C h a p t e r

5

The Circular Problem

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

- **Using the Object Snap Settings to set the Center Osnap**
- **Using a series of Circle commands, both diameters and radius**
- **Using the Zoom Extents command to display the entire drawing**
- **Using the Offset command to draw a circle**
- **Adding centerlines using the Line command**
- **Using the Move Command to Place Circles in Appropriate Position**
- **Using the Copy Command to Add a Circle in an Proper Position**
- **Using the Line Command to Add Details**
- **Using the Move Command to Add Details**
- **Using the Offset Command to Add Details**
- **Using the Fillet Command to Join Two Non-intersecting Lines**
- **Using the Trim Command to Convert Construction Lines**
- **Using the Layer Control Drop Box to Change an Entity's Layer**
- **Using the Array Command to Create Circular Patterns**
- **Using the Erase Command to Remove Lines**

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 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 allows you to attain the World Class standard.

Drawing circles using Object Snaps

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Setting Object Snaps allows you to select points precisely. You can set your object snaps by right-clicking on the Object Snaps tool on the bottom of the page and clicking Settings, which will bring up the Drafting Settings window and the Object Snap tab. You can also right click on the Object Snap tool and click on the different Object Snap icons in the menu that pops up to toggle them on and off without bringing up the Drafting Settings window. In the circular problem, you need to select the Object Snap Mode Center for grabbing the center point of a circle, recognized by an orange circle when selected (Figure 5.1).
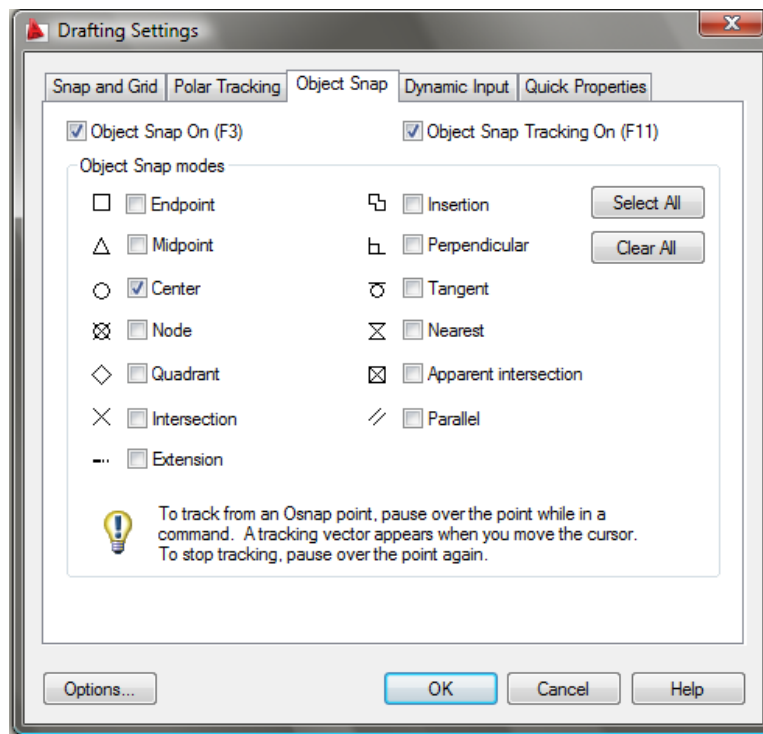

Figure 5.1 – Setting the center Osnap

In the Circular problem, 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 initially placed on the correct layer, especially when there are arrays involved.

Creating Layers in an AutoCAD Drawing

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You are already familiar with the concepts of layers from when you created layers in the Rectangular problem. Remember, you will draw the object lines on layer 0, the AutoCAD default layer, which is automatically set when a new file begins. You can see the current layer listed in the Layer Control list box in the Layers panel under the Home tab (Figure 5.2). To create new layers using the Layers Properties Manager, select the Layer Properties button from the Layers panel.
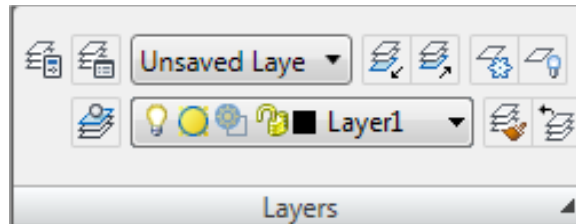
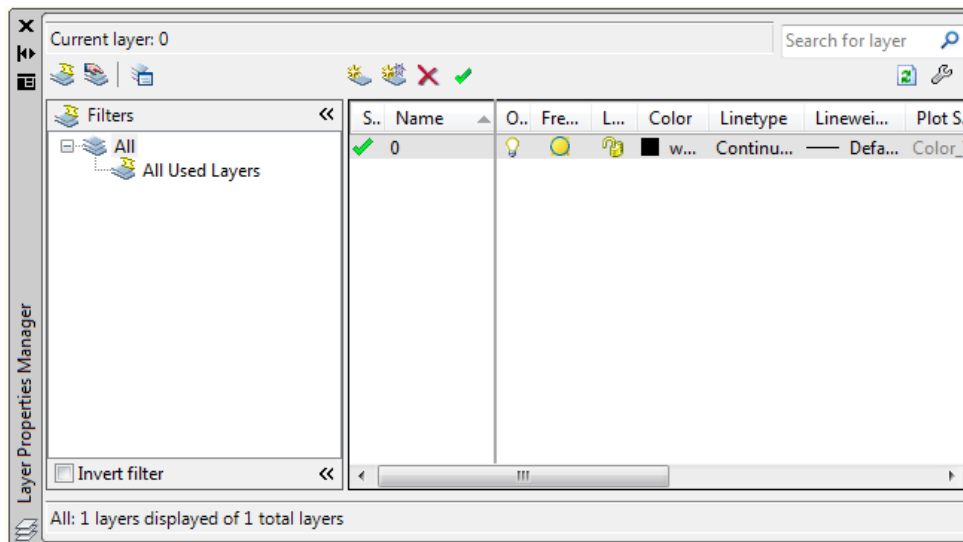**Figure 5.2 - The Layer Panel**

Figure 5.3 - The Layer Properties Manager window

To create this drawing, you need make three new layers: the Dimension layer for dimensions, the Center layer for centerlines, and the Text layer for text and notes. To create the layers, select the New button to add a new layer under the 0 layer. Immediately, the layer name is ready for editing, so type "Dimension" and hit ENTER twice to add your next new layer. Type "Center" and hit ENTER twice to add another new layer. Type "Text" and ENTER once to finish adding new layers. Next, you need to assign the layers different colors to help distinguish between them. To change the Dimension layer color, place your mouse arrow over the color box in the Dimension layer row and single left click with your mouse. The Select Color window will appear in front of the Layers Properties Manager (Figure 5.4). Select red from the Standard Colors row and select the OK button. You will now see that Dimension layer is red. Repeat the process to make the Center layer green and the Text layer blue.
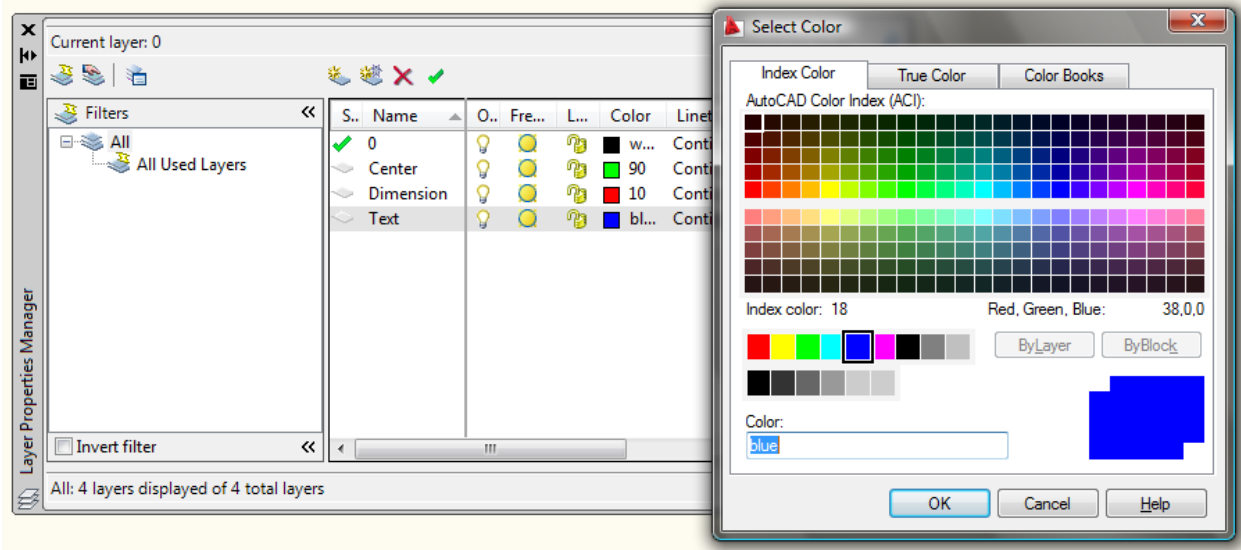
Figure 5.4 - The Select Color window

To finish creating and modifying layers, you need to change the Continuous linetype to a Center linetype for your Center layer. Place your mouse arrow over the text "Continuous" in the Center layer row and single left click with your mouse. The Select Linetype window will appear in front of the Layers Properties Manager (Figure 5.5). Note that only the Continuous Linetype is loaded, so select the Load button and the Load or Reload Linetypes window will appear. Scroll down, highlight the CENTER linetype as shown in Figure 5.6, and select the OK button. Highlight the CENTER linetype in the Select Linetype window and select OK. You will now see that the Center layer has a CENTER linetype.
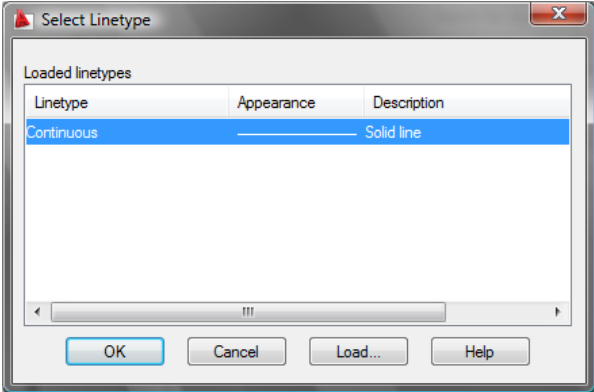

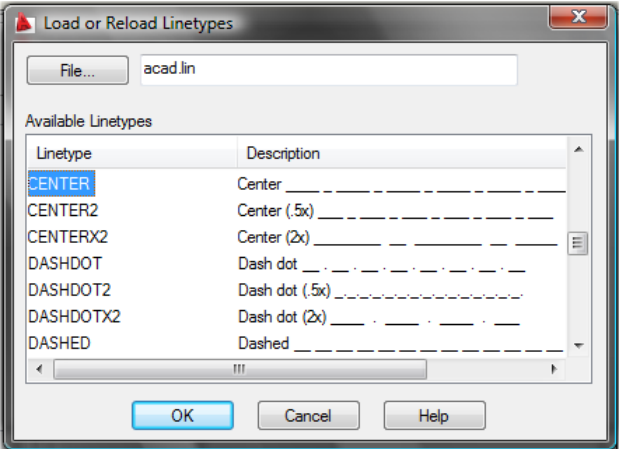**Figure 5.5 - The Select Linetype window**

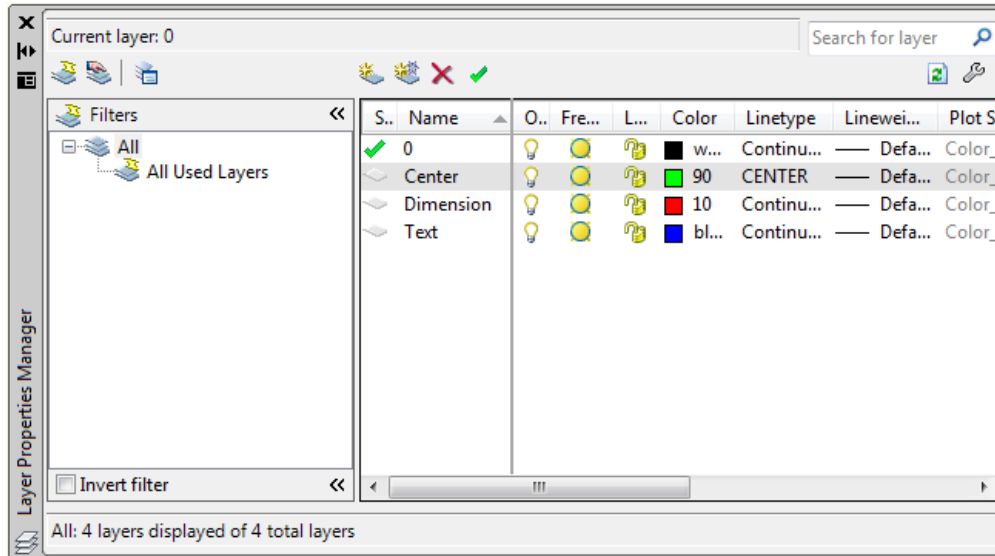**Figure 5.6 - Load or Reload Linetypes**

Figure 5.7 - Load or Reload Linetypes

The strategy in doing the Circular problem is to draw the perimeter circle first to define the outside of the drawing, then to draw in the other circles, then place in the centerlines, and move all circles in their proper position. You will finish the drawing by adding the minor details such as slots and keyways. Circular drawings that have repeating patterns will be constructed with arrays. In this problem, you will use three arrays to efficiently lay out object and centerlines. As in the Rectangular problem, you will reinforce the lessons of dimensioning and placing notes on the drawing. This drawing is also a timed problem, where the time to use the tools will represent your understanding and ability to operate the individual functions.

Drawing circles using Object Snaps
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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 one circle with the radius of 4.4375. Finally, you will draw the last circle using the Offset command, which you will do in the next section of this problem.

To draw the first five circles, select the Circle command on the Draw panel under the Home tab and pick a center point randomly on the center of your display with the left mouse button. You need to type "d" for diameter when the command line prompts you to "specify radius of circle or [Diameter]". Then type "12" and ENTER when the command line prompts you to "specify diameter of circle" (Figure 5.8). A 12-unit circle will appear on the display (Figure 5.9).

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <1.0000>: d
Specify diameter of circle <2.0000>: 12

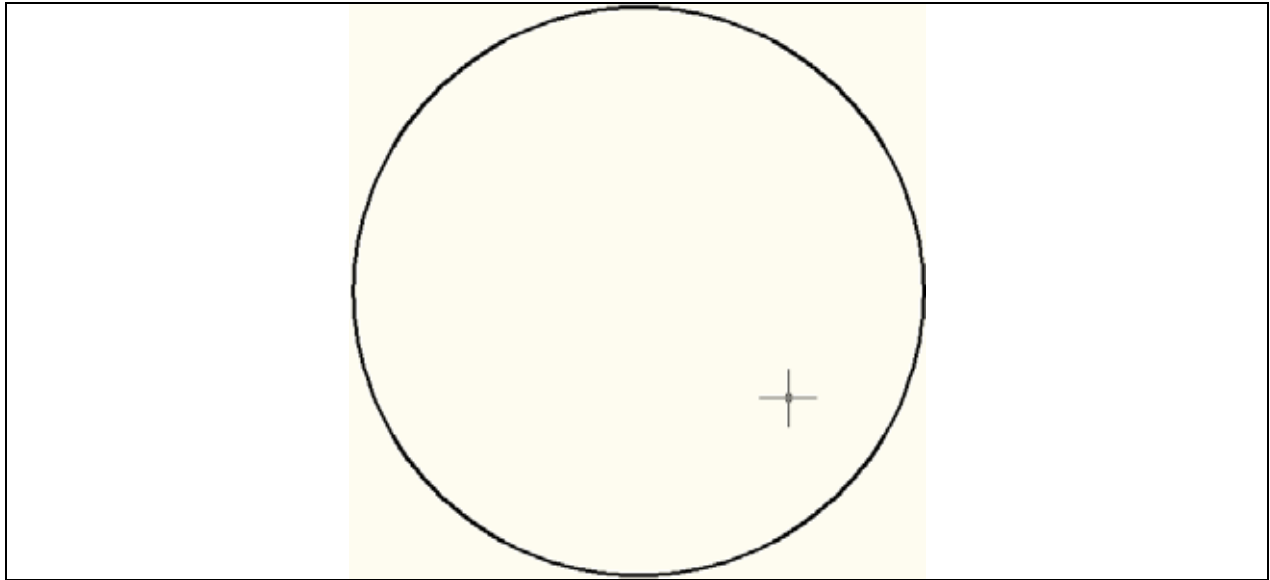**Figure 5.8 – Command line prompts for the 12 unit circle**

Figure 5.9 – The 12 unit circle which is the outside perimeter of the Circular problem

Using the Zoom Extents Command to Display the Entire Drawing
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The 12-unit circle may not fit completely in the display, so you should use one of the zooming functions to fix the problem. Select the Zoom Extents tool from the Utilities panel under the Home tab. You may have to expand the zoom menu by clicking on the name of the current zoom tool. Select the Extents from the drop-down menu tool (Figure 5.10) and the entire 12-unit circle will appear in the graphical display.
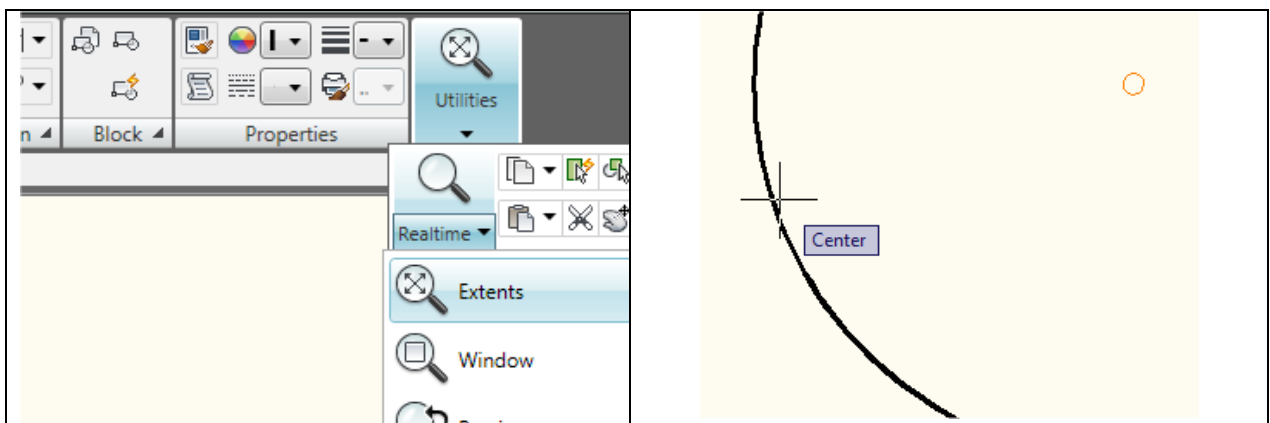

Figure 5.10 – Utilities Panel   **Figure 5.11 – Using the Center Osnap**

Adding more Circles using the Diameter Option of the Circle Command

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Next, select the Circle command on the Draw panel under the Home tab. Place the aperture cursor over the 12-unit circle object line and a Center Osnap symbol (orange circle) will appear in the middle of the first circle drawn (Figure 5.11). Select the center point of the second circle by selecting the center with the left mouse button. You need to type "d" for diameter when the command line prompts you to "specify radius of circle or [Diameter]". Then type "5" and ENTER when the command line prompts you to "specify diameter of circle". Repeat the Circle command by hitting ENTER, and place the 2.5, 0.75, and 0.625 circles inside the 12-diameter circle sharing the same center points. There will be five concentric circles as shown in Figure 5.12.

You have been typing "d" for the diameter option, so 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. Some CAD operators will add the Circle Center Diameter tool, which is covered in World Class Advanced CAD. Many Architects and Engineers will create mark up drawings with diameters, because diameters conform 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.
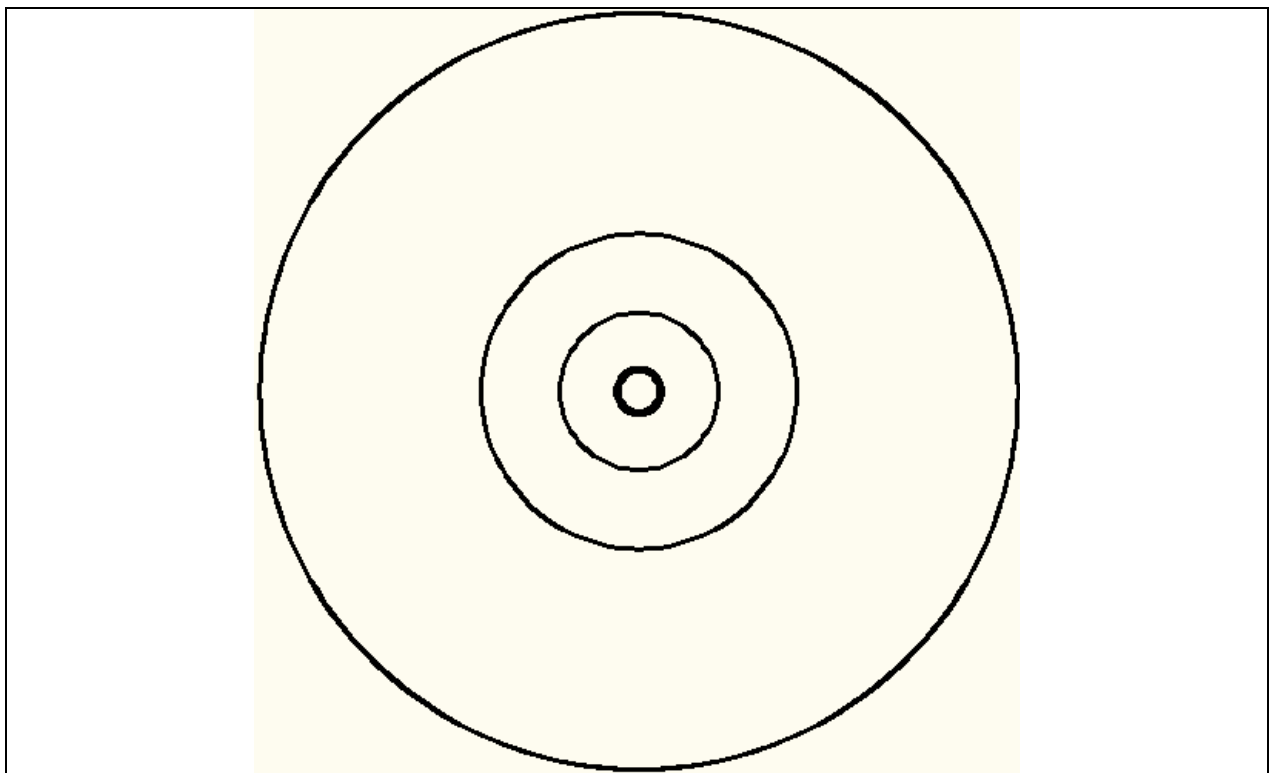

Figure 5.12 – Placing the 5, 2.5, 0.75 and 0.625 diameter circles in the drawing

Adding a Circle using the Radius Option of the Circle Command
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You need to add a sixth concentric circle by pressing ENTER to repeat the Circle command. Select the center point of this circle by placing the cursor over the 12-diameter circle and picking the same center point with the left mouse button. When the command line prompts you to "specify radius of circle or [Diameter]", type "4.4375" and ENTER (Figure 5.13). A new large circle will be drawn between the 5 and 12-diameter circle.

Command: _circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <13.0000>: 4.4375

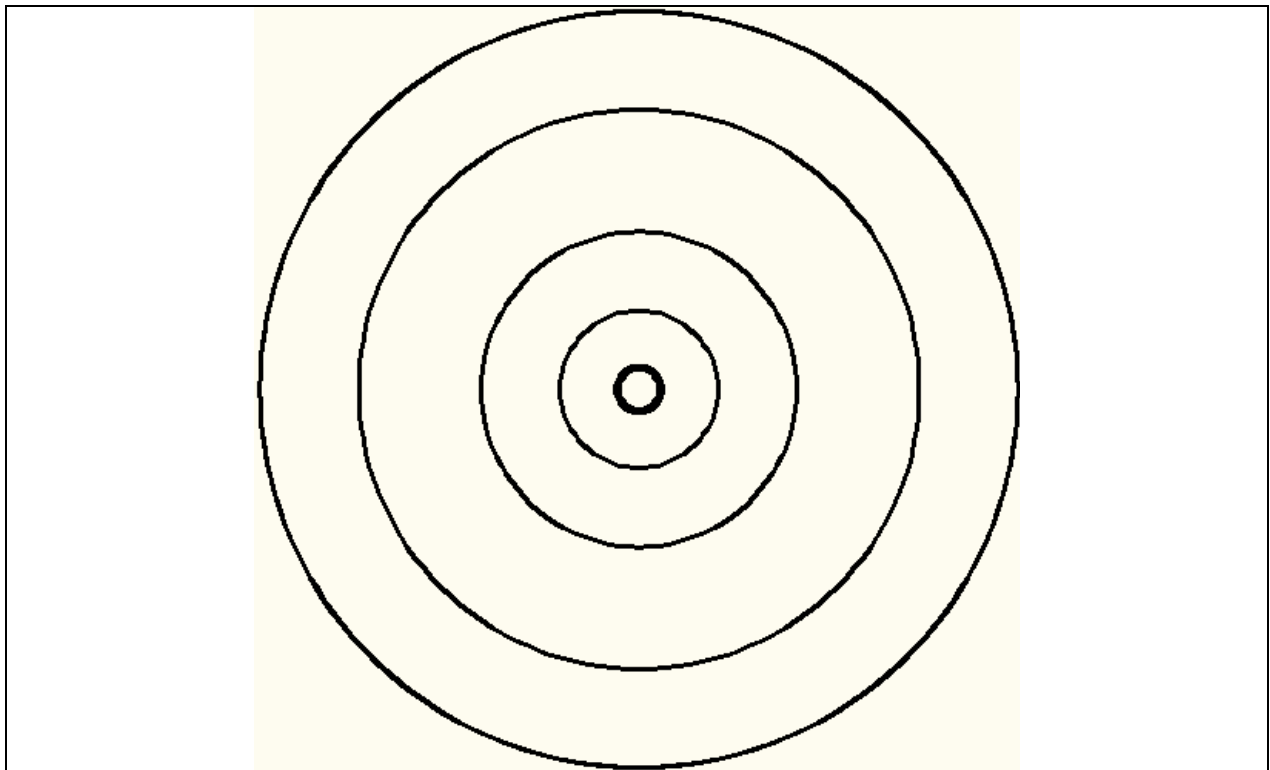**Figure 5.13 – Command line prompts for the 4.4375 radius circle**

Figure 5.14 – The 4.4375 radius circle

Using the Offset Command to Add Another Circle
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To add the seventh circle, you are not going to use the Circle command, but you will use the Offset command on the Modify panel. Select the Offset tool and then specify the offset distance by typing 1.125 and ENTER at the command line. Next, select the 4.4375 radius circle you just drew. 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. A new circle will be added to your drawing as shown in Figure 5.15.

From the time you opened the new drawing until the seven circles are drawn, the timer has been running, so check your time by typing "Time" at the command line. Check your time against the World Class CAD standard and train yourself to draw this section of the Circular problem to time and with zero errors.
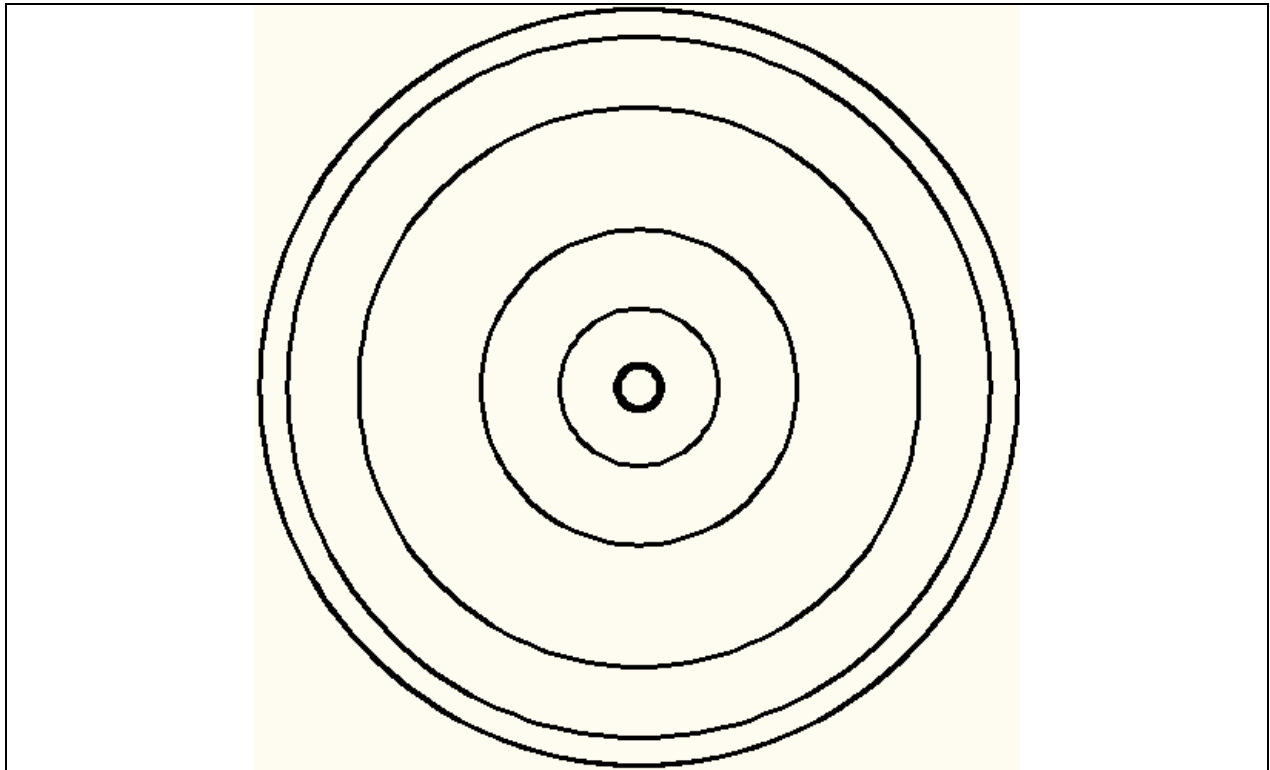
Figure 5.15 – Using the Offset command to draw the seventh circle

* World Class CAD Challenge 02-10 * - Create a New file, create the three new layers, set the Object Snap settings and draw the seven circles in 120 seconds. Continue this drill four times, each time completing the drawing under 120 seconds to maintain your World Class ranking.

Adding Centerlines to the Drawing Using the Line Command

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 were following the strategy utilized in the Rectangular problem. Since the polar or circular array will be employed in this construction technique, you will only need to draw a single quadrant centerline, which will then be copied and rotated using the Array command on the Modify panel.

First, zoom in closer to your work by using the Zoom Window command on the Utilities panel. Select the Zoom Window tool in the same manner you selected the Zoom Extents tool earlier. To zoom, specify the first corner and the opposite corner, picking two points that will make a window similar to the one shown in Figure 5.16.
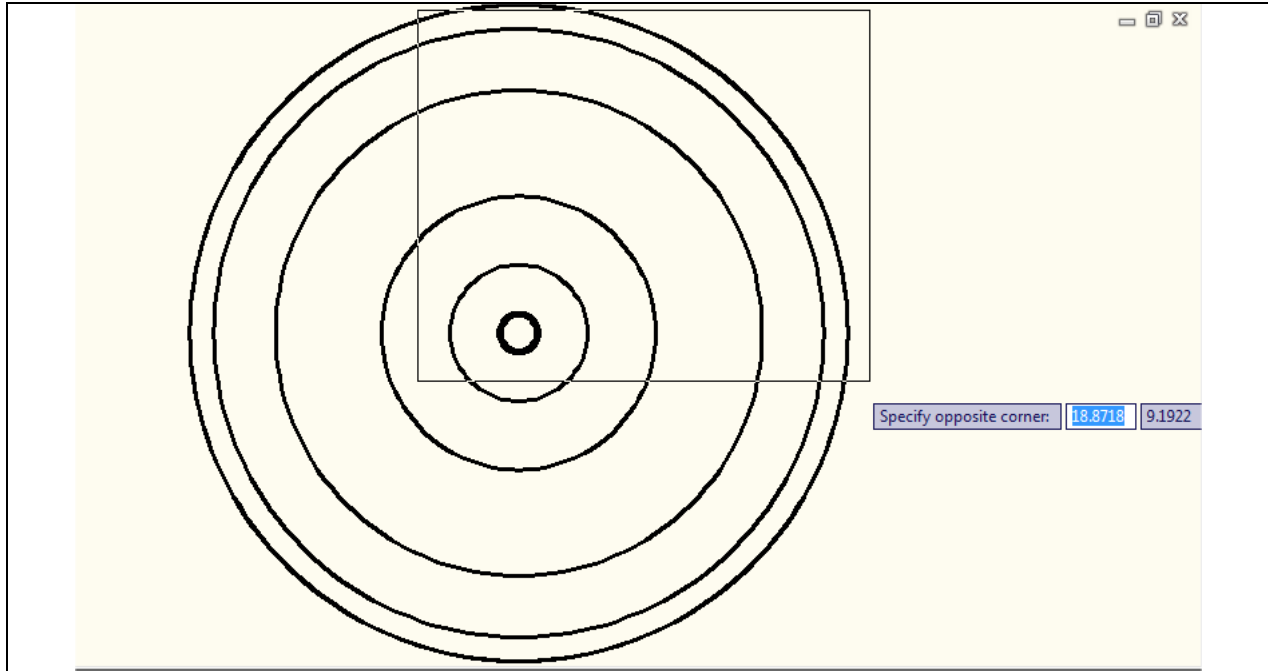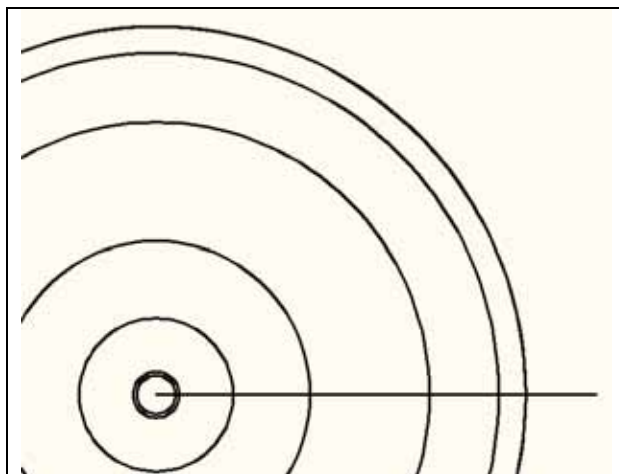

Specify opposite corner: 18.8718 9.1922

Figure 5.16 – Using the Zoom Window command to get in close to your work

Select the Line command on the Draw panel. Select the first point of the line by placing the mouse cursor on top of any circle, thereby causing the Center Osnap to appear, and then select the center with the left mouse button. The second point will be located roughly 0.25 units past the outside circle. Ensure that the Ortho setting is on, enabling lines to only be drawn in horizontal or vertical mode. The center line should look like the one shown in Figure 5.17.

Figure 5.17 – The horizontal centerline

Expert operators trained in the World Class CAD techniques will change the OSNAP settings from time to time in a drawing to save time. Select the Object Snap Setting tool by right-clicking on the Object Snaps tool on the bottom of the screen and selecting Settings from the menu. Press the "Clear All" button on the Drafting Settings window and then check the Endpoint, Midpoint, Quadrant, Intersection and Perpendicular Object Snaps. Then select "OK" to confirm the changes. (Figure 5.18)
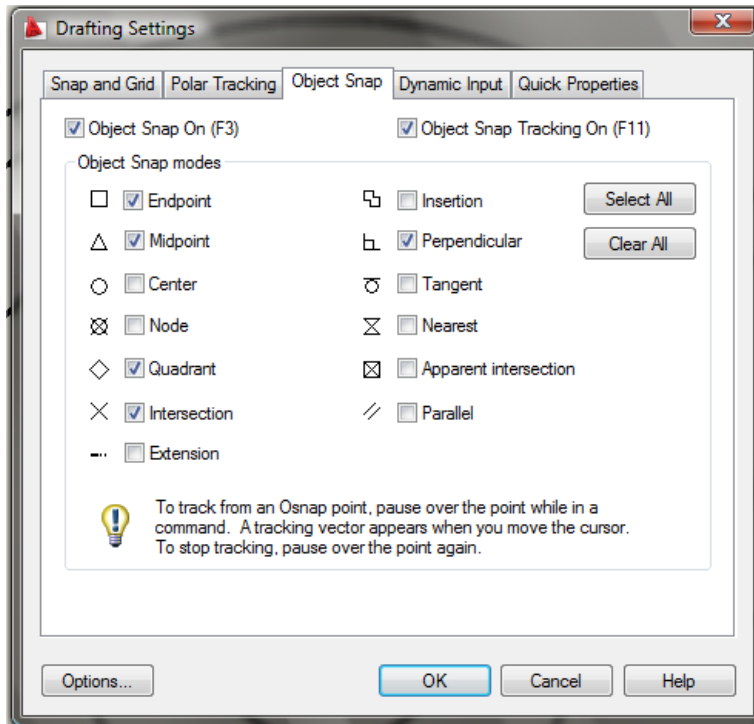
**Figure 5.18 – Changing the Object Snap settings to maintain drawing efficiency**

Using the Move Command to Place Circles in Appropriate Position
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The 0.625 and 0.75 circles need to be moved into their appropriate position using the Move command. After selecting the Move tool from the Modify panel, select the 0.75 circle. You will be prompted to select other objects, so press ENTER to proceed to the second part of the Move command.
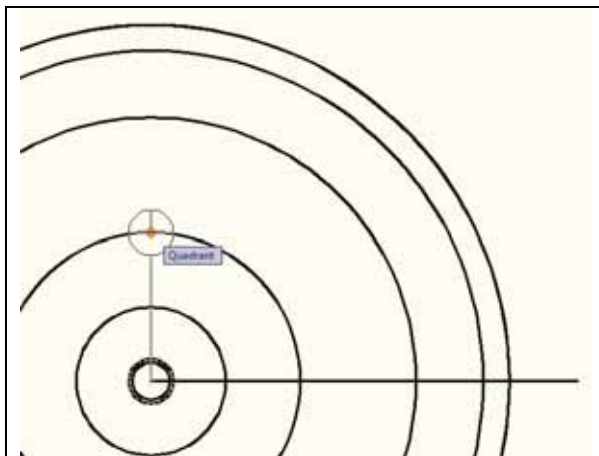
At the prompt, "specify base point or displacement", select the left endpoint of the centerline, which represents the center of the part. The "second point of displacement" is the north quadrant of the 2.5-unit circle. After picking the second point, notice that the 0.75 circle is now centered on the north quadrant of the 2.5-unit circle (Figure 5.19).

Figure 5.19 – Moving the 0.75 circle
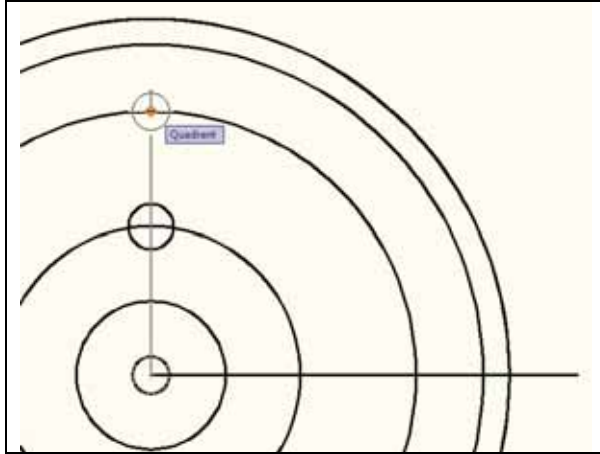
You need to move the 0.625 circle to the north quadrant of the 4.4375 radius circle using the same technique: select the 0.625 circle, ENTER, select the Endpoint of the centerline, and then the north quadrant of the 4.4375 radius circle. Now the 0.625 circle has moved to the north quadrant, as shown in figure 5.20.

Figure 5.20 – Moving the 0.625 circle

Using the Copy Command to Add a Circle in an Proper Position
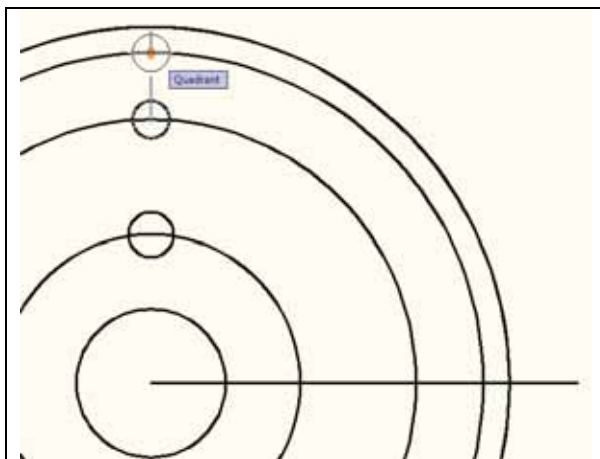
To create a slotted hole in this part, you need to copy the 0.625 circle to the north quadrant of the next circle. Select the Copy tool on the Modify panel. After selecting the circle, press ENTER to proceed to the second stage of the Copy command. For the first base point, pick the quadrant at the center of 0.625 circle, and for the second base point of displacement pick the quadrant of the circle that is 1.125 to the outside, as shown in Figure 5.21.

Figure 5.21 – Copying the 0.625 circle

* World Class CAD Challenge 02-11 * - Create a New file, create the three new layers, set the Object Snap settings and draw the seven circles, move and copy the two circles in 140 seconds. Continue this drill four times, each time completing the drawing under 140 seconds to maintain your World Class ranking.

Using the Line Command to Add Details

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 centerline for the 0.75 circle. In the Rectangular Problem, you used the center mark provided on the Dimension panel. The center marks have two lines perpendicular to each other and are appropriate in rectangular, but in circular parts, center marks are a combination of straight lines intersecting the

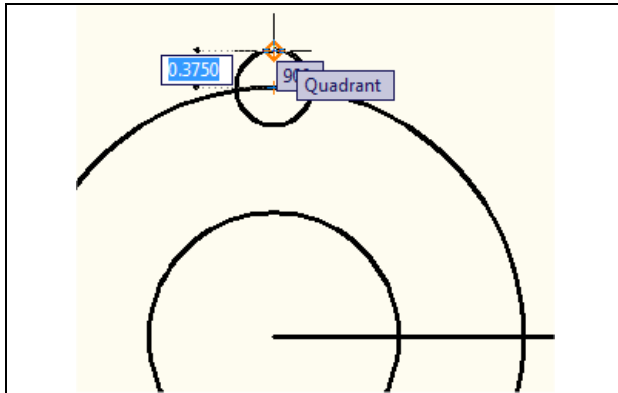circular part's center and arced segments defining the distance from the part's center.

Figure 5.22 – Drawing a center mark

Draw a line segment by picking the Line tool on the Draw panel. Pick the north quadrant of the 5.0 diameter circle. Draw a line to the north quadrant of the 0.75 circle as shown in Figure 5.22. Press ENTER to finish the line segment.
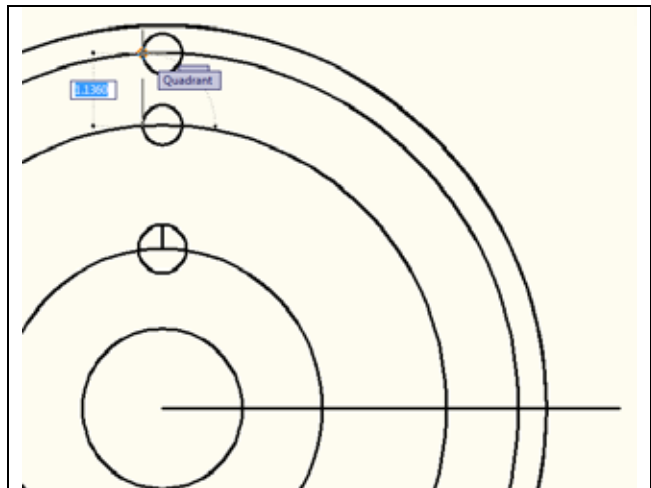
To continue forming the slotted hole, you need to draw lines from the tangent points of each circle. In most cases, the Quadrant Osnap tool will remain a primary tool to draw slotted details. Draw a line segment by picking the Line tool on the Draw panel and picking the west quadrant of the lower 0.625 circle. Draw a line to the west quadrant of the upper 0.625 circle as shown in Figure 5.23. Hit ENTER to finish the line segment.
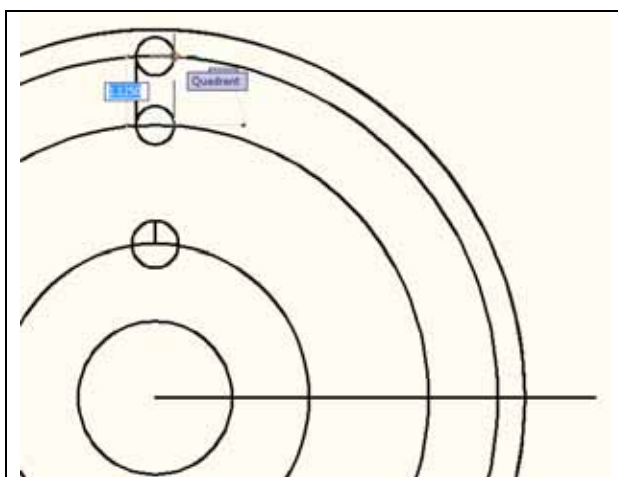

Figure 5.23 – Drawing the first slot line

Figure 5.24 – Drawing the second slot line

To add the second line, you need to draw a line segment by picking the Line tool on the Draw panel and picking the east quadrant of the lower 0.625 circle. Draw a line to the east quadrant of the upper 0.625 circle as shown in Figure 5.24. When practicing placing lines on the Circular problem, you can press ENTER to repeat the previous command such as Line instead of selecting the Line tool on the Draw panel.

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 5.25, 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 for the first point and any point above the first.
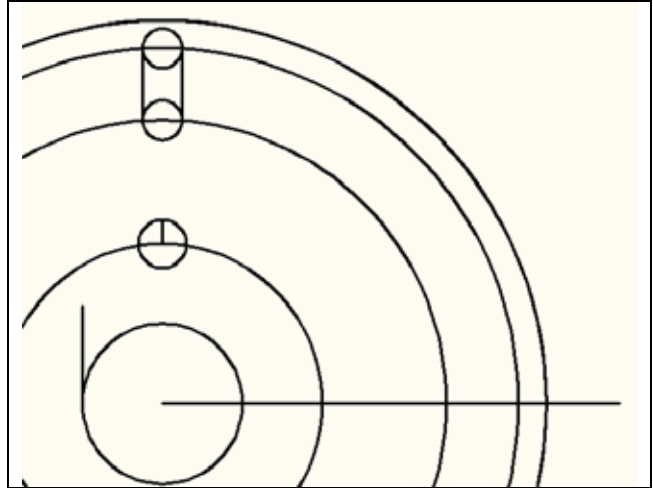

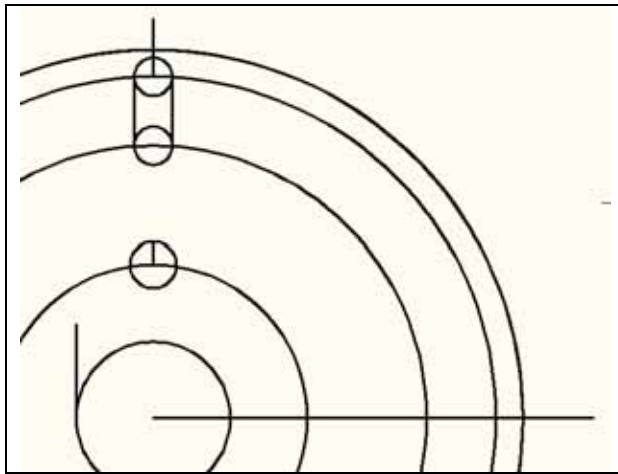**Figure 5.25 – Draw the keyway line**

Figure 5.26 – Drawing a construction line

The top of the slotted hole at the near the north quadrant of the circular part will be open, so you will draw a centerline to construct the detail. Select the Line tool on the Draw panel, picking the center of the top 0.625 circle for the first point. For the second point of the line, pick any point above the perimeter of the outermost circle. (Figure 5.26)

Using the Move Command to Add Details
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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 Modify panel, and select the keyway line and ENTER to proceed to the second stage of the Move command. Next, you will be prompted to select the base point or displacement. Pick a point in any open area on the viewing screen where your Object Snaps will not interfere with the command. After that, move the line to the right with your mouse to orient the direction of the placement and type 2.687 and ENTER. The keyway line is now in the appropriate location. (Figure 5.27)
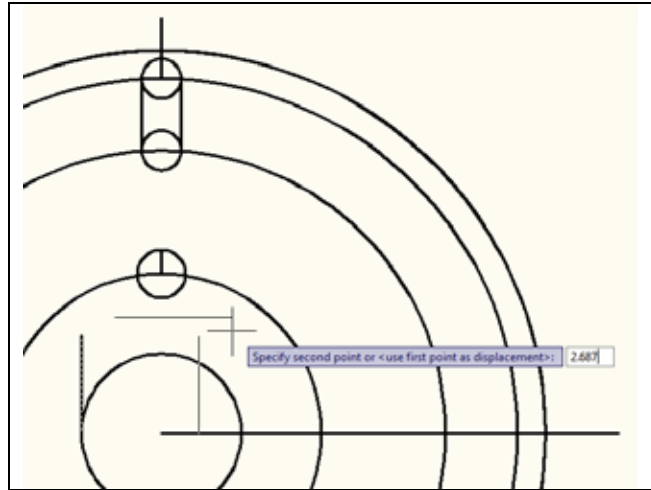

Figure 5.27 – Moving the keyway line

You need to reposition the centerline drawn on the 0.75 circle by selecting the Move tool on the Modify panel and ENTER. You will be prompted to select the base point or displacement. Pick the midpoint of the centerline using the Midpoint Object Snap that is automatically loaded in the Object Snap Settings window. After that, move the line down with your mouse so that it aligns with the center of the 0.75 circle and left-click. The centerline is now in the proper location. (Figure 5.28)
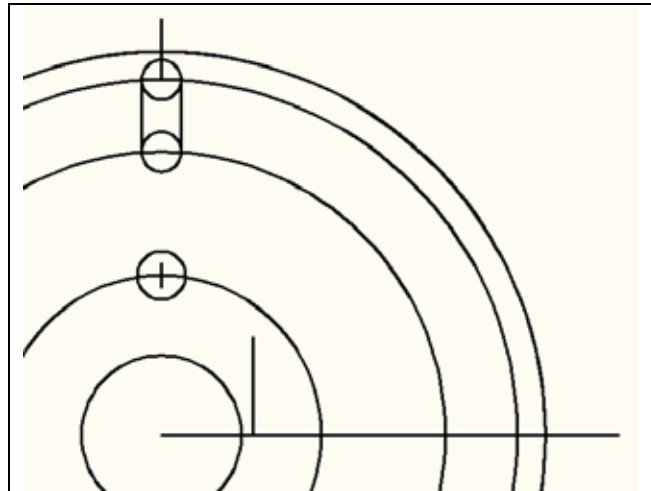

Figure 5.28 – Centering the center mark

Using the Offset Command to Add More Detail
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When your initial lines are drawn accurately, 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 Modify panel and type 0.1875 for the offset distance and ENTER. Next select the keyway centerline and then pick above it to copy a line 0.1875 above the original line. In the same Offset command, select the keyway centerline again and then pick below it to copy a line 0.1875 below the centerline.
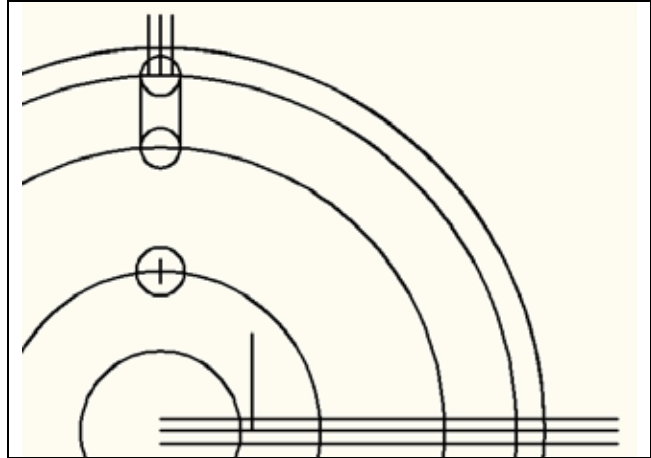

Figure 5.29 – Offsetting lines

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 pick to the left of the centerline to copy a line 0.1875 to the left. Yet again, in the same Offset command, select the slotted hole's centerline and then pick to the right of the centerline to copy a line 0.1875 to the right. (Figure 5.29)

* World Class CAD Challenge 02-12 * - Create a New file, create the three new layers, set the Object Snap settings and draw the seven circles, move and copy the two circles, and add lines for the keyway and slotted holes in 190 seconds. Continue this drill four times, each time completing the drawing under 190 seconds to maintain your World Class ranking.

Using the Fillet Command to Join Two Non-intersecting Lines

The fillet command, when the radius is set to zero, becomes an effective trimming and joining tool. You will form the keyway with the Fillet tool. This is the first time you are using the Fillet command to join two line segments that could intersect, but do not.
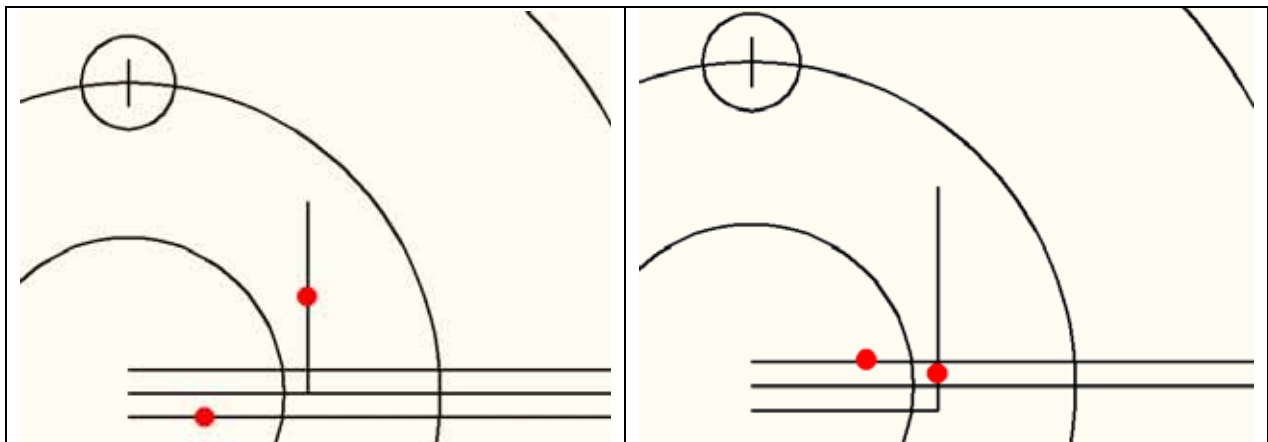

Figure 5.30 – Forming the key by filleting

Figure 5.31 – Forming the key by filleting

In Figures 5.30 and 5.31, zero radius corners are created by filleting the keyway and keyway offset lines. You can see where to select with your pick box by following the red dots in the pictures above. Select the Fillet command on the Modify panel, and type "R" and ENTER to allow you to set the radius. Type "0" and ENTER. Also, hit "M" and ENTER as you will be doing more than one 0 radius fillet. As in Figure 5.30, select the lower offset line and the keyway line to form the first corner. Next, select the top keyway offset line and the keyway line.

Using the Trim Command to Convert Construction Lines

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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.
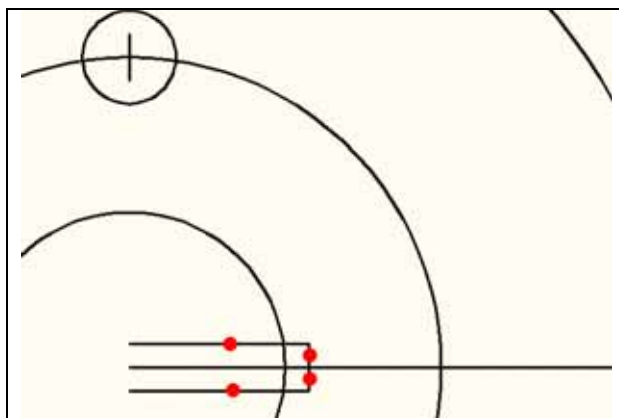
To remove the two line segments entering the 2.5 diameter circle and open the circle to the 0.375 key width, use the Trim tool on the Modify panel. Press ENTER to make every object line a cutting line. Now make four selections, the two-keyway centerline offset lines entering the 2.5 diameter circle, and the two arc segments between the two offset lines. The four red circles on Figure 5.32 show where to select to correctly trim the detail.

Figure 5.32 – Forming the key by filleting

To create the slot shown in Figure 5.34, you need to remove ten line and arc segments. Repeat the Trim tool on the Modify panel. Press ENTER to make every object line a cutting line, and then make the ten selections as shown by the red circles in Figure 5.33.
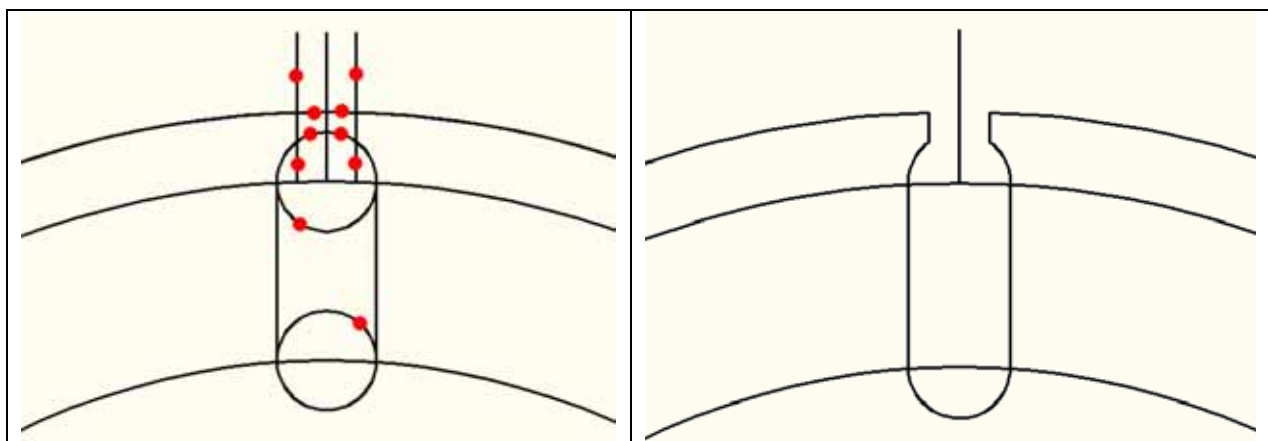

Figure 5.33 – Detailing the slot with Trim   Figure 5.34 – The detailed slot

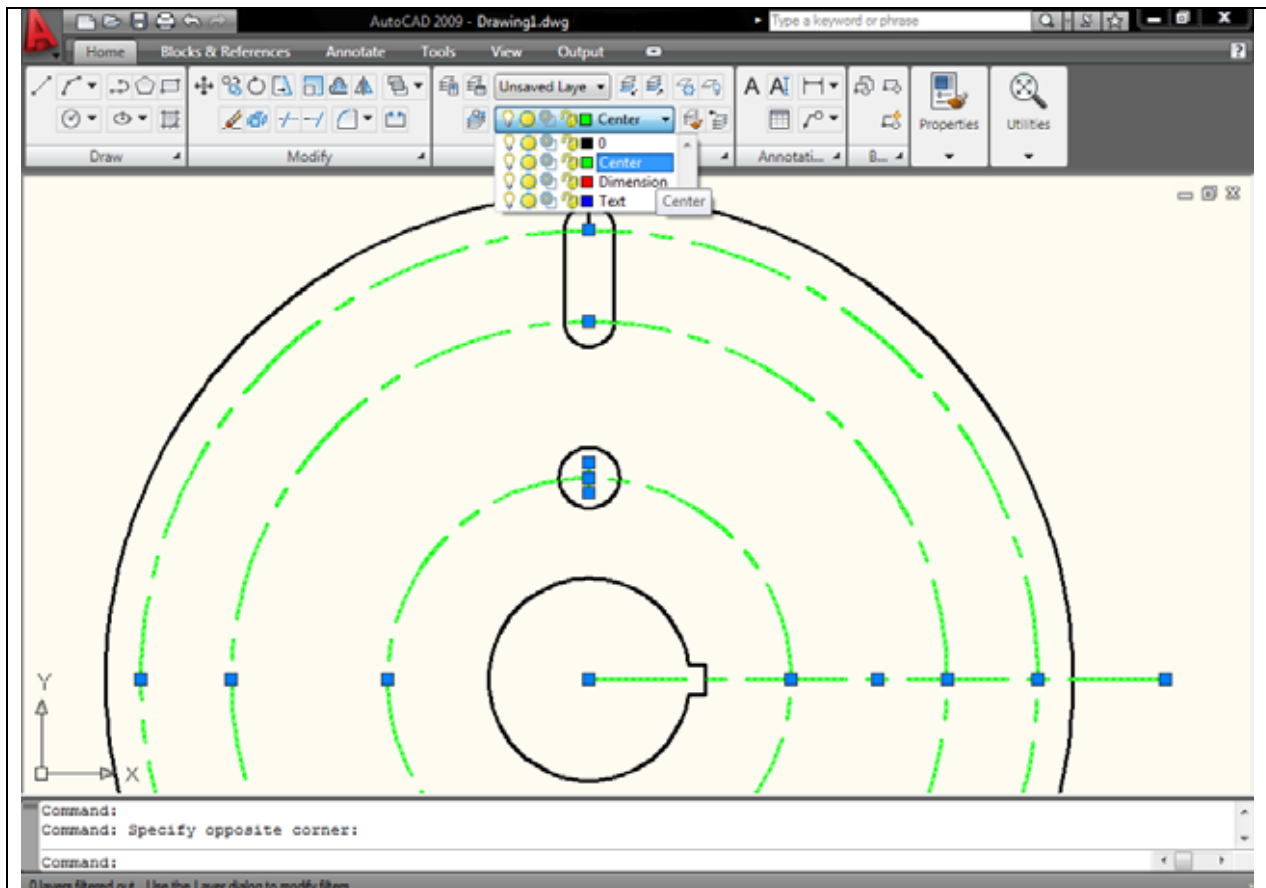# Using the Layer Control Drop Box to Change an Entity's Layer

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Figure 5.35 – Objects place on the Center layer

You may have already noticed that drawing entities, such as lines, arcs and circles, can be selected prior to choosing an AutoCAD tool. In a situation where you need to place certain line and circle entities 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 Layer Control drop box Layer panel that is presently displaying layer "0", select the Center layer. You will notice the selected entities will turn green, which the color that was assigned to the Center layer. Press the "ESC" key to remove the entities from the selection set.

Using the Array Command to Create Circular Patterns

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Changing the entities to the proper layer will save time, since the Array command will create multiple copies of lines and circles many times over. In the Rectangular problem, you were exposed to the rectangular array. Now you will perform four separate polar arrays, the first of which will be the 0.75 diameter circle and centerline six times. After erasing unneeded vertical centerlines, you will conduct a second polar

array of the major centerline. The third array will both rotate and copy the slotted hole and vertical centerline twenty degrees counterclockwise. After erasing superfluous lines and arcs, the fourth polar array will finish a complex circular part containing hundreds of entities. You may choose to Zoom Extents your drawing, so it will be easier to see and select all of the objects that will be arrayed.

Select the Array tool on the Modify panel (you may have to expand the panel to find the Array tool, like you did in the rectangular problem), and the Array window will appear (Figure 5.36). Pick the Polar Array radial button, then the Select Objects button. Select the 0.75 diameter circle and its centerline on the display screen and hit ENTER to return to the Array window. Next, select the Pick Center Point Button, which brings you back to the drawing; select the center of the part.
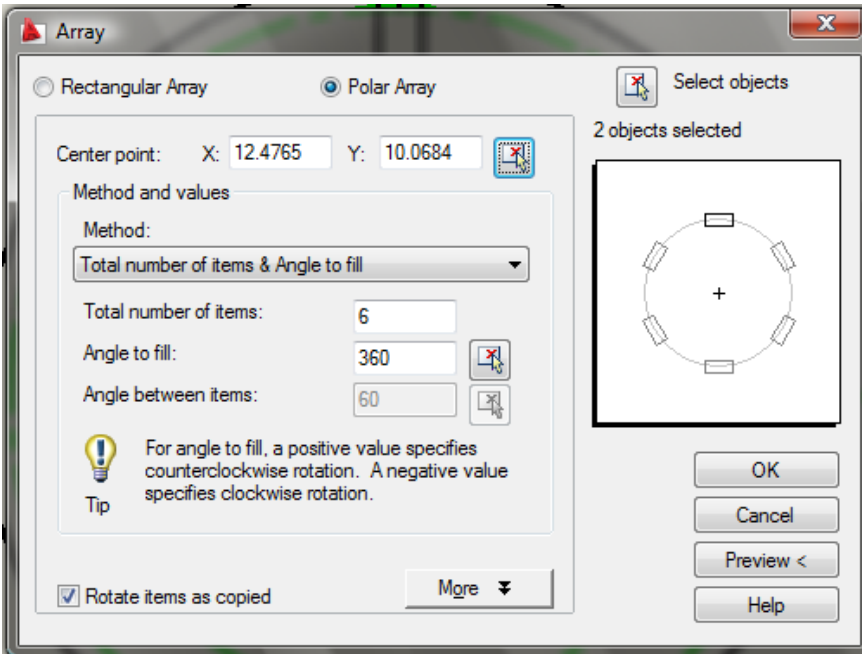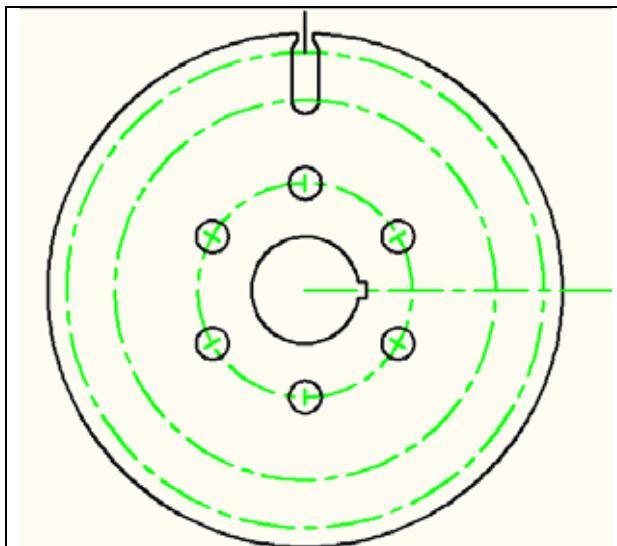
Figure 5.36 – The Array window

Pick the center of the part by selecting the Endpoint of the centerline that is also the center of the part. Change the "total number" textbox to "6" and select the Preview button. The array will be shown on the display screen, along with an Array Accept, Modify or Cancel window. If the Circular problem looks like Figure 5.37, press the Accept button and the array will be completed. If the Circular problem does not look like Figure 5.37, press the Modify button repeat the array process.

Figure 5.37 – The polar array of the 0.75 circle and centerline

# Using the Erase Command to Remove Three Lines

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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 on object or dimension line can cause problems. You have arrived at a moment when it is necessary to erase the two centerlines and a construction line.
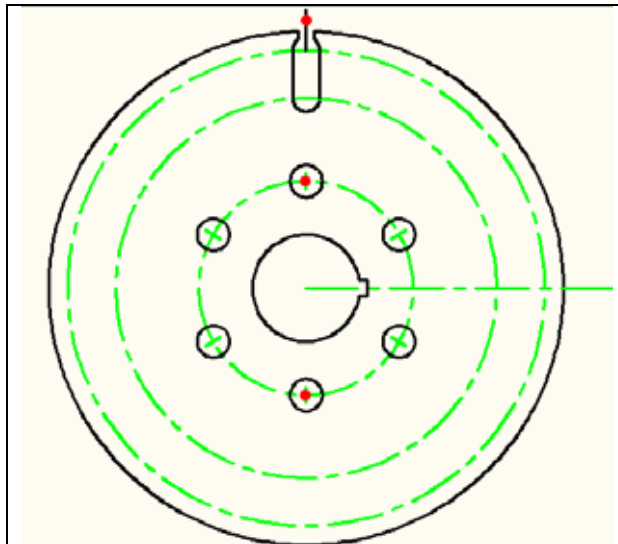
Select the Erase tool on the Modify panel, and select the three entities on the part as shown in Figure 5.38. This will remove any smaller centerlines that would be covered up by the main centerline. Next, you need to perform the second of the four arrays, copying and rotating the main centerline around the circular part.

Figure 5.38 – Erasing three lines

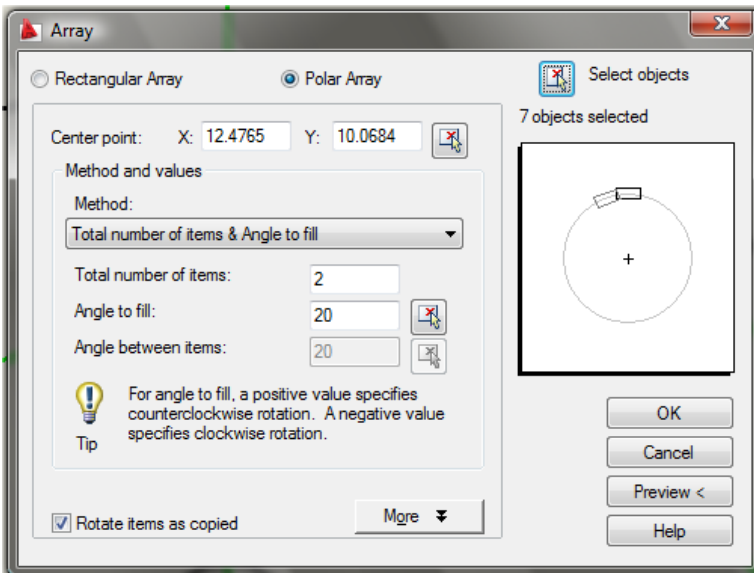## Using the Array Command to Create More Circular Patterns

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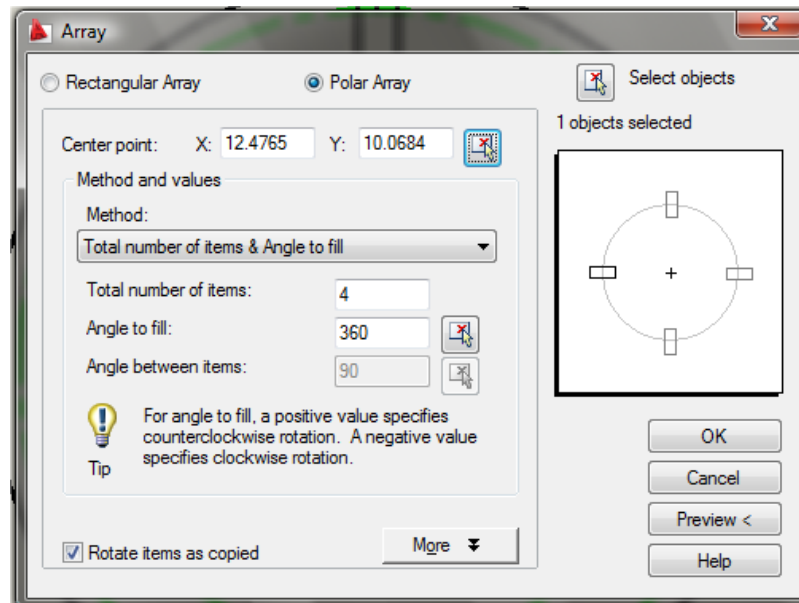
Select the Array tool on the Modify panel to access the Array window (Figure 5.36). Pick the Polar Array radial button, and then the Select Objects button. Select the main centerline on the display screen and hit ENTER to return to the Array window. Next, select the Pick Center Point Button, which will bring you back to the drawing to select the center of the part.

Figure 5.39 – The Array window
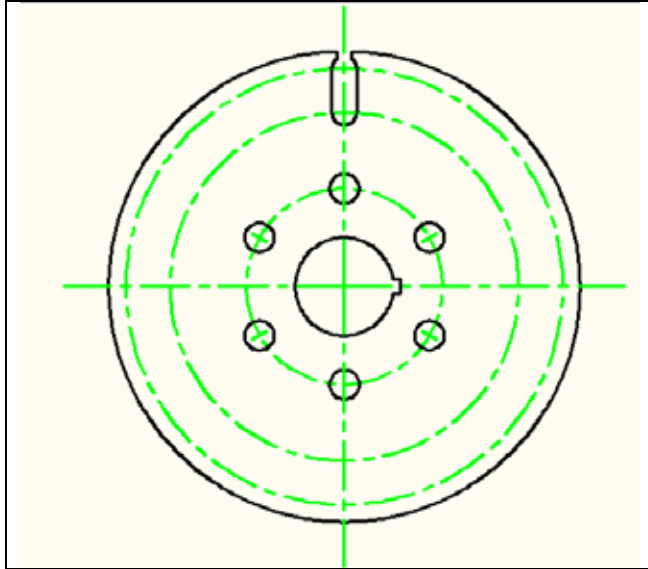
Figure 5.40 – Polar array of the main centerline

Pick the center of the part by selecting the endpoint of the centerline that is also the center of the part. Change the "total number textbox" to "4" and select the Preview button. The array will be shown on the display screen. If the Circular problem looks like Figure 5.40, right-click and the array will be completed. If the Circular problem does not look like Figure 5.40, left-click to return to the Array window and correct the settings.

Figure 5.41 – The Array window

Select the Array tool again on the Modify panel and the Array window will appear (Figure 5.41). Pick the Polar radial button, then the Select Objects button. Select the slotted hole using a selection window (Figure 5.42) on the display screen, and then top main centerline. Hit ENTER to return to the Array window.
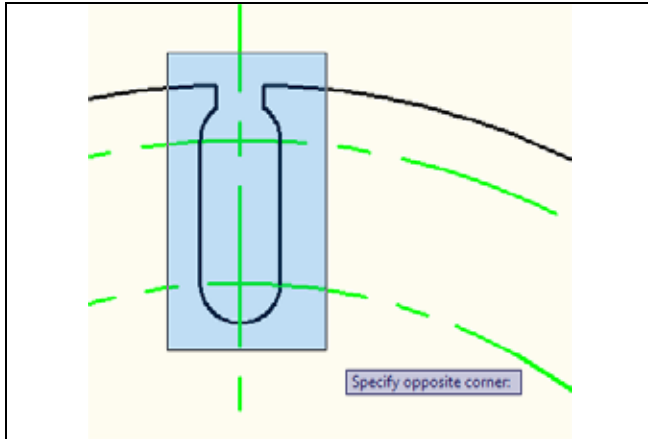
Figure 5.42 – Selecting the slot

Next, select the Pick Center Point Button, which will bring you back to the drawing to select the center of the part. Pick the center of the part by selecting the endpoint of the centerline that is also the center of the part. Change the total number textbox to "2" and the angle to fill textbox to 20 degrees. Select the Preview button.
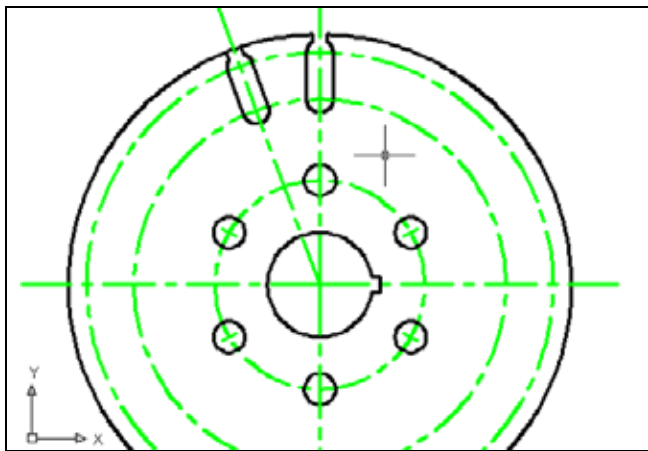

Figure 5.43 – The array of the slotted hole

The array will be shown on the display screen. If the Circular problem looks like Figure 5.43, right-click to accept the Array. If the Circular problem does not look like Figure 5.43, left-click to return to the Array window and correct the settings.

Using the Trim and Erase Command to Remove Arcs and Lines

Select the Trim tool on the Modify panel and select the right line on the arrayed slotted hole using a selection window as shown in Figure 5.44. Press ENTER and select the outside 12.0 diameter circle. The circle will be trimmed back to the right line of the slotted hole. Select the Erase tool on the Modify panel and remove the arrayed slotted hole as shown in Figure 5.45.
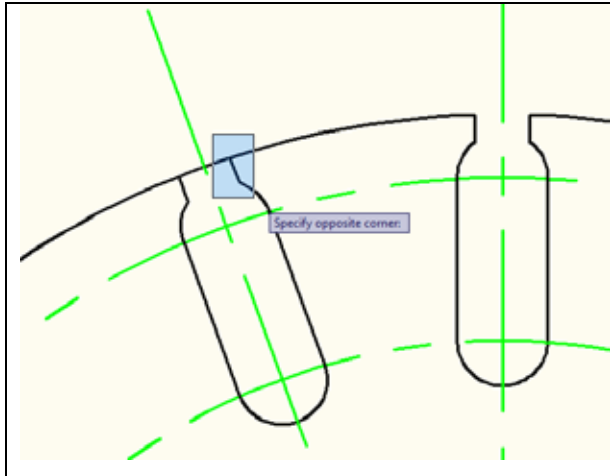
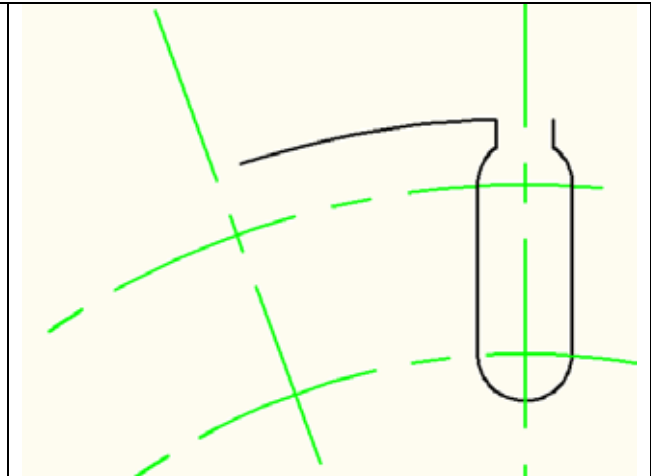**Figure 5.44 – Trimming the perimeter**   **Figure 5.45 – Erasing the second slot**

With the initial slotted hole and the short arc that is attached to the left side remaining, it is now time to complete the Circular problem and array this pattern 18 times.

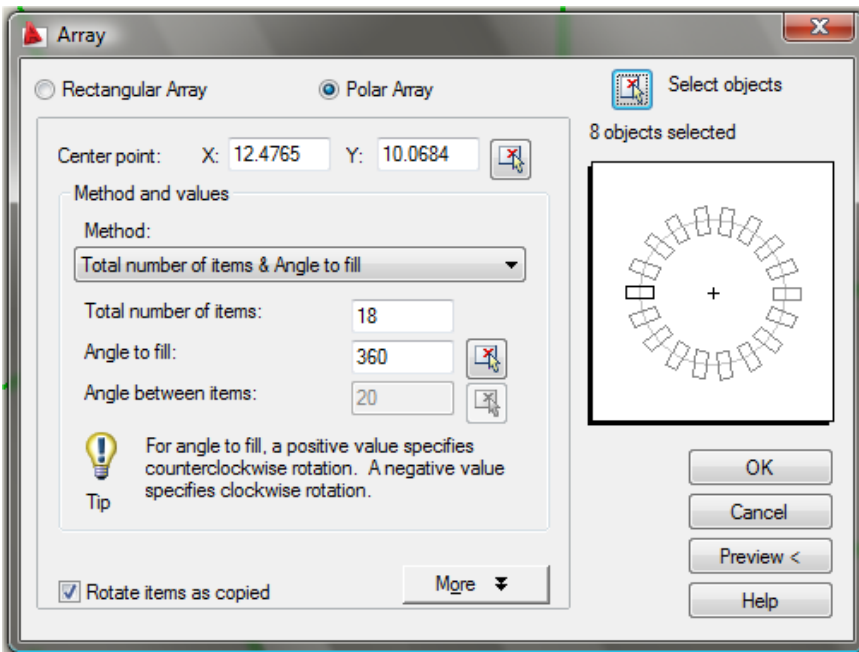## Using the Array Command to Create More Circular Patterns

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Select the Array tool on the Modify panel, which will bring up the Array window (Figure 5.46). Pick the Polar Array radial button, then the Select Objects button. Select the slotted hole using a selection window (Figure 5.47), as well as the arc to its left. Hit ENTER to return to the Array window.

Figure 5.46 – The Array window
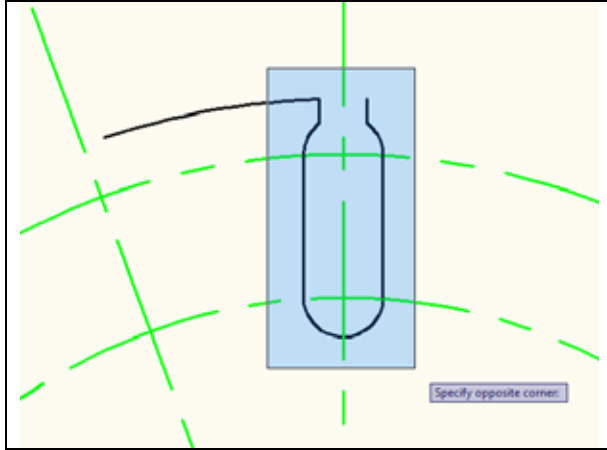
Next, select the Pick Center Point Button, which will bring you back to the drawing. Pick the center of the part by selecting the endpoint of the centerline that is also the center of the part. Change the Total number textbox to "18" and the Angle to fill textbox to 360 degrees. Select the Preview button.

Figure 5.46 – Selecting for the final array

The array will be shown on the display screen. If the Circular problem looks like Figure 5.47, right-click and the array will be completed. If the Circular problem does not look like Figure 5.47, left-click to return to the Array window and correct the settings.
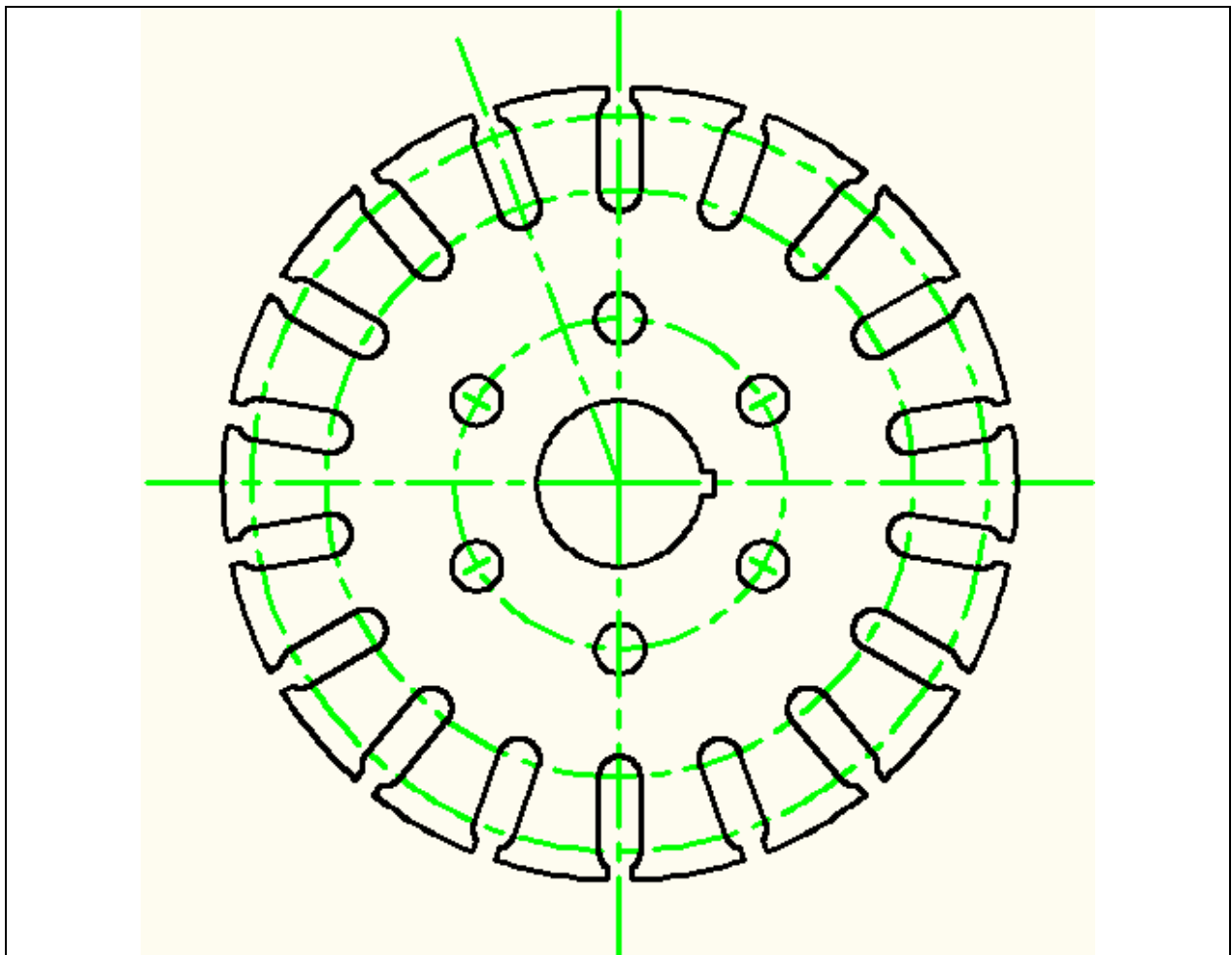

Figure 5.47 – The finished Circular problem

* World Class CAD Challenge 02-13 * - Create a New file, create the three new layers, set the Object Snap settings and draw the seven circles, move and copy the two circles, and 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 under 5 minutes to maintain your World Class ranking.