Chapter 5

The Circular Problem

In this chapter, you will learn the following to World Class standards:

- **Using the Tools and Panels for Simple 2D Computer Aided Drafting (CAD)**
- **Using the Intellisketch Settings to Set the Center Intellisketch**
- **Using a Series of Circle Commands, Both Diameters and Radii**
- **Using the Fit Command to Display the Entire Drawing**
- **Using the Offset Command to Draw a Circle**
- **Adding Centerlines Using the Line Command**
- **Using the Move and Copy Commands to Place Circles in their Appropriate Positions**
- **Using the Line, Move and Offset Commands to Add Detail**
- **Using the Fillet Command to Join Two Lines Not Intersecting**
- **Using the Trim Command to Convert Construction Lines**
- **Using the Move Elements Tool to Move an Entity to a New Layer**
- **Use the Circular Pattern Tool to Create a Circular Array**
- **Select Lines for Deletion**

The circular problem is the second problem in your training and it is just as important to achieve the drawing standards covered here as it was in the rectangular problem. Where the rectangular problem is an instruction of basic concepts in the straight lined and minor curved components, the circular problem addresses the unique challenges of mostly circular parts. The circular problem allows you to learn more skills early in your CAD drawing career, but also reinforces many concepts from your first lesson. It is this repetition that will allow you to attain the World Class standard.

Drawing Circles using IntelliSketch Icons

Setting the IntelliSketch icons allows you to select points precisely. On the **Sketching** tab, there is a panel completely dedicated to these helpful icons. The **IntelliSketch** panel has the basic icons for you to select, as well as a more advanced selection if you click on the arrow in the lower right hand corner of the panel (Figure 5.1). To view the advanced IntelliSketch options, click on the **Relationships** tab on the IntelliSketch window. For this problem, you will not need the advanced options until later. To begin drawing your circles, you need to make sure that the Center IntelliSketch option is selected.

Figure 5.1 – The IntelliSketch Panel and Bottom Right Arrow for Advanced Options

In the circular drawing, as in many drawings, it is essential that layers be created before any drawing begins. You will find that we will not have to return to change drawing entities later if the objects are placed on the correct layer, especially when there are arrays involved.

Creating Layers in a Solid Edge Drawing

You are already familiar with the concept of layers from when you created layers in the rectangular problem. Remember, you will draw the object lines on the Default layer, which is automatically set when a new file begins. You can see the current layer in the Layer Window which is a hidden tab on the left side of the screen (Figure 5.2). To create new layers, select the **New Layers** tool on the Layers Window (Figure 5.3).
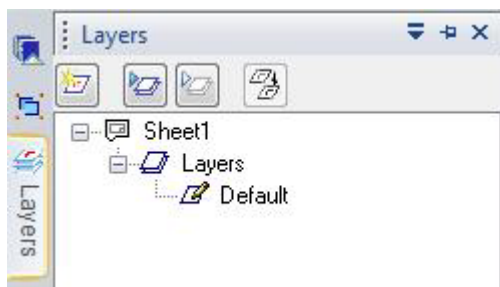

Figure 5.2 – Layers Window

Figure 5.3 – New Layer Tool

To create this drawing you need to make three new layers: the Dimension layer for dimensions, the Center layer for centerlines and the Text layer for text and notes. Begin by selecting the **New Layer** tool on the Layers Window. Immediately, the layer name is ready for entry, so type "**Dimension**" and hit **Enter**. Select the New Layer tool again and type in "**Center**" for the second new layer. Finally, make the third layer and type in "**Text**".
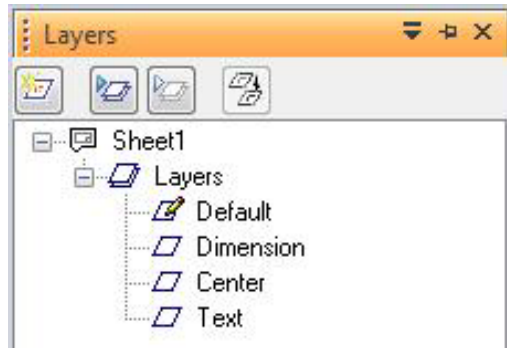

Figure 5.4 – Three New Layers

Drawing Circles Using IntelliSketch Icons

You need to draw a series of five circles using the **Circle** command that have the following diameters: 12, 5, 2.5, 0.75, and 0.625. Then you will need to draw on circle of radius 4.4375. Finally, you will draw the last circle using the **Offset** command, which you will do in the next step of this problem.

To draw the first five circles, select the **Circle** command from the **Draw** panel and the Circle by Center Point Window will appear on the left of the drawing screen (Figure 5.5). Select a point on the screen and type "**12**" in the diameter box. To finish drawing your circle, right click with your mouse and a 12 inch diameter circle will be drawn (Figure 5.6).
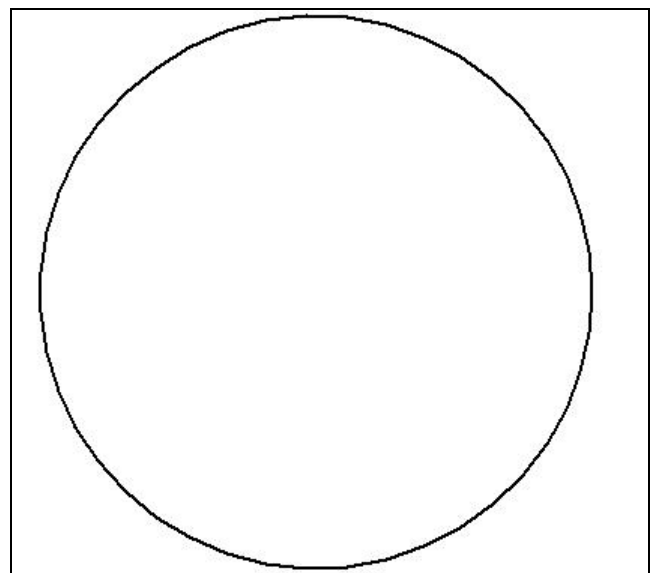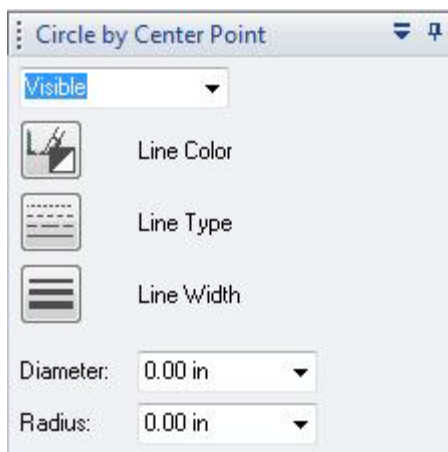

Figure 5.5 – Circle by Center Point Window   **Figure 5.6 – 12 Inch Diameter Circle**

Using the Fit Command to Display the Entire Drawing

If the 12-unit circle does not fit completely in the display, you can use one of the **Zoom** tools in the lower right hand corner of the program window (Figure 5.7). The best command to have the entire drawing fit in the display is the **Fit** tool. After fitting the drawing to the display, you can use any of the other **Zoom** tools to view the drawing how you want.
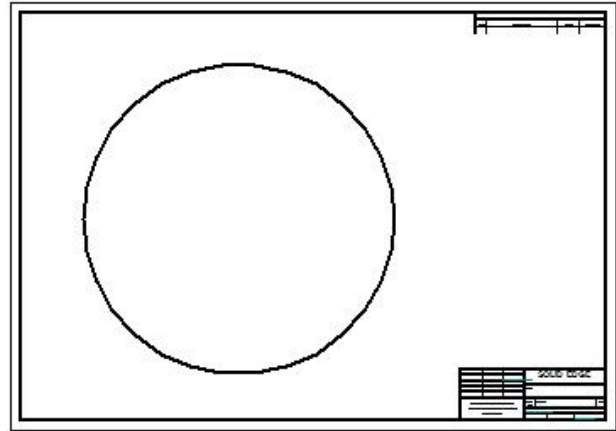

Figure 5.7 – Zoom Tools

Figure 5.8 – 12-Unit Circle in Full Display

Adding More Circles Using the Diameter Option of the Circle Command

Next, reselect the **Circle** command on the **Draw** panel. Place the cursor over the 12-unit diameter object line and the Center IntelliSketch icon should appear in the middle of your first drawn circle (Figure 5.9). Select the center point of the circle by clicking with your left mouse button. To draw the 5-unit diameter circle, you need to type "**5**" into the diameter box on the Circle by Center Point Window and hitting **Enter**. Click with the right button on your mouse to end the command. Now you need to place the other three circles in the same way. Find the Center IntelliSketch icon of the 12-unit diameter circle and draw the 2.5, 0.75 and 0.625 diameter circles. If the program automatically switches to the radius box after you have drawn one of your circles, make sure that you switch back to the diameter box before drawing the circle.

You have been using the diameter option so that you do not have to calculate the radius, which is half of the diameter. Some CAD technicians feel that they can make the mental calculation so they can avoid the extra step. Many Architects and Engineers will create mark up drawings with diameters because this dimensioning of circles conforms to better tolerance control. Remember radius dimensions with normal company tolerances will change the part size by a factor of two. Try to use diameters whenever possible when drawing and dimensioning.
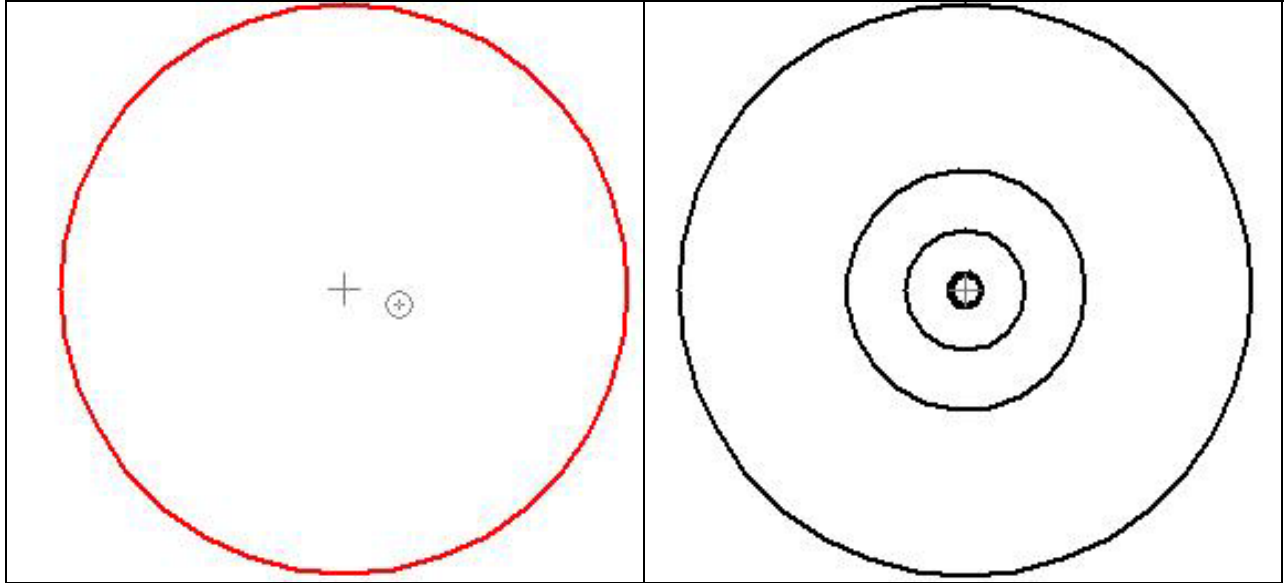
Figure 5.9 – Center IntelliSketch Icon   **Figure 5.10 – 5, 2.5, 0.75 and 0.625 Circles**

Adding a Circle Using the Radius Option of the Circle Command

You need to add a sixth concentric circle by repeating the **Circle** command. Once again, you need to select the center point of the 12-unit circle and clicking with your left mouse button. In the Circle by Center Point Window, type "**4.4375**" into the radius box at the bottom (Figure 5.5). A new large circle will be drawn between the 5 and 12-unit diameter circles.
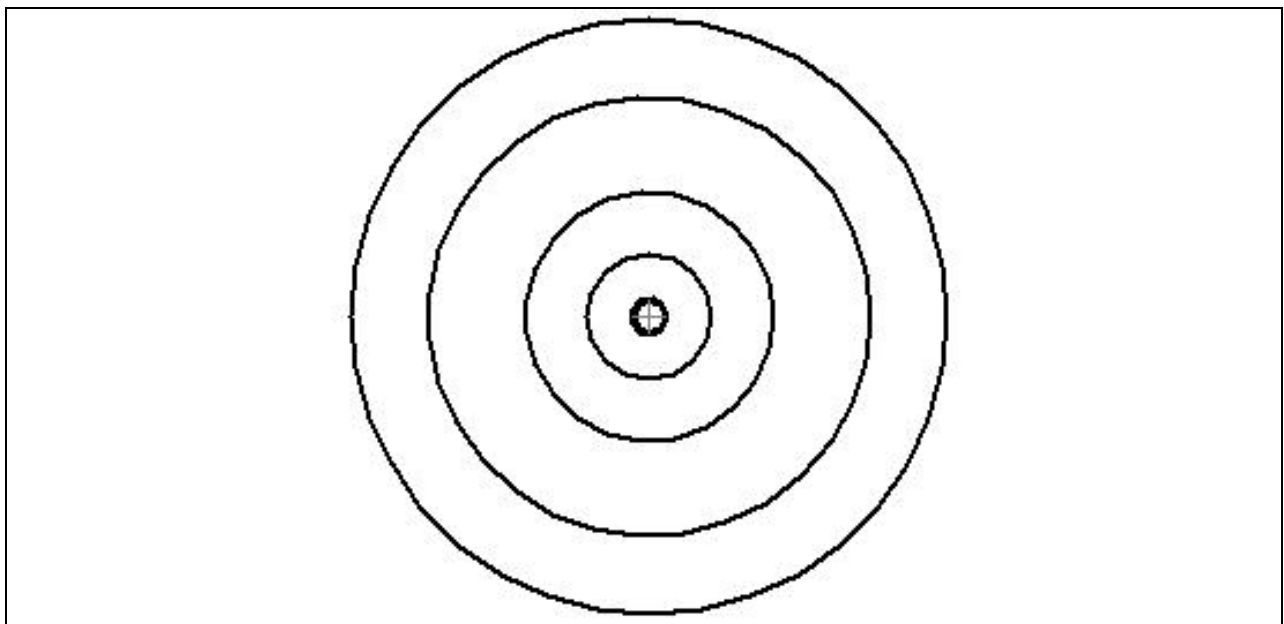
Figure 5.11 – The 4.4375 Radius Circle

Using the Offset Command to Add Another Circle

To add the seventh circle, you are not going to use the **Circle** command. Instead, you are going to use the **Offset** tool on the **Draw** panel. Select the **Offset** tool and then select the 4.4375 radius circle you just drew. In the Offset Window that appears, type "**1.125**" for the distance and then click on the "Accept" button (Figure 5.12). Now, with the **Offset** command you will have two choices; picking to the inside of the chosen circle or to the outside. In this problem, you will need to select to the outside of the circle (Figure 5.13). A new circle will be added to your drawing as shown in Figure 5.14. From the time you opened the new drawing until the seven circles are drawn, see how long it has taken you to finish these steps. Check your time against World Class standard and train yourself to draw this section of the circular problem to time and with zero errors.
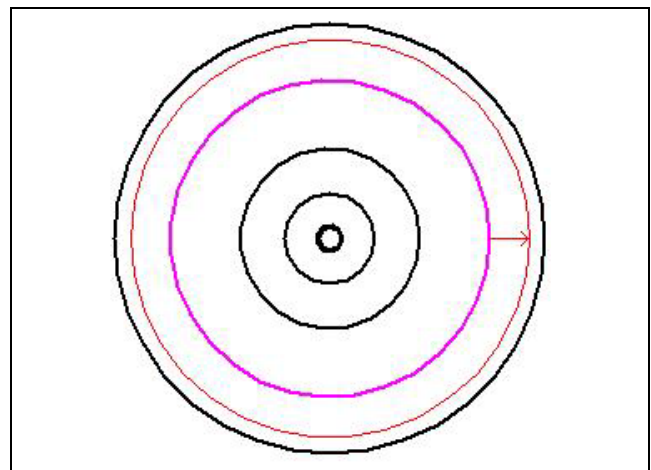

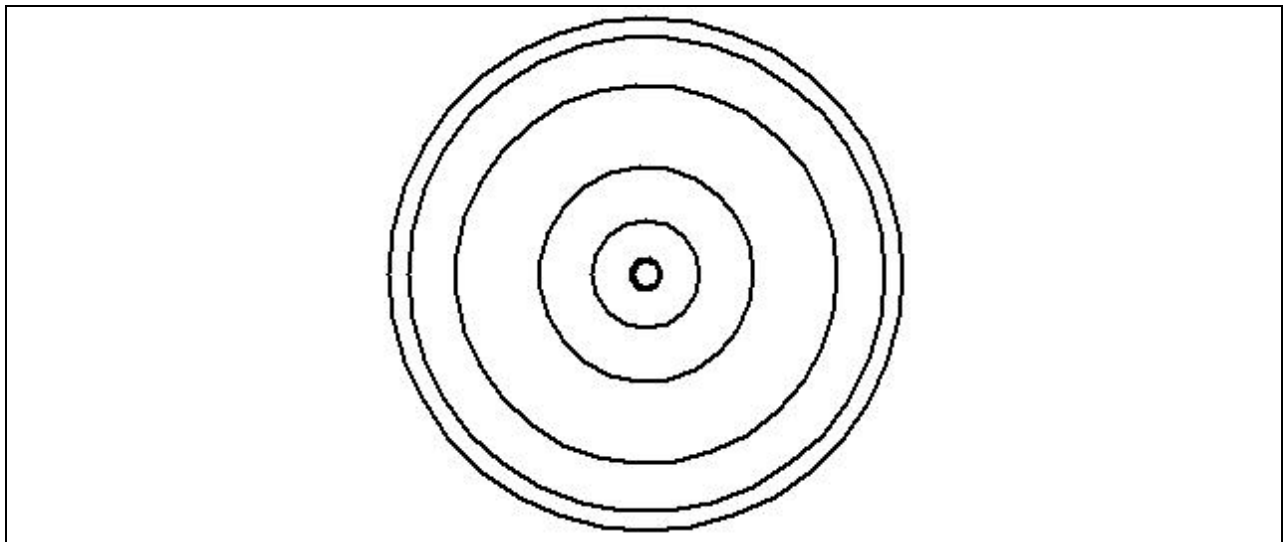**Figure 5.12 – Offset Window**   **Figure 5.13 – Offset Direction**

**Figure 5.14 – Using Offset Command to Draw the Seventh Circle**

* World Class CAD Challenge 22-10 * (See End of Chapter)

# Adding Centerlines to the Drawing Using the Line Tool

Just like the rectangular problem, centerlines are used to show symmetry and are added to assist in drawing and dimensioning. A CAD technician would draw the centerlines from quadrant to quadrant of the 12-unit circle, if he/she was following the strategy utilized in the rectangular problem. Since a circular pattern will be employed in this construction technique, you will only need to draw a single quadrant of the centerline, which will when be copied and rotated using the **Circular Pattern** command on the **Draw** panel.

First, zoom in closer to your work using the **Zoom Area** tool at the bottom right of the program window. After selecting the **Zoom Area** tool, specify the first corner and the opposite corner, picking two points that will make a window similar to the one in Figure 5.15.
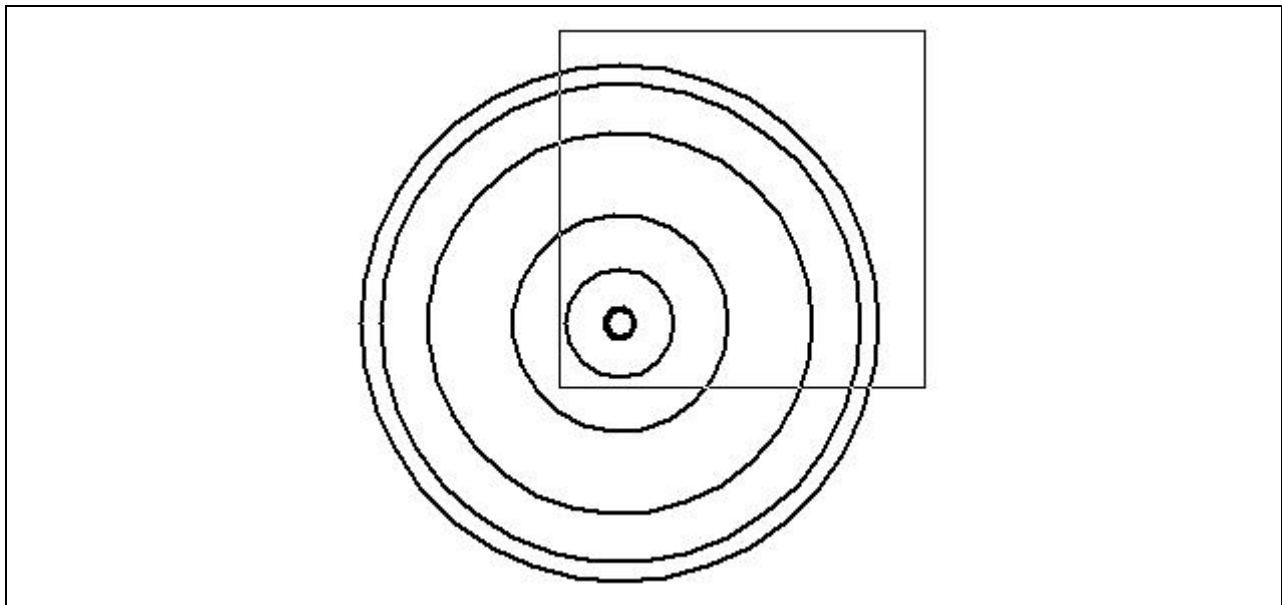
Figure 5.15 – Using the Zoom Area Command to Zoom In On Your Work
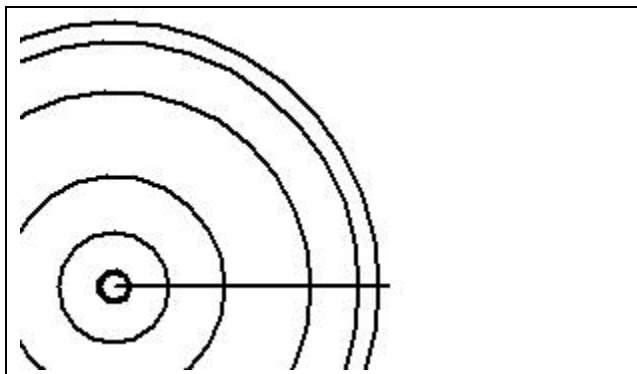
Select the **Line** command on the **Draw** panel. Select the first point of the line by placing the cursor over any of the circles, thereby causing the Center IntelliSketch icon to appear. Select the center with your left mouse button and draw the line roughly 0.25 past the outside circle. For the angle in the Line Window, make sure it is 0.00. Right click with your mouse to end the command. The line should look like Figure 5.16.

Figure 5.16 – The Horizontal Centerline

Expert operators in the World Class CAD techniques will change the IntelliSketch settings from time to time to save time when drawing. Go to the **IntelliSketch** panel on the tab and click on the lower right arrow or the advanced options (Figure 5.1). When the IntelliSketch Window appears, click on the **Relationships** tab. Make sure that the following IntelliSketch tools are selected: Endpoint, Midpoint, Silhouette point (Quadrant), Intersection point and Perpendicular. Then click "OK" to finish setting the icons.
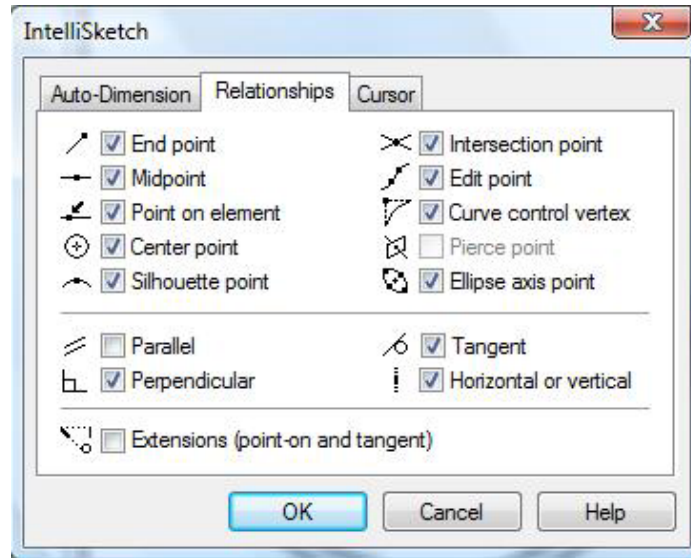

Figure 5.17 – IntelliSketch Window and Selected Icons

Using the Move Command to Place Circles in Appropriate Positions

The 0.625 and 0.75 circles need to be moved into their appropriate positions using the **Move** command. After selecting the **Move** tool from the **Draw** panel, select the 0.75 circle. After selecting the Move tool, pick the 0.75-unit circle. After picking the circle, there will be two move selections to make, both involving IntelliSketch icons.
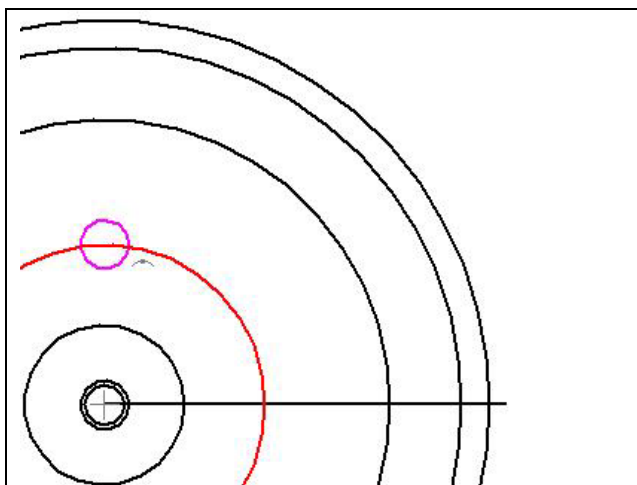

You will be asked to mark the point from where you want to move the circle, so pick the endpoint of the centerline near the center of the circles. When asked where you want to move the circle, pick the north quadrant of the 2.5-unit circle (Figure 5.18). Left click with your mouse and the command will finish leaving you with the 0.75 circle in the correct position.

Figure 5.18 – Moving the 0.75 Circle

You need to move the 0.625 circle to the 4.4375 radius circle using the same technique: select the 0.625 circle, pick the endpoint of the centerline and then pick the north quadrant of the 4.4375 radius circle. Now the 0.625 circle is moved to the appropriate place as in Figure 5.19.
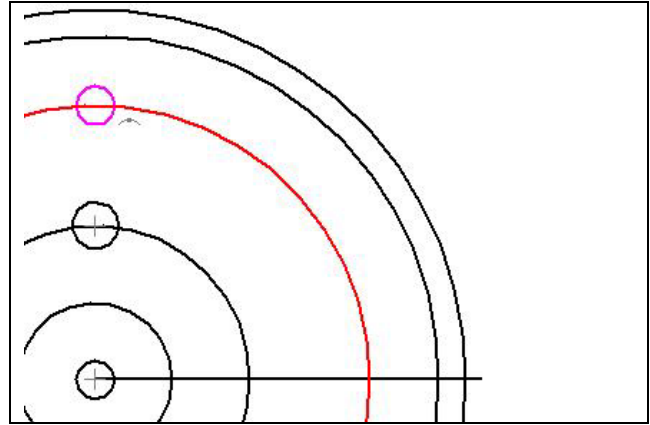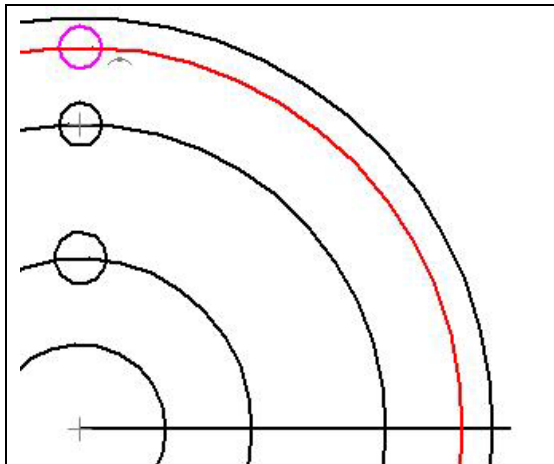
Using the Copy Command to Add a Circle in a Proper Position

To create a slotted hole in this part, you need to copy the 0.625 circle to the north quadrant of the next circle. Now, the **Copy** tool is part of the **Move** tool. In the upper left hand corner of the Move Window is the **Copy** tool. Select the **Copy** tool and then just follow the normal **Move** procedures to move and copy the 0.625 circle to the north quadrant of the 1.125 offset circle as in Figure 5.20. After have moved and copied the circle, right click with your mouse to end the command.

Figure 5.20 – Copying the 0.625 Circle

* World Class CAD Challenge 22-11 * (See End of Chapter)

Using the Line Command to Add Detail

You will now use a series of lines in your circular part to add details such as centerlines, the slotted hole and a keyway to prevent the part from slipping during rotation. First you will draw a center line for the 0.75 circle. In the rectangular problem, you used the center mark provided on the Annotation panel. The center marks have two lines perpendicular to each other and are appropriate in rectangular shapes, but in circular parts, center marks are a combination of straight lines intersecting the circular part's center and arced segments defining the distance from the parts center.
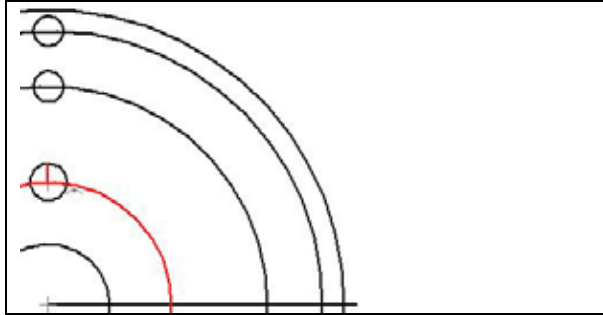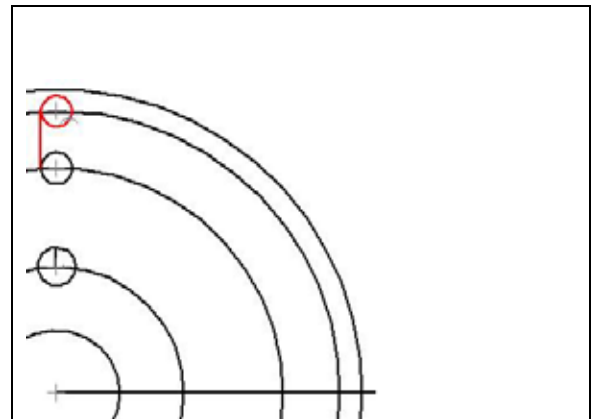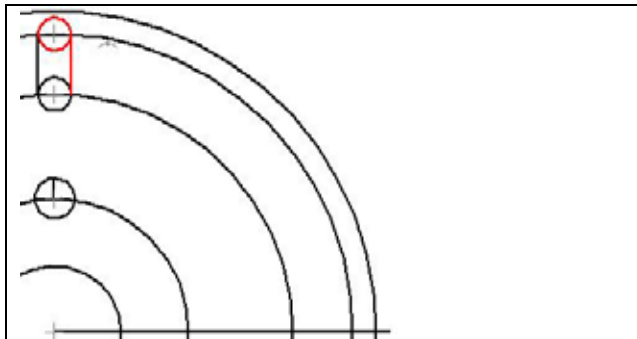
Draw a line segment by picking the **Line** tool on the **Draw** panel and picking the north quadrant of the 5.0 diameter circle. Draw the line to the north quadrant of the 0.75 circle as shown in Figure 5.21 and left click with your mouse. Right click with our mouse to end the command. Next, you will proceed to detailing the two 0.625 holes.

Figure 5.21 – Drawing the Center Mark

To continue forming the slotted hole, you need to draw lines from the tangent points of each 0.625 circle. In most cases, the Quadrant IntelliSketch tool will remain as a primary tool to draw slotted details. Draw a line segment by picking the **Line** tool from the **Draw** panel and picking the west quadrant of the lower 0.625 circle. Draw the line to the west quadrant of the upper 0.625 circle as shown in Figure 5.22 and left click with your mouse. Once again, right click with your mouse to end the line segment.


**Figure 5.22 – Drawing the First Slot Line**

To add the second line, draw a line segment by picking the **Line** tool and selecting the east quadrant of the lower 0.625 circle. Draw the line to the east quadrant of the upper 0.625 circle as shown in Figure 5.23 and left click with your mouse. Right click to end the line segment.

Figure 5.23 – Drawing Second Slot Line

Keyways are present on many circular parts and in this case one half of a key would be in the shaft and the other in the circular part. The key prevents slippage during the rotation of the assembly. In Figure 5.24, you see a line segment of any length drawn vertically from the west quadrant of the 2.5 circle. Again, select the **Line** tool. Pick the west quadrant of the 2.5 circle and draw a line vertically.
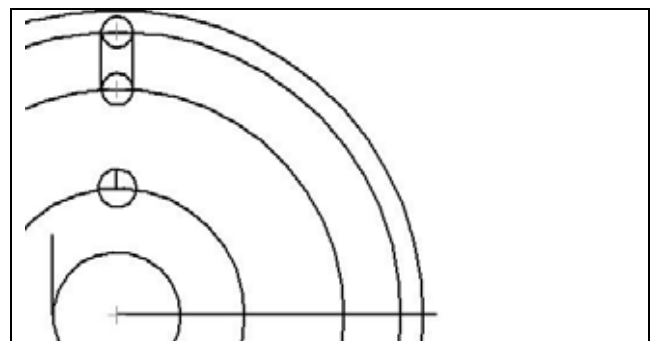
Figure 5.24 – Draw the Keyway Line
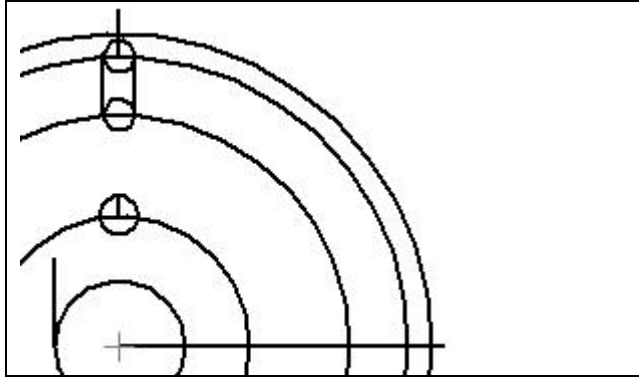
**Figure 5.25 – Drawing a Construction Line**

The top of the slotted hole at the crown of the circular part will be open, so you will draw a centerline to construct the detail. Select the **Line** tool from the **Draw** panel, picking the north quadrant of the offset circle that crosses through the upper 0.625 circle. For the second point of the line, pick any point above the perimeter of the outside circle (Figure 5.25).

Using the Move Command to Add Detail

As you continue to learn to group commands for efficient computer aided drawing, you need to move the keyway line into position.

Select the **Move** tool on the **Draw** panel and select the keyway line. When asked for the point from where you want to move the line select the top endpoint of the keyway line. After that, move the line to the right with your mouse and type "**2.687**" into the "**X:**" textbox on the Move Window. Left click to move the part, making sure that it is being moved in a straight line across your drawing (Figure 5.26).
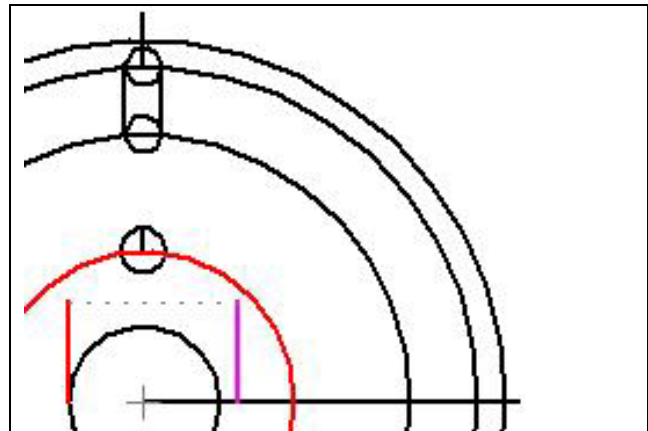
Figure 5.26 – Moving the Keyway Line

You need to reposition the centerline drawn on the 0.75 circle by selecting the Move tool. Select the center mark line, and then pick its midpoint using the IntelliSketch icon. After that, move the line down with your mouse and pick the Endpoint IntelliSketch icon by left clicking with your mouse. The centerline is now in the proper location (Figure 5.27).
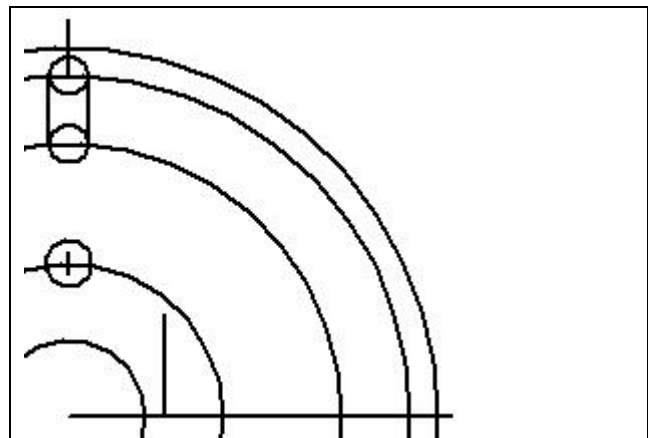
Figure 5.27 – Moving the Center Mark

Using the Offset Command to Add More Detail

When your initial lines are drawn accurately then a command such as **Offset** will add additional detail to the drawing without the numeric baggage associated with the base **Line** command.

It just so happens that the width of the slot breakout from the centerline and the distance of the keyway width from the centerline is the same measurement of 0.1875. Select the **Offset** tool on the **Draw** panel and type "**0.1875**" in the distance textbox on the Offset Window. Next, select the keyway centerline, hit the Accept button on the Offset Window and pick above the original centerline. Right click to restart the command. Pick the original line again, hit Accept and draw another line below the original centerline.
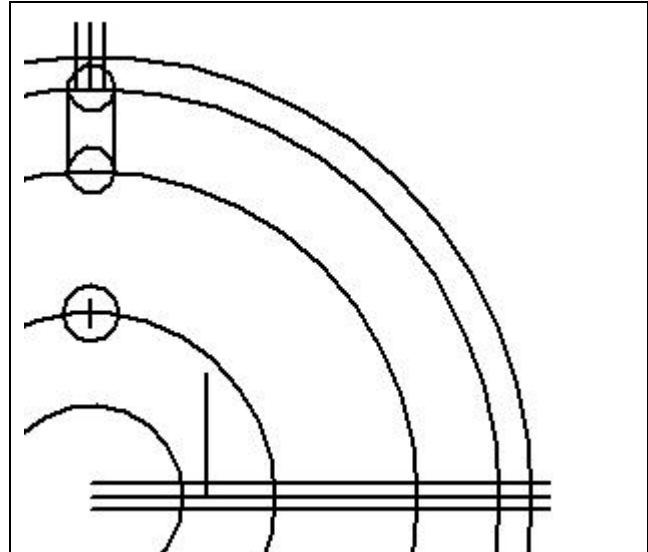

Figure 5.28 – Offsetting Lines

You do not have to exit the **Offset** command to open up the top of the slotted hole since the detail has the same offset distance. Select the slotted hole's centerline and then hit the Accept button on the Offset Window. Move the line to the left and left click with your mouse to move the line. Right click with your mouse to stop the command. Reselect the original center line, hit Accept and offset it to the right. Left click and then right click to end the command (Figure 5.28).

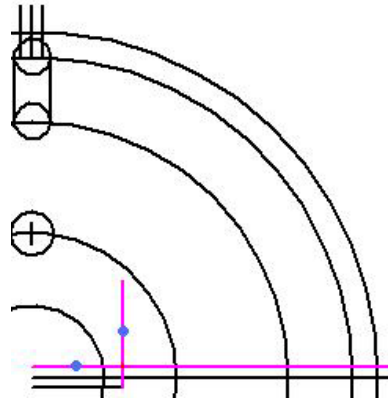*** World Class CAD Challenge 22-12 * (See End of Chapter)**

Using the Fillet Command to Join Two Lines Not Intersecting

The **Fillet** command, when the radius is set to zero, becomes an affective trimming tool and joining tool. You will use the **Fillet** tool to form the keyway. However, in the Solid Edge program, a fillet radius of zero creates an error in the drawing. To compensate, you will use a radius of 0.01. This is a positive value that is only slightly greater than zero so the program will accept it and the trimming the part calls for can still be achieved. This also the first time you will be using the **Fillet** command to join two line segments that could intersect, but do not.
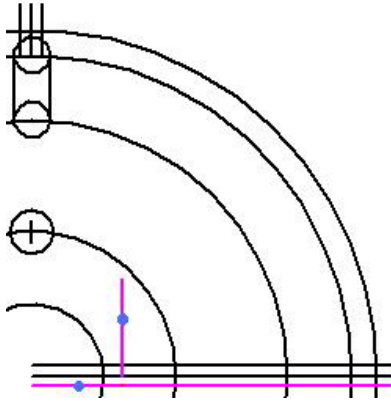
**Figure 5.29 – Forming the Key by Filleting**   **Figure 5.30 – Forming the Key by Filleting**

In Figures 5.29 and 5.30, corners are created by using the 0.01 radius of the **Fillet** tool and filleting the keyway and keyway offset lines. Select the **Fillet** command on the **Draw** panel and type "**0.01**" in the Radius textbox at the bottom of the Fillet Window. As in Figure 5.29, select the lower offset line and the keyway line to form the first corner. When placing the direction of the fillet, left click with your mouse on the left side of the keyway line. Once the first corner is drawn, select the upper offset line and the keyway line (Figure 5.30). Left click to the left of the keyway line and below the lower offset line to place the filleted corner.

Using the Trim Command to Convert Construction Lines

In efficient Computer Aided Design, you will use construction lines to create a new drawing, but what differentiates the World Class CAD method from other training courses is the conversion of construction lines into object lines. Therefore, you will rarely find yourself erasing unnecessary lines, arcs or circles.
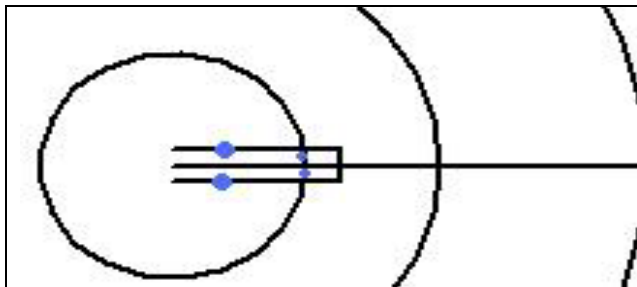
To remove the two line segments entering the 2.5 diameter circle and open the circle to the 0.375 key width, use the **Trim** tool on the **Draw** panel. You need to trim four lines; the two-keyway centerline offset lines entering the 2.5 diameter circle, and the two arc segments between the two offset lines (Figure 5.31).

Figure 5.31 – Forming the Key by Trimming

To create the slot shown in Figure 5.33, you need to remove ten line and arc segments. Repeat the **Trim** tool on the **Draw** panel. Now, make the ten selections as shown in Figure 5.32.
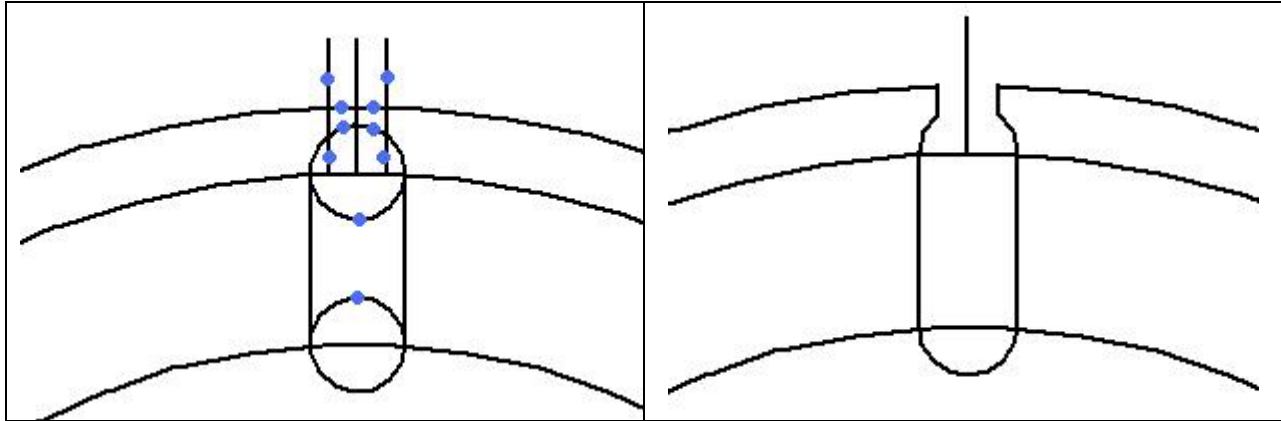
Figure 5.32 – Detailing the Slot with Trim   **Figure 5.33 – Detailed Slot**

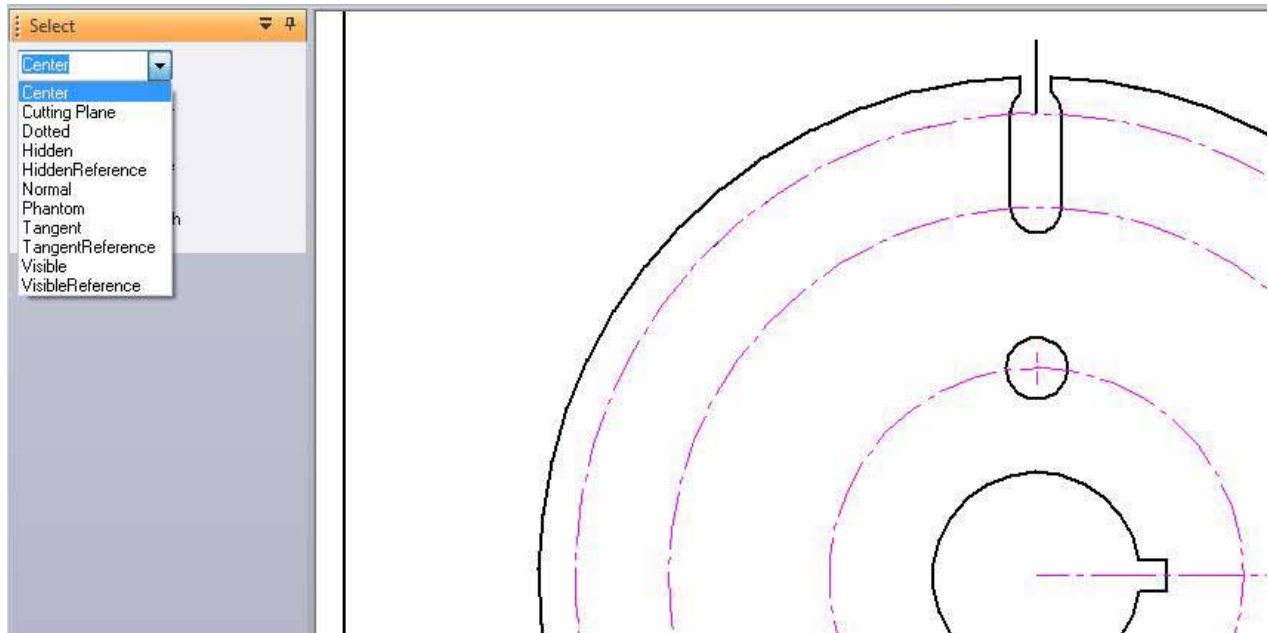# Using the Move Elements Tool to Change an Entity's Layer

Figure 5.34 – Changing the Lines and Arcs to the Center Line Type

You may have already noticed by accident that drawing entities, such as lines, arcs and circles can be selected prior to choosing a Solid Edge tool. In a situation where you need to change the line type of lines and circles and put them on the Center layer, the selections are made first. Select all three middle circles, the single centerline and the 0.75 circle's centerline. In the Selection Window, click on the top drop down arrow and select the Center line type (Figure 5.34). With the objects still selected, go to the Layers Window on the left part on the screen. On the top of the window, select the **Move Elements** tool. In the Move Elements Window that appears, select the Center layer and click "OK".
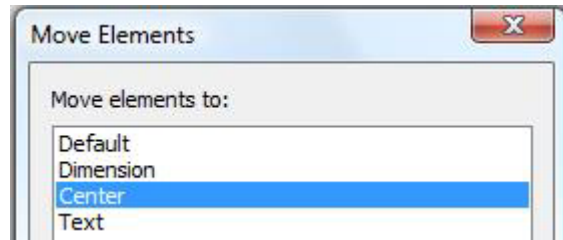
Figure 5.35 – Move Elements Tool

Figure 5.36 – Move Elements Window

Using the Circular Pattern Command to Create a Circular Array

Changing the entities to the proper layer will save time since the **Circular Pattern** command will create multiple copies of lines and circles many times over. In the rectangular problem, you were exposed to the rectangular pattern tool. Now, you will perform four separate circular patterns, the first to copy the 0.75 diameter circle and centerline six times. After finishing the pattern, you will have to ungroup the arrayed objects before you can trim away the unneeded vertical centerlines. Next, you will conduct a second pattern on the major centerline. The third circular pattern will both rotate and copy the slotted hole and vertical centerlines twenty degrees counterclockwise. After erasing superfluous lines and arcs, the fourth pattern will finish a complex circular part containing hundreds of entities.
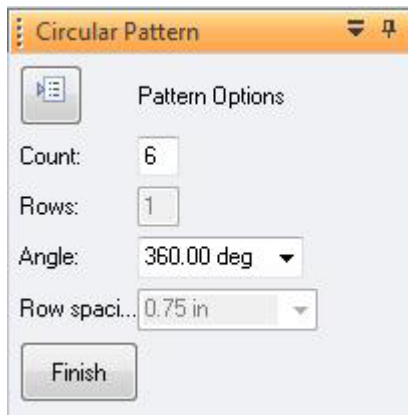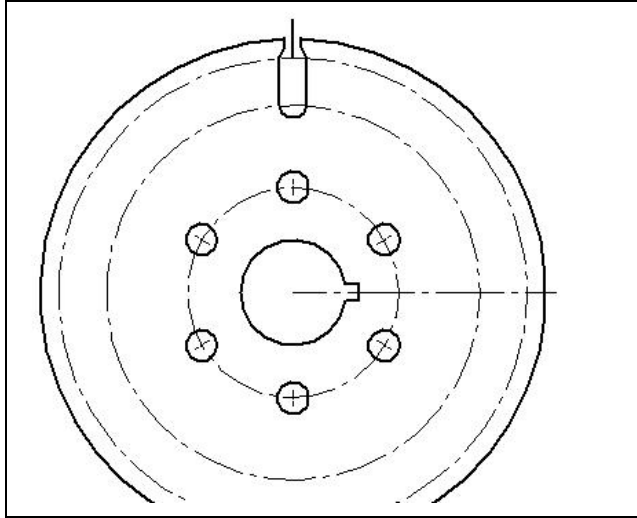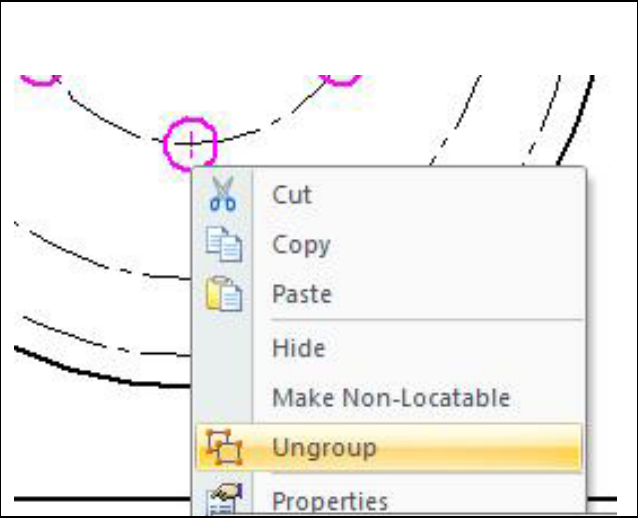

Figure 5.37 – Circular Pattern Window

First, select the 0.75 circle and its centerline. Next, Select the **Circular Pattern** tool on the **Draw** panel and the Circular Pattern Window will appear on the left of the screen. Also, pick the endpoint of the main centerline as the center point for you circular pattern. In the Count box type "**6**" .and in the Angle box type "**360.0**" (Figure 5.37). Click the Finish button at the bottom of the window and the pattern will appear as in Figure 5.38.
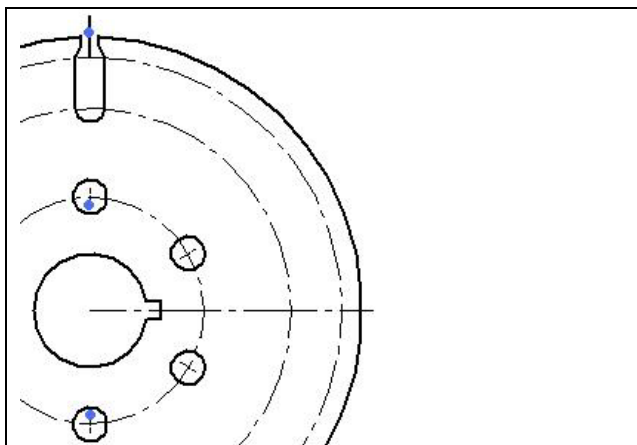
Figure 5.38 – Finished Circular Pattern   **Figure 5.39 – Ungrouping the Pattern**

In order to proceed to the next step of the problem, you are going to have to ungroup the circles and their centerlines. You may have noticed that when you place the cursor over one of the 0.75 circles, all of them are selected. This is because they are all part of a group created by the circular pattern. To ungroup the circles and centerlines, right click with your mouse on one of the circles. Then, click on the Ungroup button on the list of actions (Figure 5.39). This will separate all of the circles. Next, you need to separate the circles from their centerlines. To do this, you have to right click and Ungroup each circle individually. With all the parts ungrouped, continuing with the problem will be much easier.

Selecting Three Lines to be Deleted

When creating a drawing that will result in a complex Computer Aided Design image, you will find out many weeks from now that having a smaller line segment hidden below an object or dimension line can cause problems. You have arrived at a moment when it is necessary to erase two centerlines and a construction line.


Pick the **Select** tool on the **Select** panel. Next select each of the lines either together using the **Shift** key or individually. When the object is selected, hit the **delete** key on your keyboard and it will erase the object that would have been covered up by the main centerline from the drawing. Delete the three lines as indicated by Figure 5.40. Next, you will perform the second of the four circular patterns; copying and rotating the main centerline around the circular part.

Figure 5.40 – Erasing Three Lines

5-16

# Using the Circular Pattern Command to Create More Circular Arrays
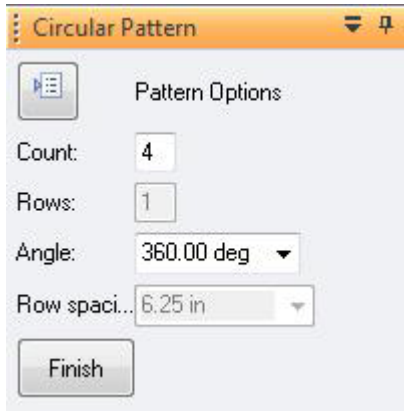
As before when making circular patterns, you must select the object(s) to be arrayed before picking the command so begin by picking the main centerline. With the centerline selected, click on the **Circular Pattern** tool. Also, pick the endpoint of the centerline as the center point for the array. Now, in the Circular Pattern Window that appears on the left of the screen type "**4**" in the Count box and type "**360**" in the Angle box. When the window appears as in Figure 5.41, click the finish button and your circular pattern will be complete.
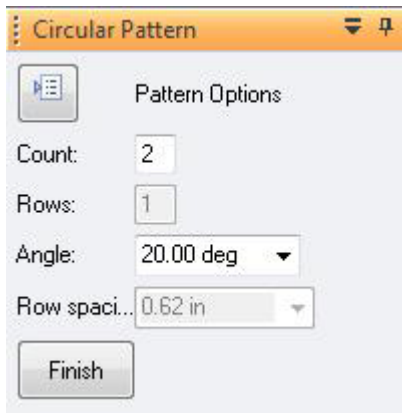
Figure 5.41 – Window for Centerline Pattern
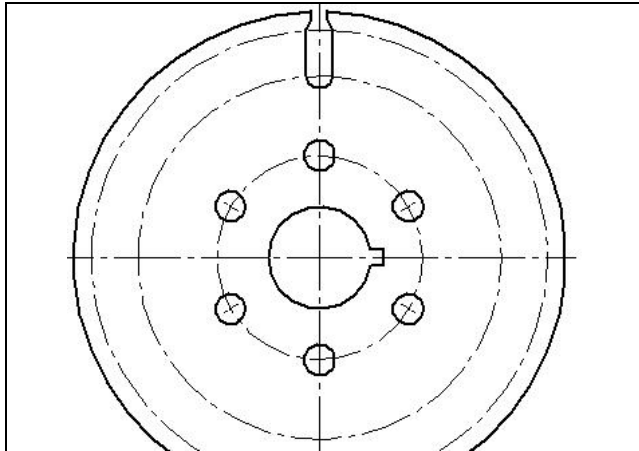
Just like with the 0.75 diameter circles and their centerlines, the pattern involving the main centerline is all one group. The ungrouping process for this circular pattern is much easier. All you need to do is right click with your mouse on one of the centerlines. Once again, a list of options will drop down. You need to select the Ungroup tool to make all the centerlines their own entity.

Figure 5.42 – Finished Circular Pattern for Centerline
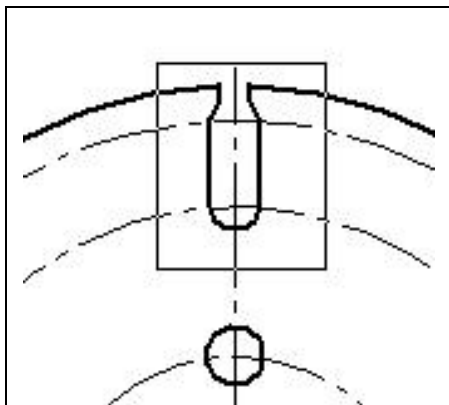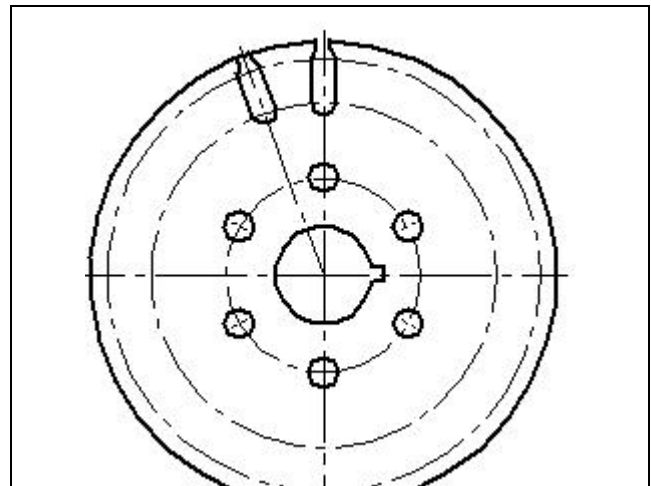

For the third circular pattern, select the slotted hole using a selection window and also select the top centerline (Figure 5.43). With your selections made, pick the **Circular Pattern** tool on the **Draw** panel and the Circular Pattern Window will appear. Pick the endpoint of one of the centerlines as the center for the array. In the window, type "**2**" in the Count box and type "**20**" in the Angle box. When the window appears as in Figure 5.44, click the finish button to complete the command.

Figure 5.43 – Selecting Slotted Hole and Centerline

Figure 5.44 – Window for 3ʳᵈ Pattern　　**Figure 5.45 – Single Copy of Slot**

Once again, you are going to have to ungroup the entities of the circular pattern. Begin by right clicking with your mouse on the top centerline. On the list of options that drops down, select the Ungroup tool. You will also have to ungroup each slotted hole from its centerline so repeat this ungrouping two more times, once for each slotted hole.

Using the Trim Command to Remove and Delete Arcs and Lines

Next, select the **Trim** tool on the **Draw** panel and trim away the two parts of the 12 diameter circle that are closing the new slot. Next, trim the outside perimeter of the 12 diameter circle except for the smaller portion between the two slots (Figure 5.46). After the circle is completely trimmed, pick the **Select** tool and select the entire second slot (Figure 5.47). With the slot selected, hit the delete key on your keyboard to erase the slot.
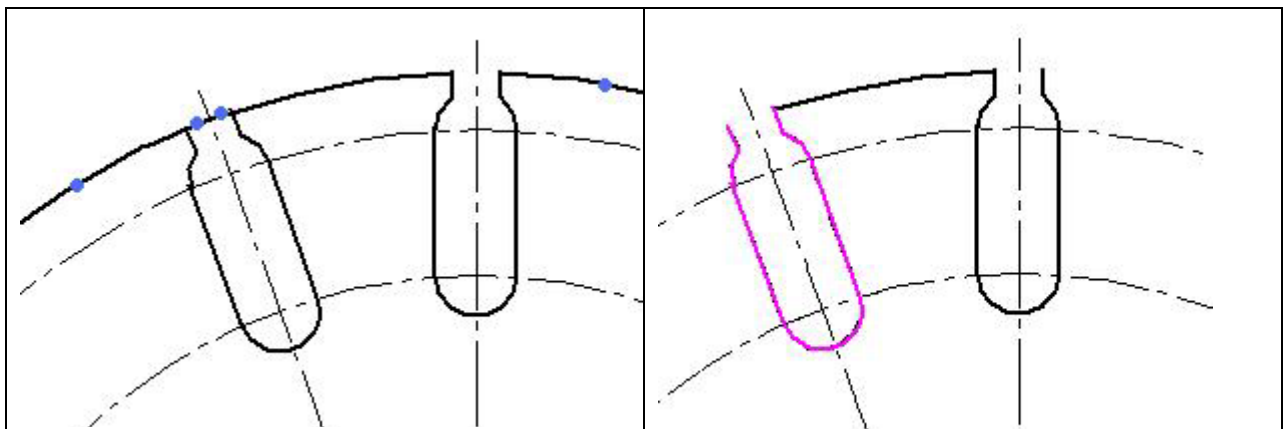

Figure 5.46 – Trimming 12 Diameter Circle　　**Figure 5.47 – Selecting 2ⁿᵈ Slot**

With the initial slotted whole and the short arc that is attached to the left side, it is now time to complete the circular problem and copy this object 18 times in a circular pattern.

5-18

Using the Circular Pattern Command to Create More Arrays

Begin by selecting the slot and the leftover arc that is to its left (Figure 5.48). Next, select the **Circular Pattern** command from the **Draw** panel and the Circular Pattern Window will appear on the left of the screen. Also, pick the endpoint of one of the centerlines as the center of the pattern. In the window, type "**18**" in the Count box and type "**360**" in the Angle box (Figure 5.49). When you finish entering the proper numbers into the boxes, hit the Finish button and your circular pattern will be complete.
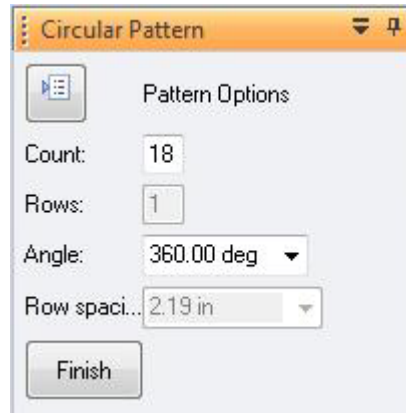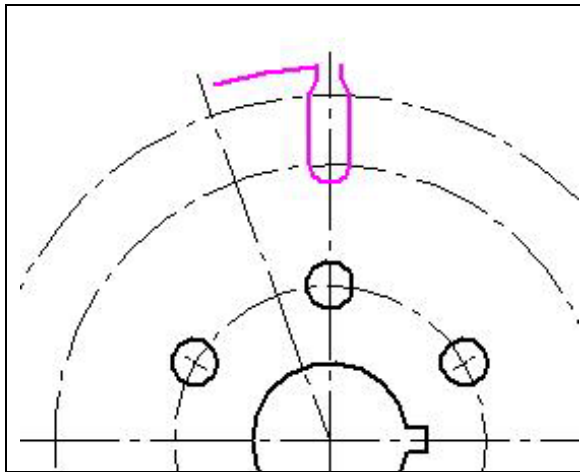
Figure 5.48 – Selecting Objects for Pattern   Figure 5.49 – Circular Pattern Window

Once you have finished the final circular pattern, remember to ungroup the arrayed entities. For now, you will only have to do it once. If your drawing appears like the one in Figure 5.50, then you have completed the circular problem.
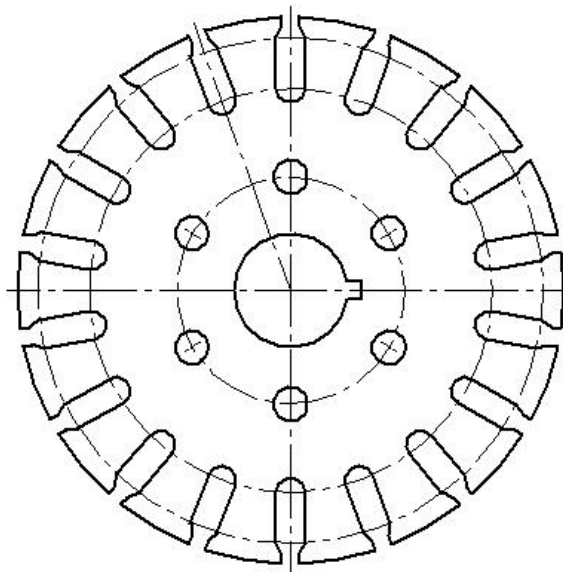
Figure 5.50 – Finished Circular Problem

* World Class CAD Challenge 22-13 * (See End of Chapter)

World Class CAD Challenges for Chapter 5

(22-10) Create a New file, create the three new layers, set the IntelliSketch icons and draw the seven circles in 120 seconds. Continue this drill four times, each time completing the drawing in less than 120 seconds to maintain your World Class ranking.

(22-11) Create a New file, create the three new layers, set the IntelliSketch icons, draw the seven circles and move and copy the two circles in 140 seconds. Continue this drill four times, each tie completing the drawing in less than 140 seconds to maintain your World Class ranking.

(22-12) Create a New file, create the three new layers, set the IntelliSketch icons, draw the seven circles, move and copy the two circles and add lines for the keyway and slotted hole in 190 seconds. Continue this drill four times, each time completing the drawing in less than 190 seconds to maintain your World Class ranking.

(22-13) Create a New file, create the three new layers, set the IntelliSketch icons, draw the seven circles, move and copy the two circles, add lines for the keyway and slotted holes and do the four arrays in 5 minutes. Continue this drill four times, each time completing the drawing in less than 5 minutes to maintain your World Class ranking.