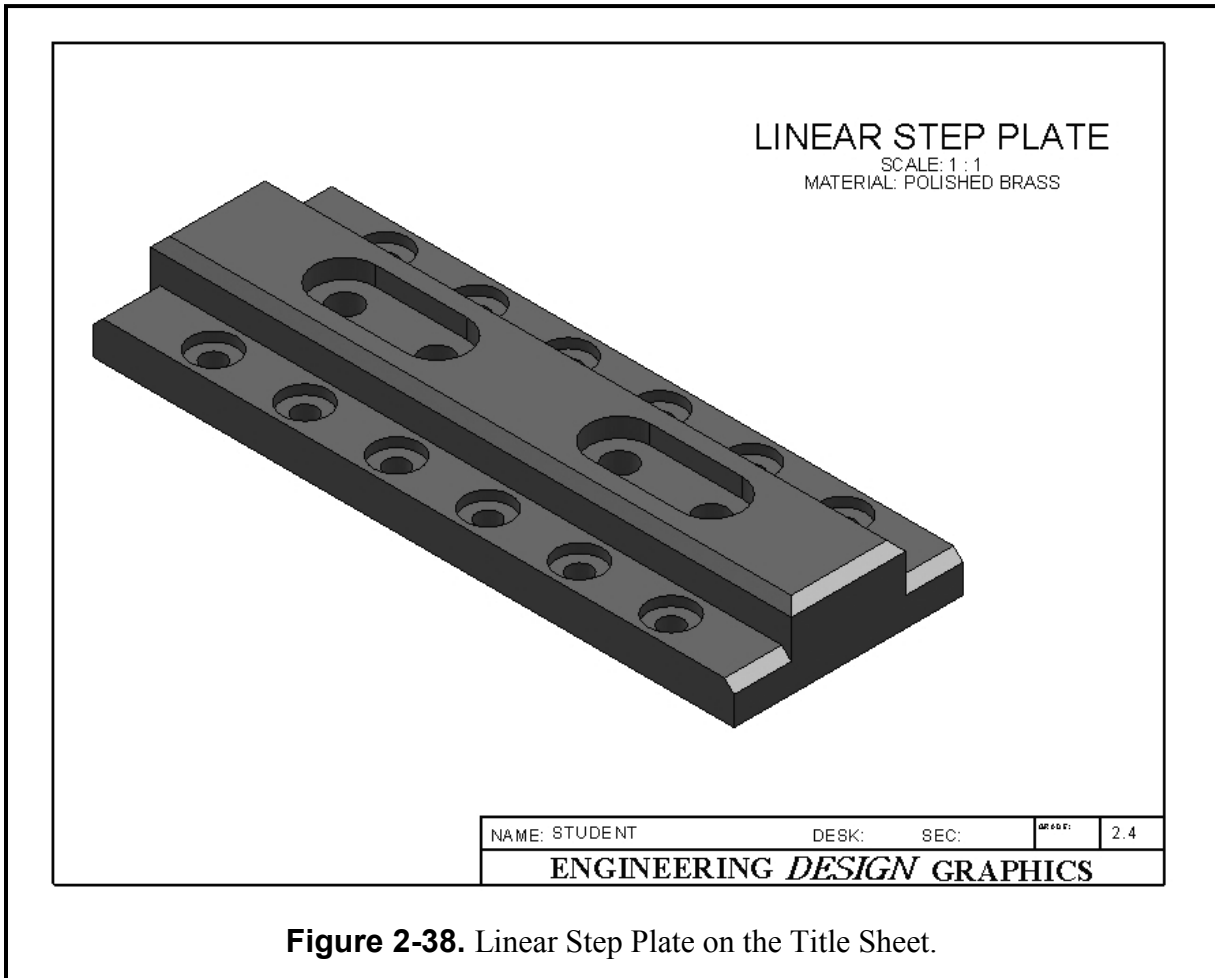
**Figure 2-36.** The Four Through Holes in the Top Surface.



**Figure 2-37.** The Completed Linear Step Plate.

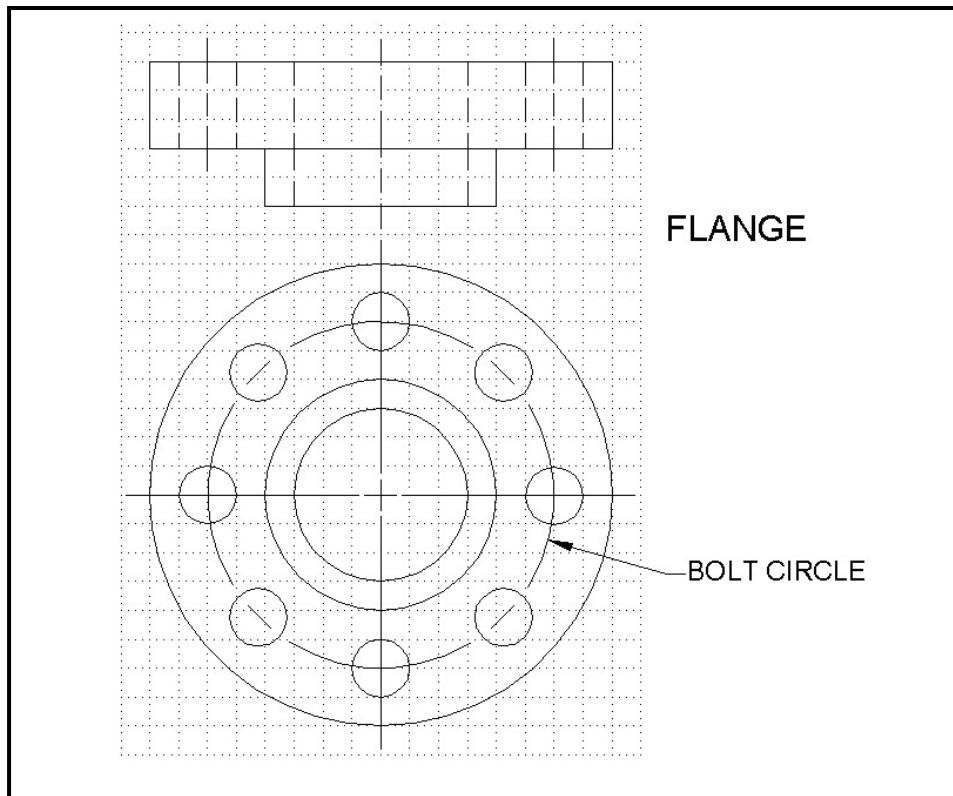
In the FeatureManager Tree, Right click on **Edit Material**, expand the **Copper Alloy** materials category and assign **Brass** to the Linear Step Plate.

The part is now finished. Return to an **Isometric** view of the finished part as shown in **Figure 2-37**. Pull down **File**, select **Save As**, type in the part name **LINEAR STEP PLATE.sldprt**, and then click **Save**. Open your **TITLE BLOCK – INCHES.drwdot** and immediately **SAVE AS – LINEAR STEP PLATE.slddrw**. Now insert the shaded Isometric Linear Step Plate view onto your **Title Block** drawing sheet created in Chapter 1 and **Print** it (see **Figure 2-38**).



## SUPPLEMENTARY EXERCISE 2-5: FLANGE

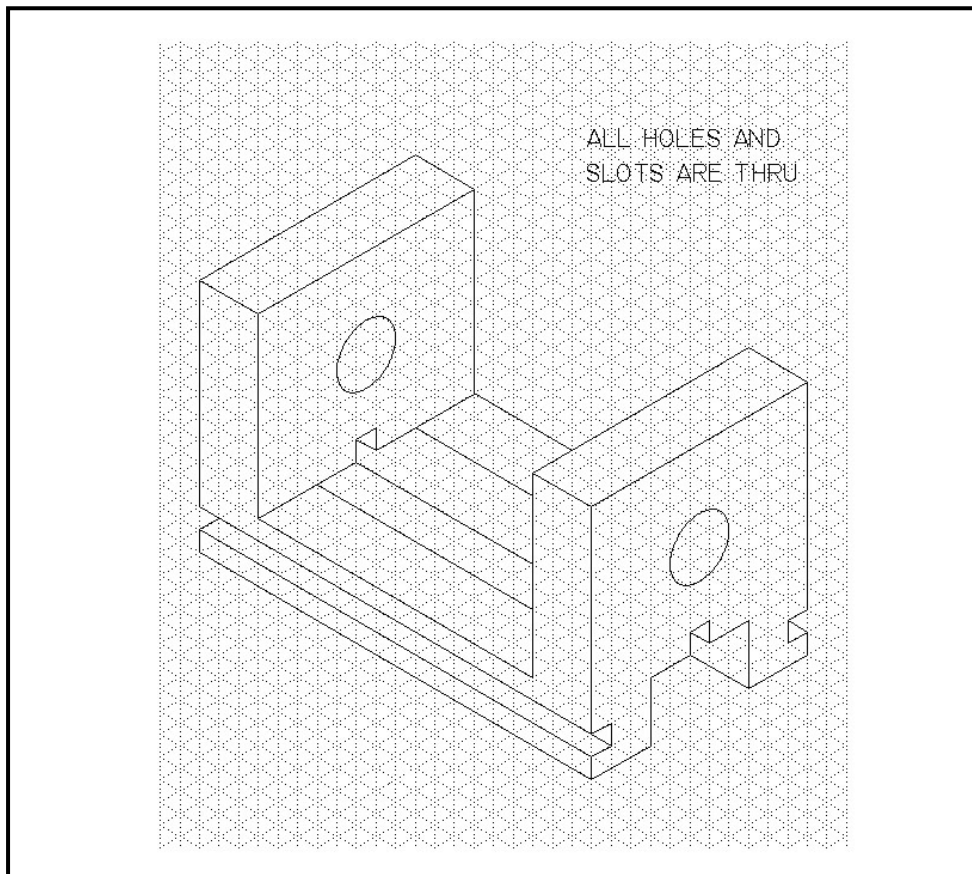
Build the following model using the **Revolve** and **Circular Pattern** commands learned in Unit 2. Add the sketch for the **Revolve** feature in the **Top Plane** using the grid divisions as a guide. Add it on a Title Block drawing and name it “**FLANGE.**”



**ASSUME THE GRID DIVISIONS TO BE 0.50 INCHES.**

## SUPPLEMENTARY EXERCISE 2-6: STEEL VISE BASE

Make a model of the figure below using the commands learned in Unit 2. Insert it onto a Title Block drawing and name it **STEEL VISE BASE**.



**ASSUME THE GRID DIVISIONS TO BE 0.25 INCHES.**