

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 arrange circles
- Using the Copy Command to add a circle
- 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. While the rectangular problem instructs basic concepts of lined components with minor curves, 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.

Using the Object Snap Settings to set the Center Osnap

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 tray and clicking Settings, which will bring up the Object Snap tab on the Drafting Settings window. You can also right click on the Object Snap tool and click on the different Object Snap icons in the menu that appears to toggle them on and off without bringing up the Drafting Settings window. In the circular problem, you need to select the Center Object Snap mode for grabbing the center point of a circle, recognized by a green circle when selected (Figure 5.1).

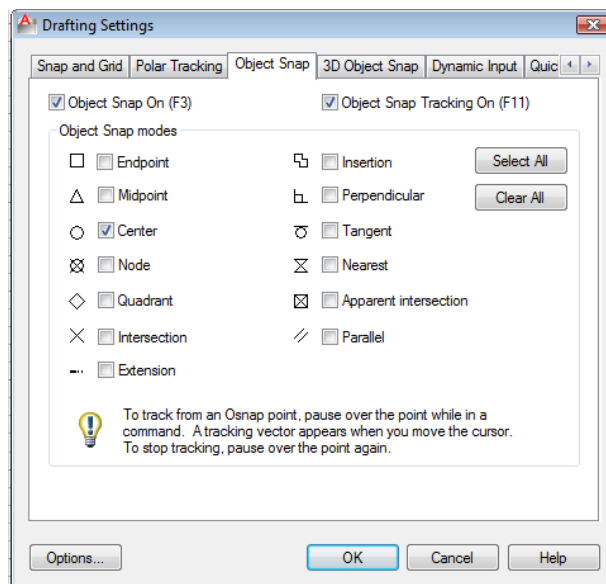


Figure 5.1 – Setting the center Osnap

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.

Creating Layers in an AutoCAD Drawing

You are already familiar with the concepts of layers from when you created layers in the dimensioning chapter of 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.

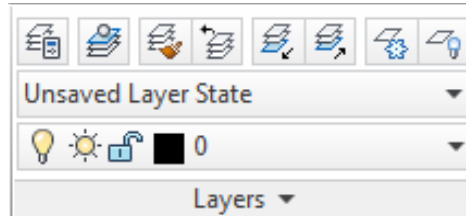


Figure 5.2 - The Layer Panel

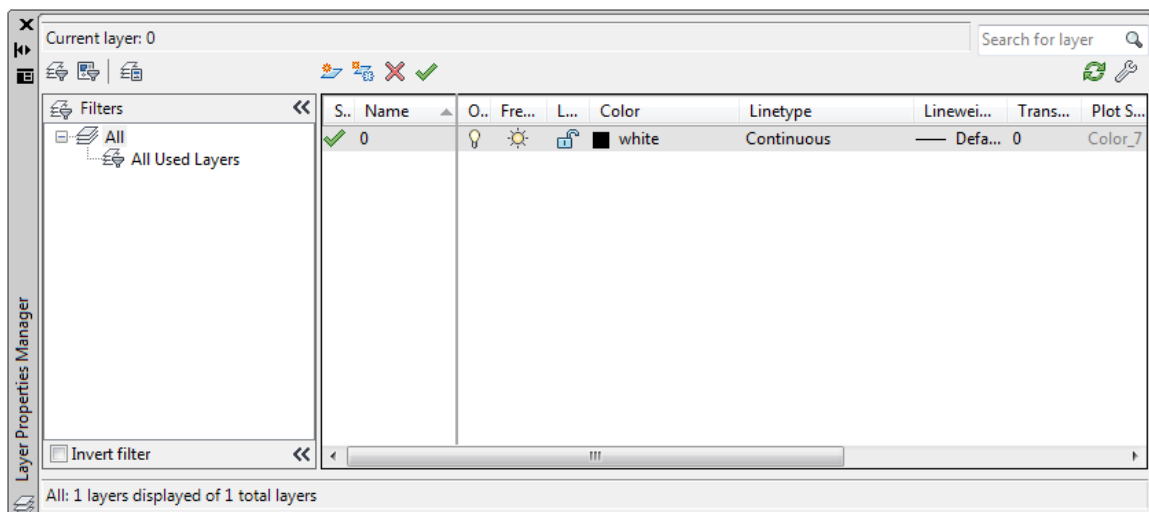


Figure 5.3 - The Layer Properties Manager window

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 distinguish between the different entities on 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.

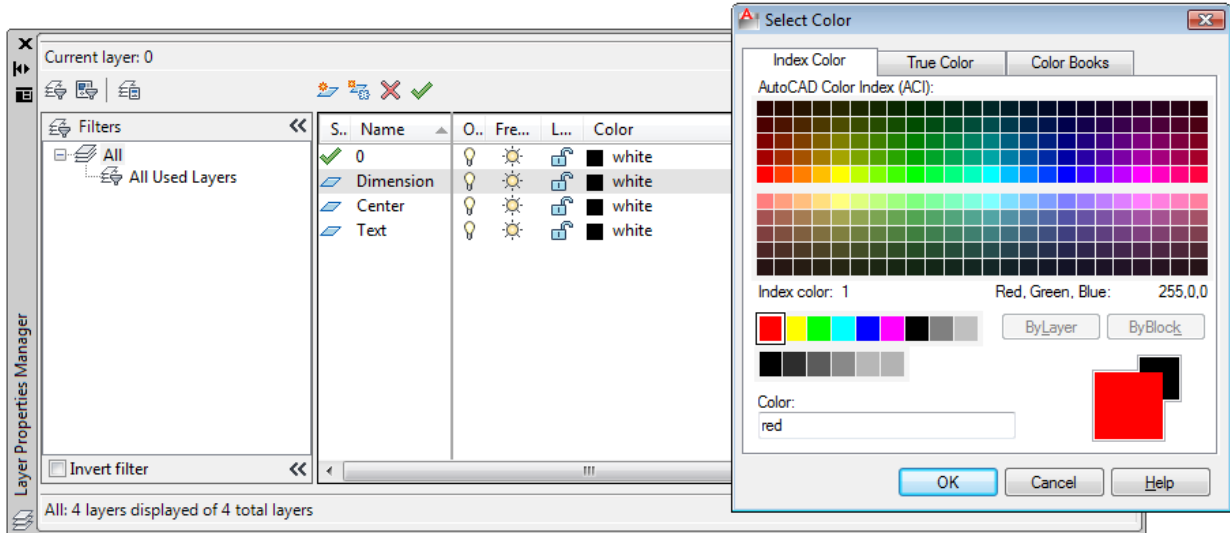


Figure 5.4 - The Select Color window

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.

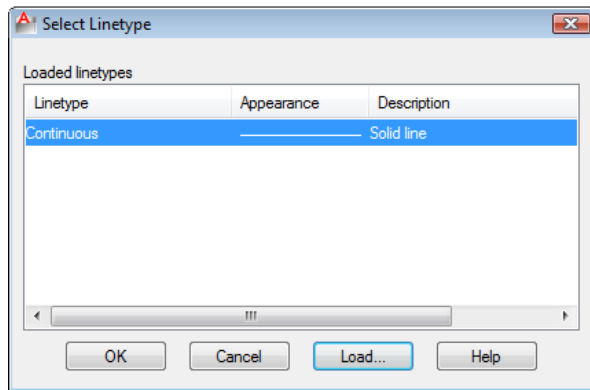


Figure 5.5 - The Select Linetype window

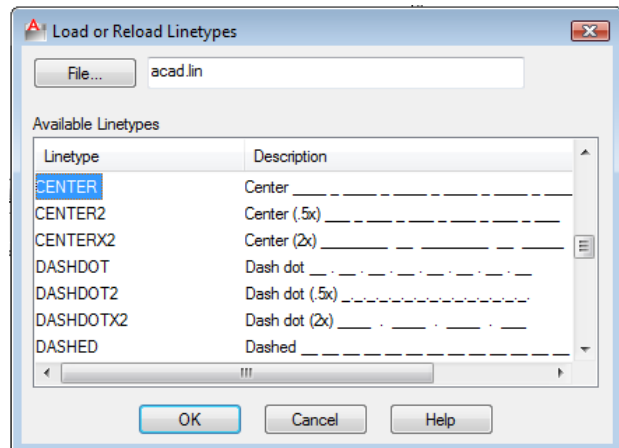


Figure 5.6 - Load or Reload Linetypes

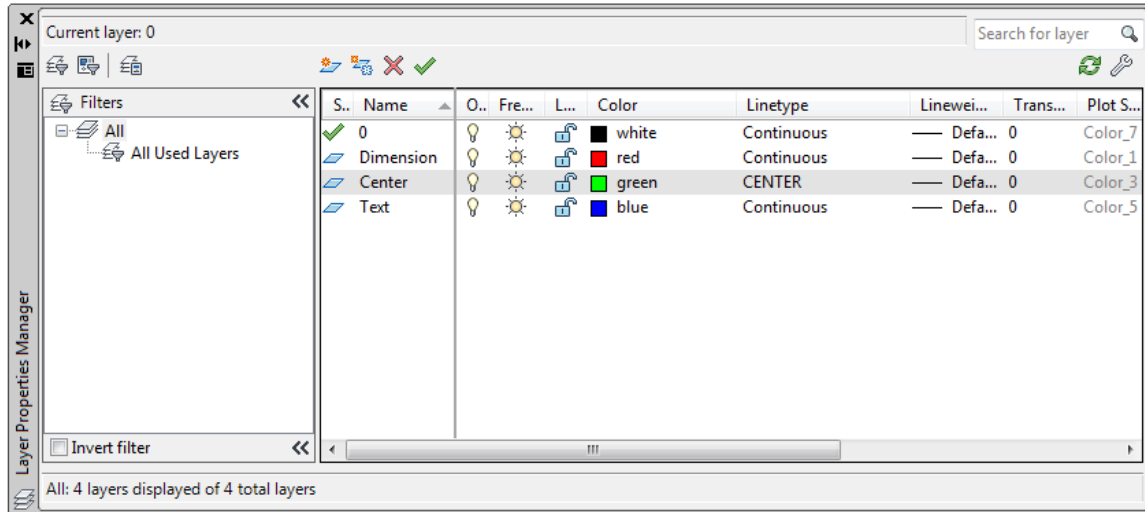
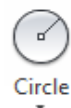


Figure 5.7 - Load or Reload Linetypes

The strategy of the Circular problem is to draw the perimeter circle first to define the outside of the drawing, then to draw the other circles inside and finally to place 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 objects 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; your time will represent your understanding and ability to operate the individual functions involved.

Drawing circles using Object Snaps

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. You will draw the last circle using the Offset command in the next section of this problem.



To begin, select the Circle command on the Draw panel under the Home tab and pick a center point on the center of your display with the left mouse button. 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/Tr (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

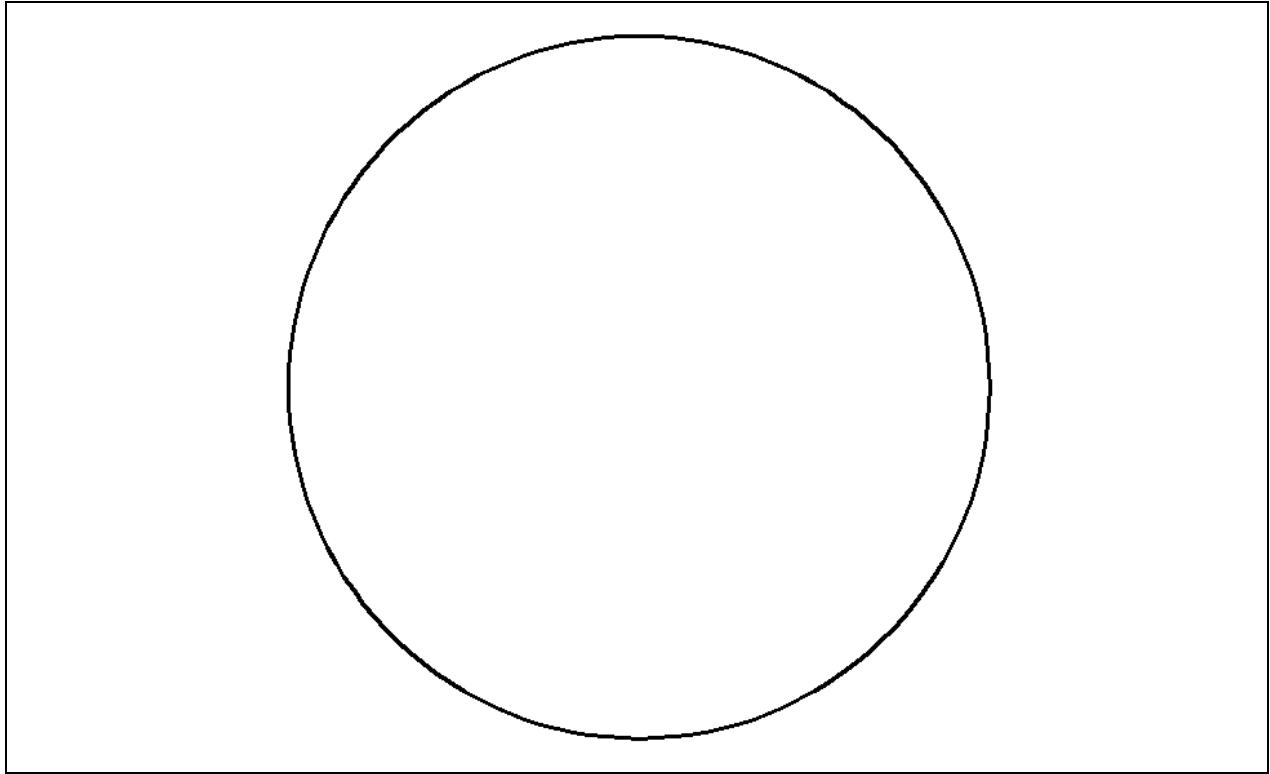


Figure 5.9 – The 12 unit circle, the outside perimeter of the Circular problem

Using the Zoom Extents Command to display the entire drawing

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 navigation bar on the right side of the graphical display (Figure 5.10). The entire 12-unit circle will appear maximized in the graphical display.

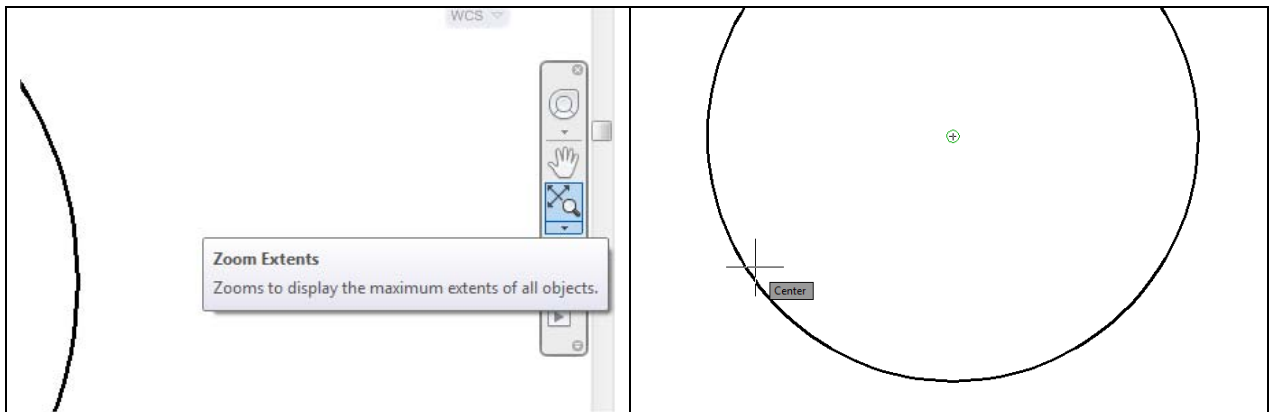
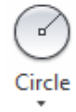


Figure 5.10 – Navigation Bar

Figure 5.11 – Using the Center Osnap

Adding more Circles using the Diameter option of the Circle Command

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 (green circle) will appear at the center of that circle (Figure 5.11). Select that point as the center point of the second circle by selecting with the left mouse button. 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” to use the diameter option of the circle command. This is 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. Many Architects and Engineers will create markup drawings dimensioned with diameters because diameters conform to better tolerance control. Remember that radius dimensions with normal company tolerances will change the part size by a factor of two. Because of this, you should try to use diameters whenever possible when drawing and dimensioning.

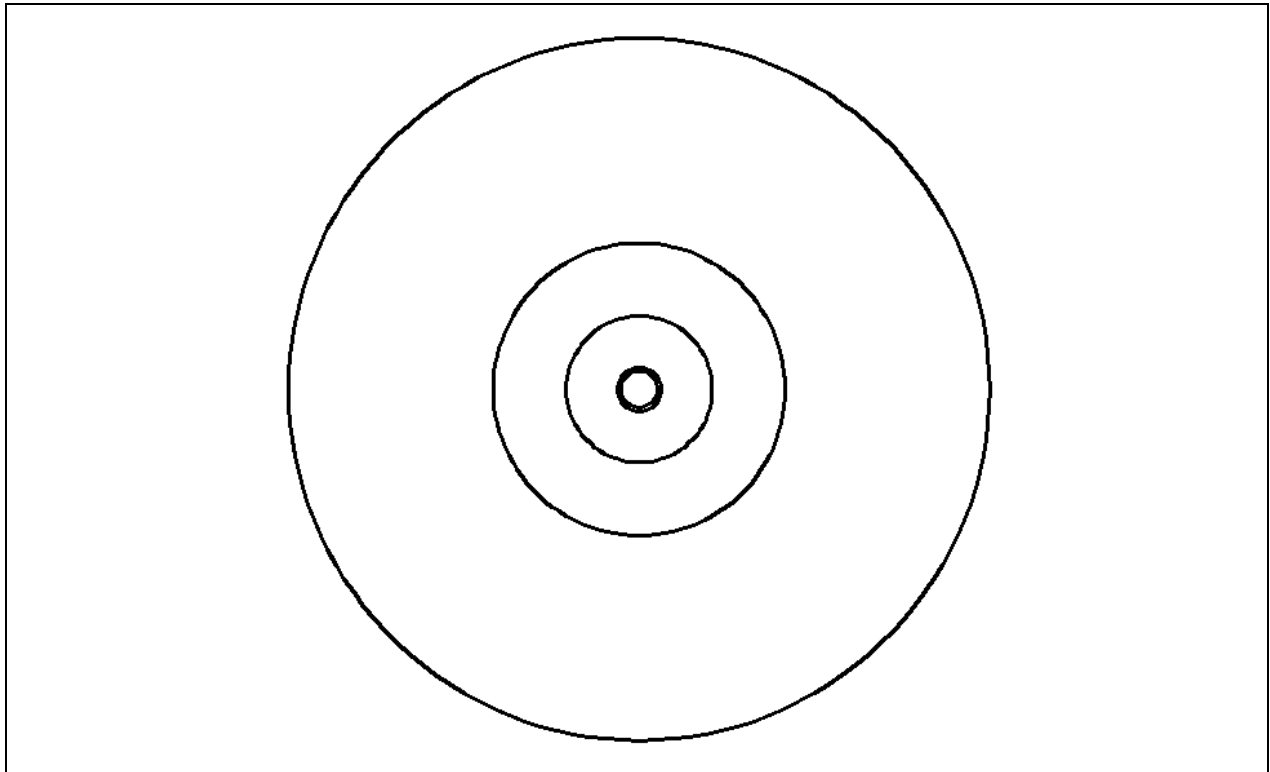


Figure 5.12 – Placing the 5, 2.5, 0.75 and 0.625 diameter circles in the drawing

Adding a Circle using the Radius option of the Circle Command

You need to add a sixth concentric circle by hitting 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, relatively large circle will be drawn between the 5 and 12-diameter circle.



```
Command: _circle Specify center point for circle or [3P/2P/Tr (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

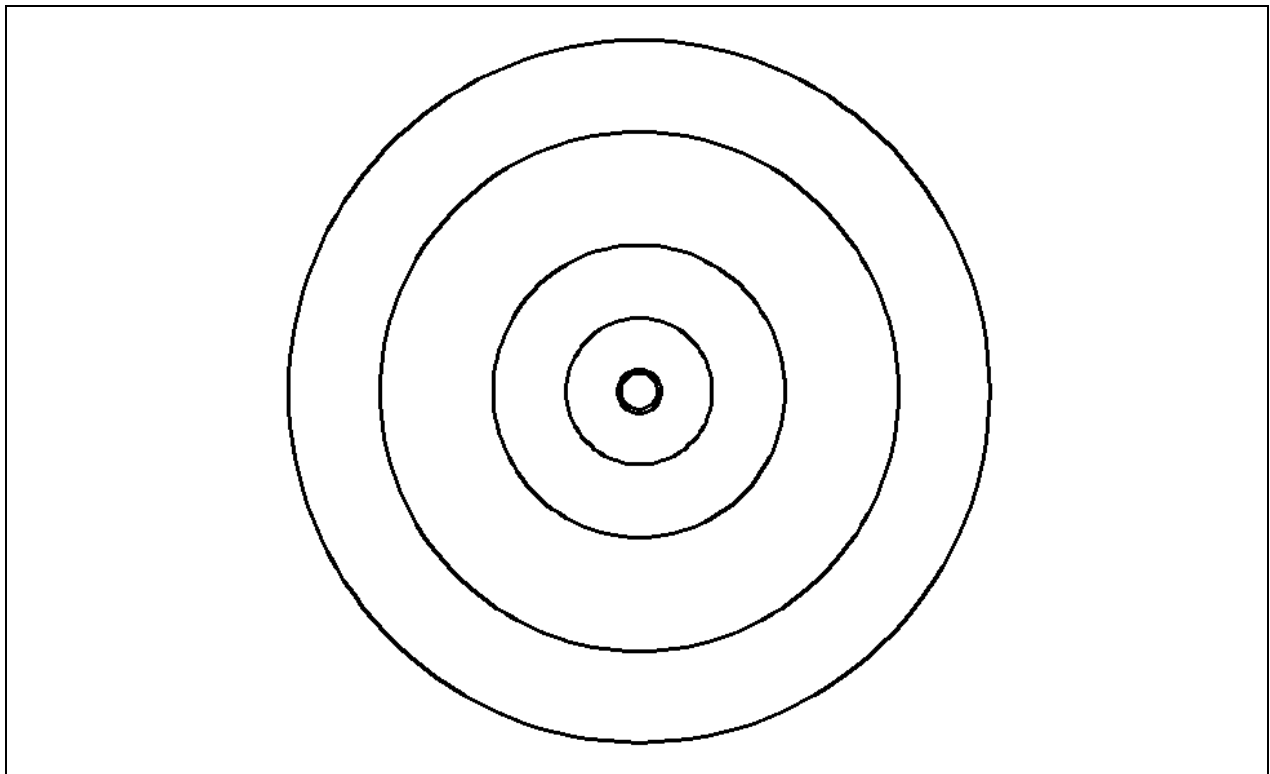


Figure 5.14 – The 4.4375 radius circle

Using the Offset Command to add another circle

To add the seventh circle, you are not going to use the Circle command. Rather, 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 1.125 units to the outside of the 4.4375 circle, 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 without errors.

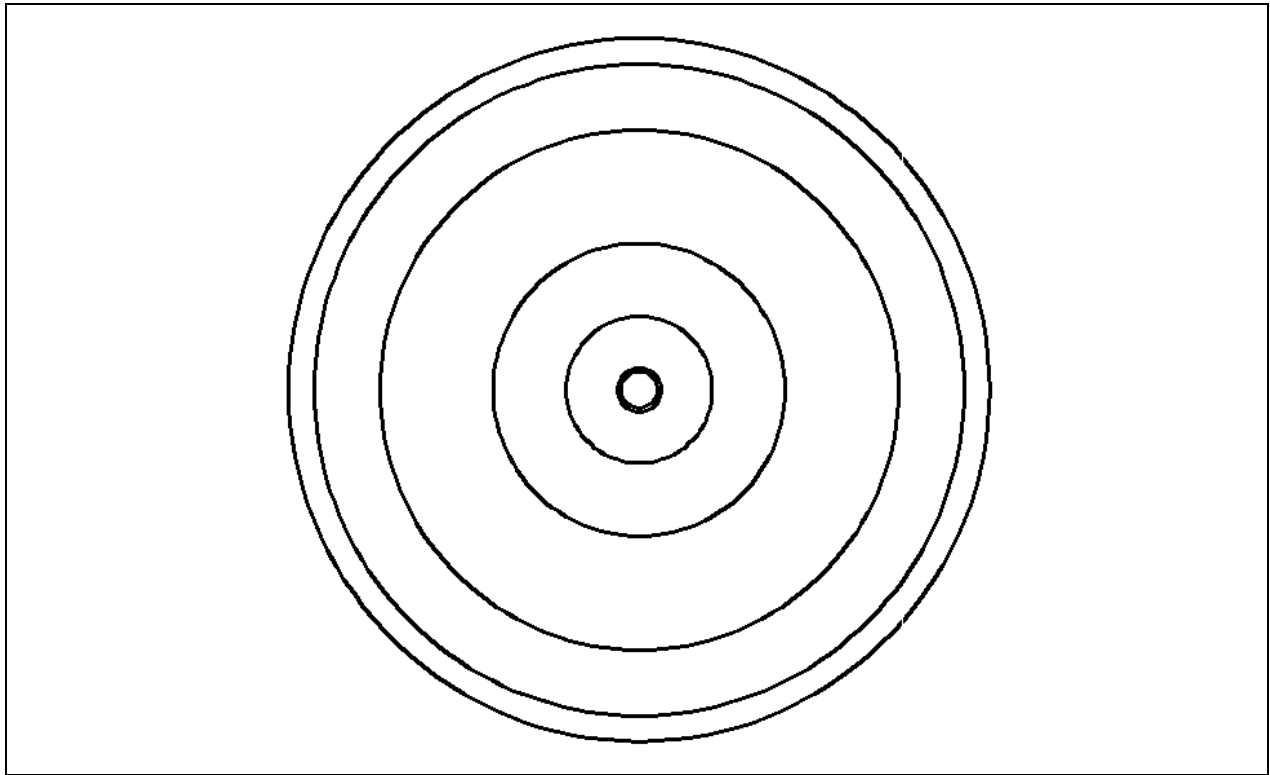


Figure 5.15 – Using the Offset command to draw the seventh circle

*** 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 in less than 120 seconds to maintain your World Class ranking.**

Adding centerlines to the drawing using the Line Command

Just like in the Rectangular problem, you will add centerlines to show symmetry and 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. Because the polar or circular array will be employed in this construction technique, you will only need to draw a single quadrant



centerline. You will eventually copy and rotate the centerline to all four quadrants using the Array command on the Modify panel.

First, zoom in closer to your work by using the Zoom Window command on the navigation bar. Select the Zoom Window tool by selecting the small triangle below the Zoom Extents icon and selecting Zoom window from the list that appears. 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.

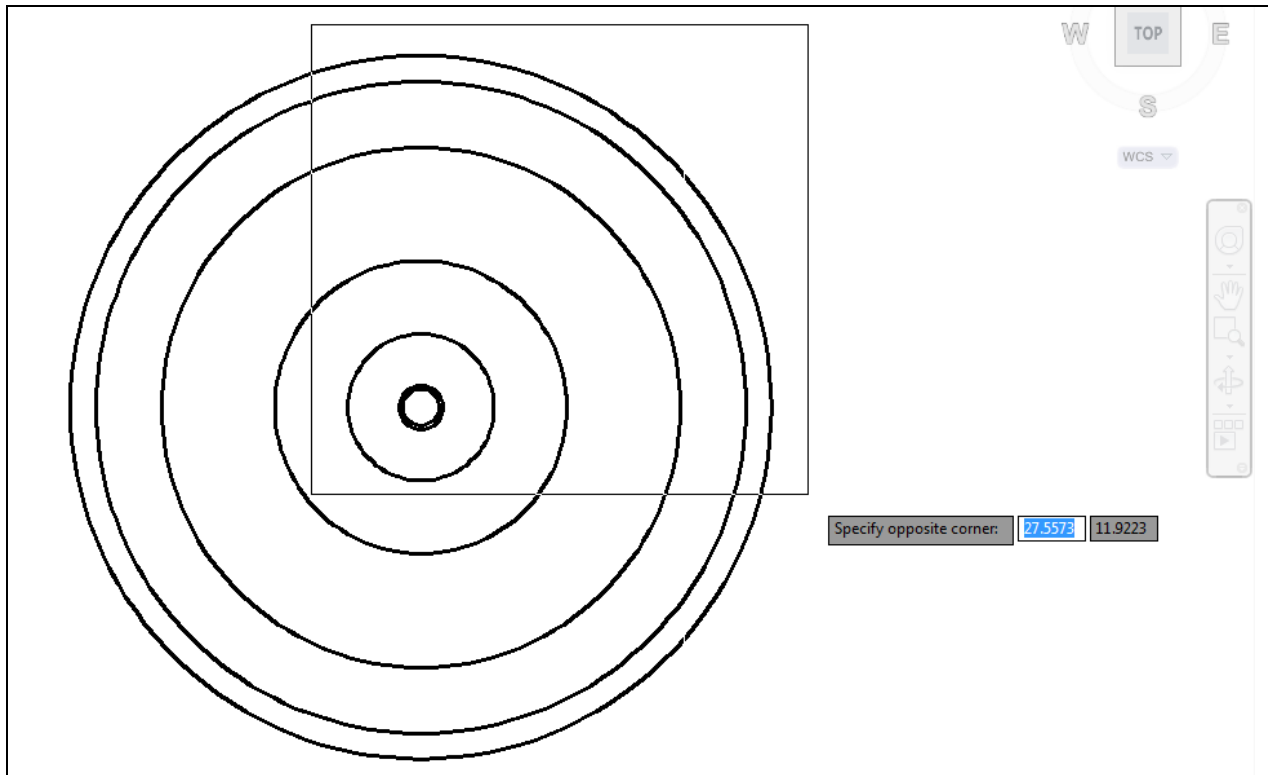
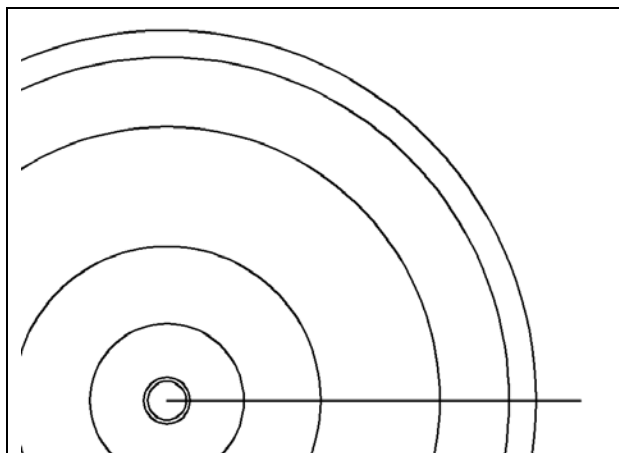


Figure 5.16 – Using the Zoom Window command to get in close to your work



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 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, keeping all lines at right angles. The center line should look like the one shown in Figure 5.17.

Figure 5.17 – The horizontal centerline

Expert CAD operators trained in World Class techniques will change the Osnap settings periodically in a drawing to save time. Bring up the Object Snap settings by right-clicking on the Object Snaps tool on the tray and selecting Settings from the menu. Press the “Clear All” button on the Drafting Settings window and then check Endpoint, Midpoint, Quadrant, Intersection and Perpendicular Object Snaps. Select “OK” to confirm the changes (Figure 5.18).

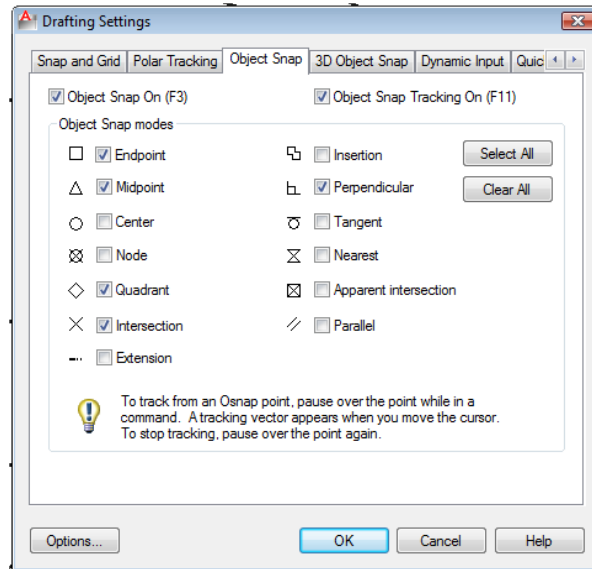

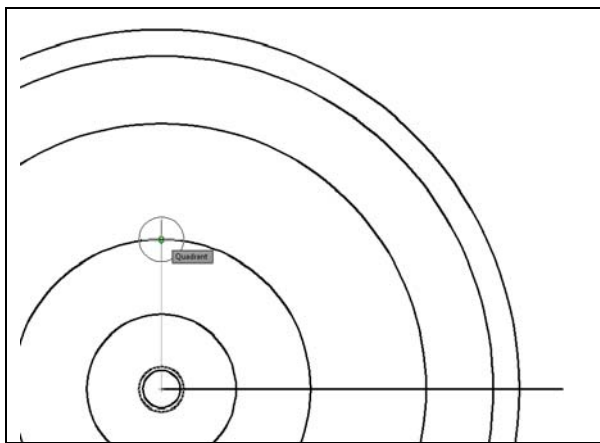


Figure 5.18 – Changing the Object Snap settings to maintain drawing efficiency

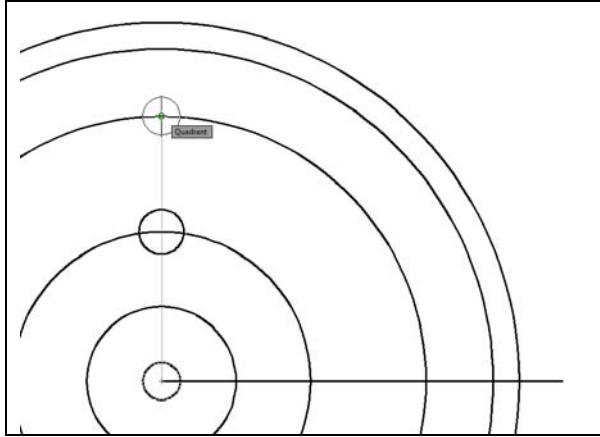
Using the Move Command to place circles in appropriate position

Use the Move command to place 0.625 and 0.75 circles into the appropriate positions  Move on the drawing. After selecting the Move tool from the Modify panel, select the 0.75 circle. You will be prompted to select other objects, so hit 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, the 0.75 circle will be centered on the north quadrant of the 2.5-unit circle (Figure 5.19).

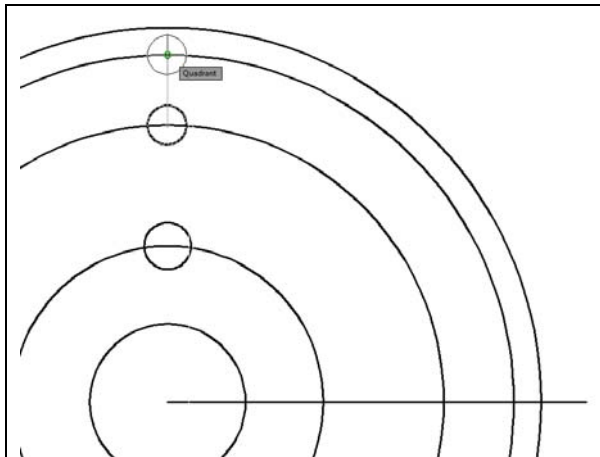
Figure 5.19 – Moving the 0.75 circle



Move the 0.625 circle to the north quadrant of the 4.4375 radius circle using the same technique: select the 0.625 circle, hit ENTER, select the endpoint of the centerline, and then the north quadrant of the 4.4375 radius circle. The 0.625 circle will have 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 the slotted hole in this part, you need to copy the 0.625 circle to the north quadrant of the next largest circle. Select the Copy tool on the Modify panel. After selecting the circle, hit 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 pick the quadrant of the circle that is 1.125 to the outside for the second base point of displacement, 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 in less than 140 seconds to maintain your World Class ranking.**

Using the Line Command to add details

You will now add a series of lines to your circular part to add details such as centerlines, the slotted hole, and a keyway that would prevent the part from slipping on its axel 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 parts. 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.

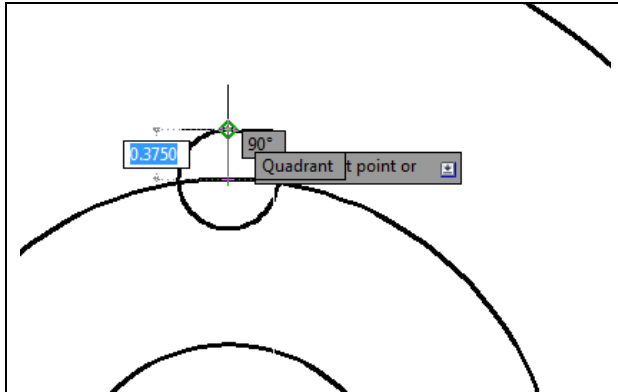


Figure 5.22 – Drawing a center mark

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. Hit ENTER to finish the line segment.

Continue forming the slotted hole by drawing lines from the tangent points of each circle. In most cases, the Quadrant Osnap tool will remain a primary tool to draw details on the slot. 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.

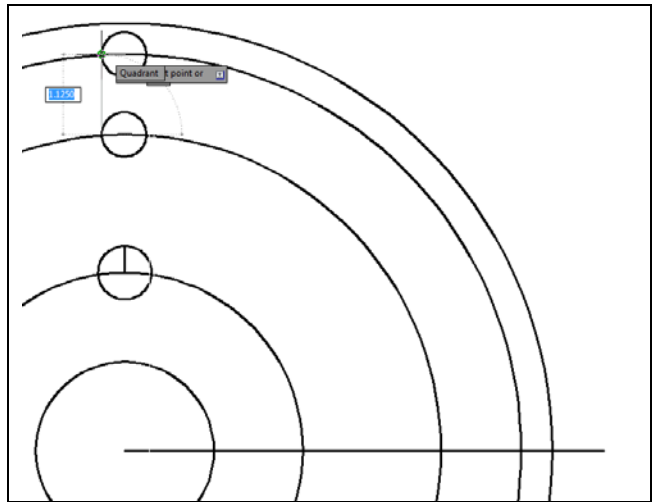


Figure 5.23 – Drawing the first slot line

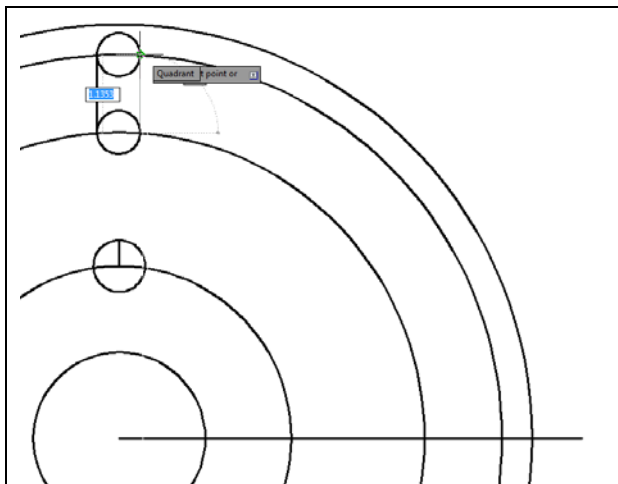


Figure 5.24 – Drawing the second slot line

To add the second line, select the Line tool on the Draw panel and pick 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 hit ENTER to repeat the previous command instead of selecting the tool again from the ribbon.

Keyways are present on many circular parts. The key prevents slippage during the rotation of the assembly. In Figure 5.25, you can see a line segment 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. This line segment is a construction line and requires no particular length.

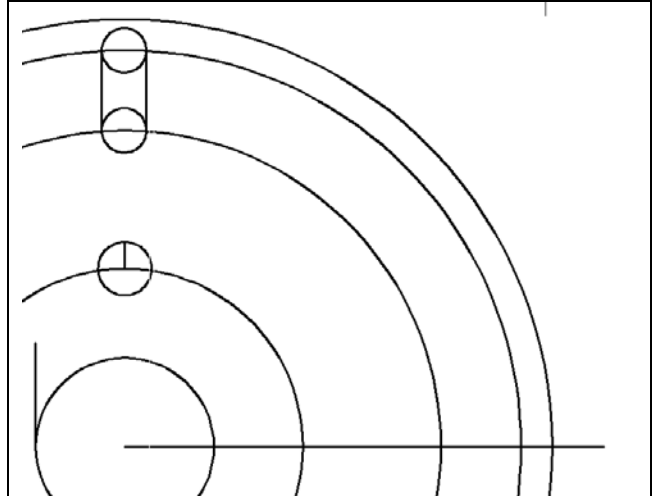


Figure 5.25 – Draw the keyway line

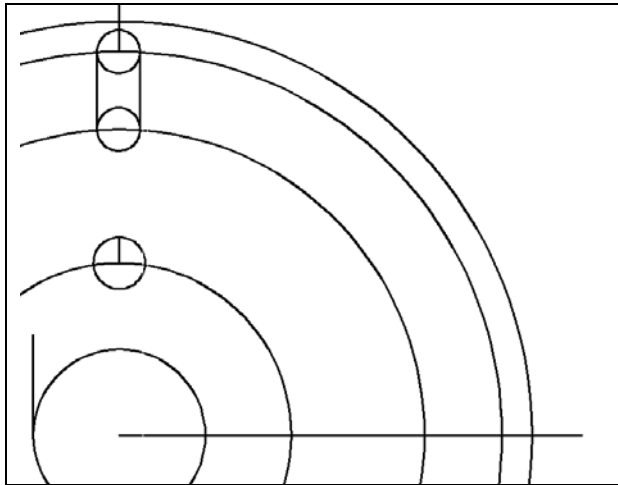



Figure 5.26 – Drawing a construction line

The top of the slotted hole at the near the north quadrant of the circular part will be open, so you need to 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). Once again this is a construction line, requiring no particular length.

Using the Move Command to add details

As you continue to draft the circular problem, you need to move the keyway line into  Move position.

Select the Move tool on the Modify panel, select the keyway line and hit 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 so that 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 movement and type 2.687 and ENTER. The keyway line will move to the appropriate location. (Figure 5.27)

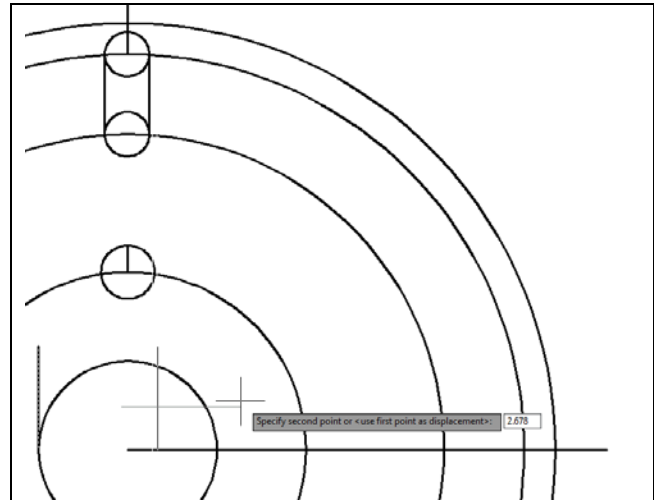


Figure 5.27 – Moving the keyway line

You also need to reposition the centerline drawn on the 0.75 circle. Select the Move tool on the Modify panel and hit ENTER. You will be prompted to select the base point or displacement. Pick the midpoint of the centerline using the Midpoint Object Snap. 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)

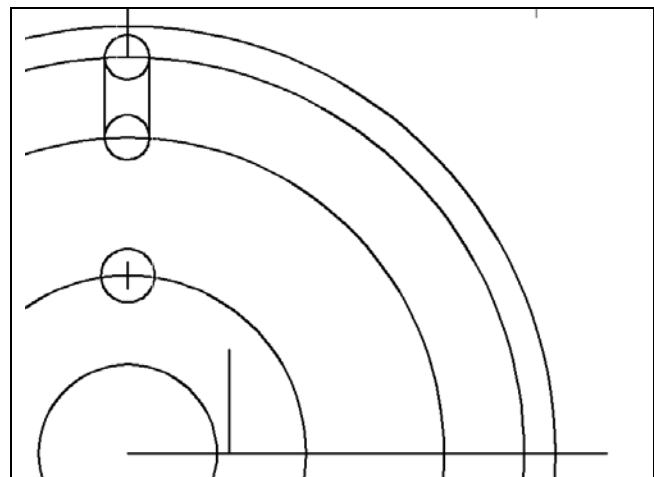


Figure 5.28 – Centering the center mark

Using the Offset Command to add more detail

When your initial lines are drawn accurately, the Offset command will allow you to quickly add details similar to the ones you have already placed. This is much faster than drawing original entities using the same similar actions with other commands.



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, type 0.1875 for the offset distance, and hit 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).

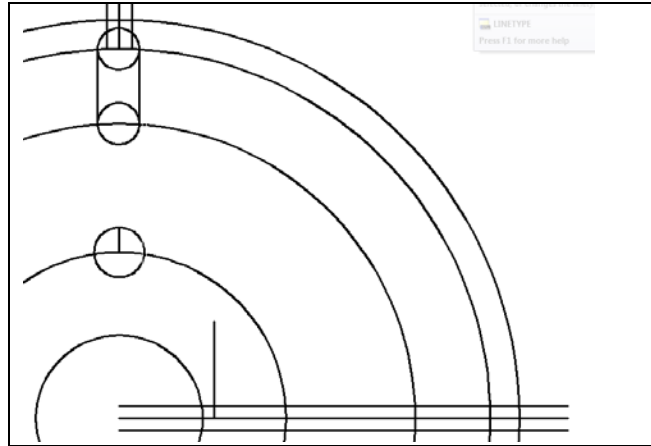


Figure 5.29 – Offsetting lines

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 in less than 190 seconds to maintain your World Class ranking.**

Using the Fillet Command to join two non-intersecting lines

The fillet command can be an effective trimming and joining tool when the radius is set to zero. 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.

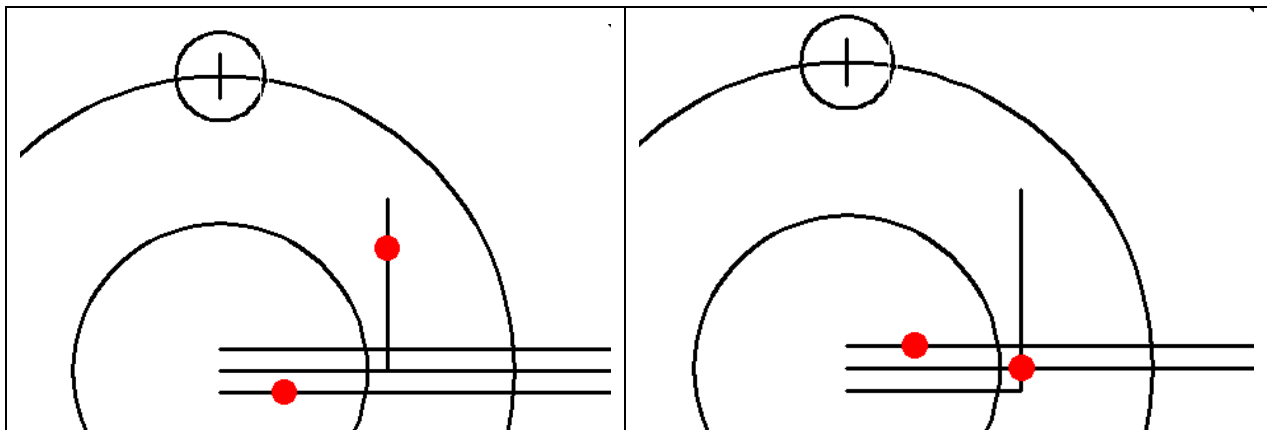


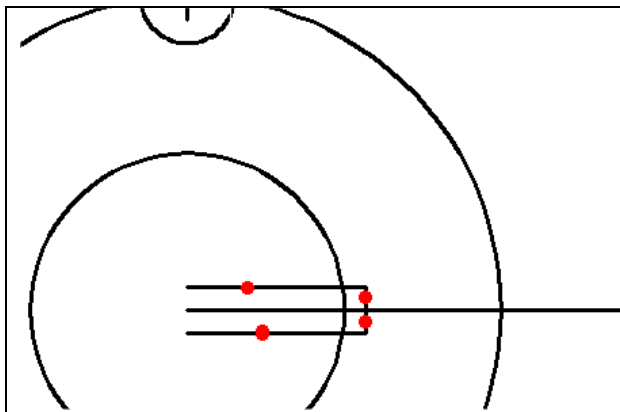
Figure 5.30 – Forming the key by filleting

Figure 5.31 – Forming the key by filleting

In Figures 5.30 and 5.31, you create zero radius corners 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

In efficient computer aided design, you use construction lines to create a new drawing. One aspect that 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. -/- Trim



To remove the two line segments entering the 2.5 diameter circle and to open the circle to the 0.375 key, use the Trim tool on the Modify panel. Hit ENTER to make every object line a cutting line. Now make four selections, two on each of the two-keyway centerline offset lines entering the 2.5 diameter circle and two on each of 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 – Trimming construction lines

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. Hit 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.

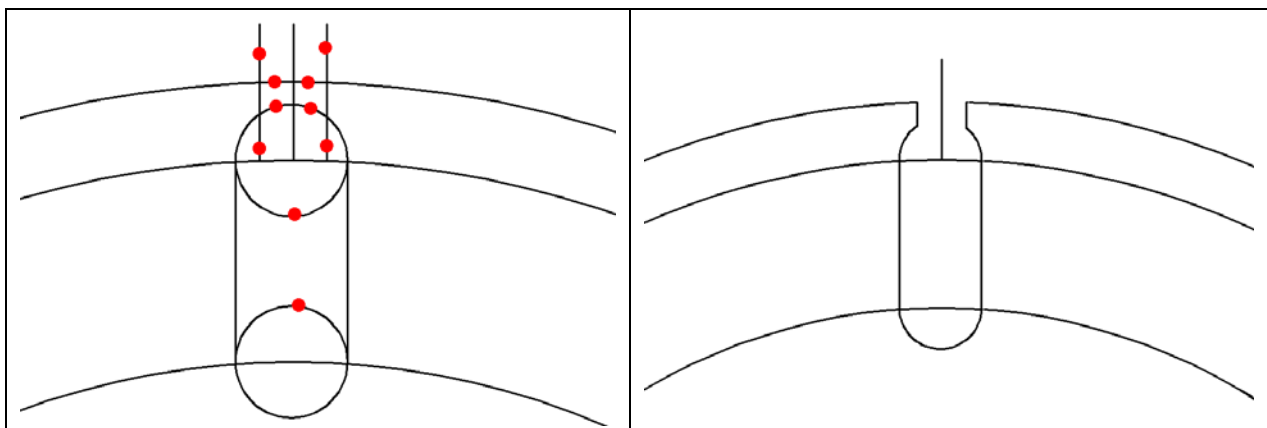


Figure 5.33 – Detailing the slot with Trim

Figure 5.34 – The detailed slot

Using the Layer Control Drop Box to change an entity's layer

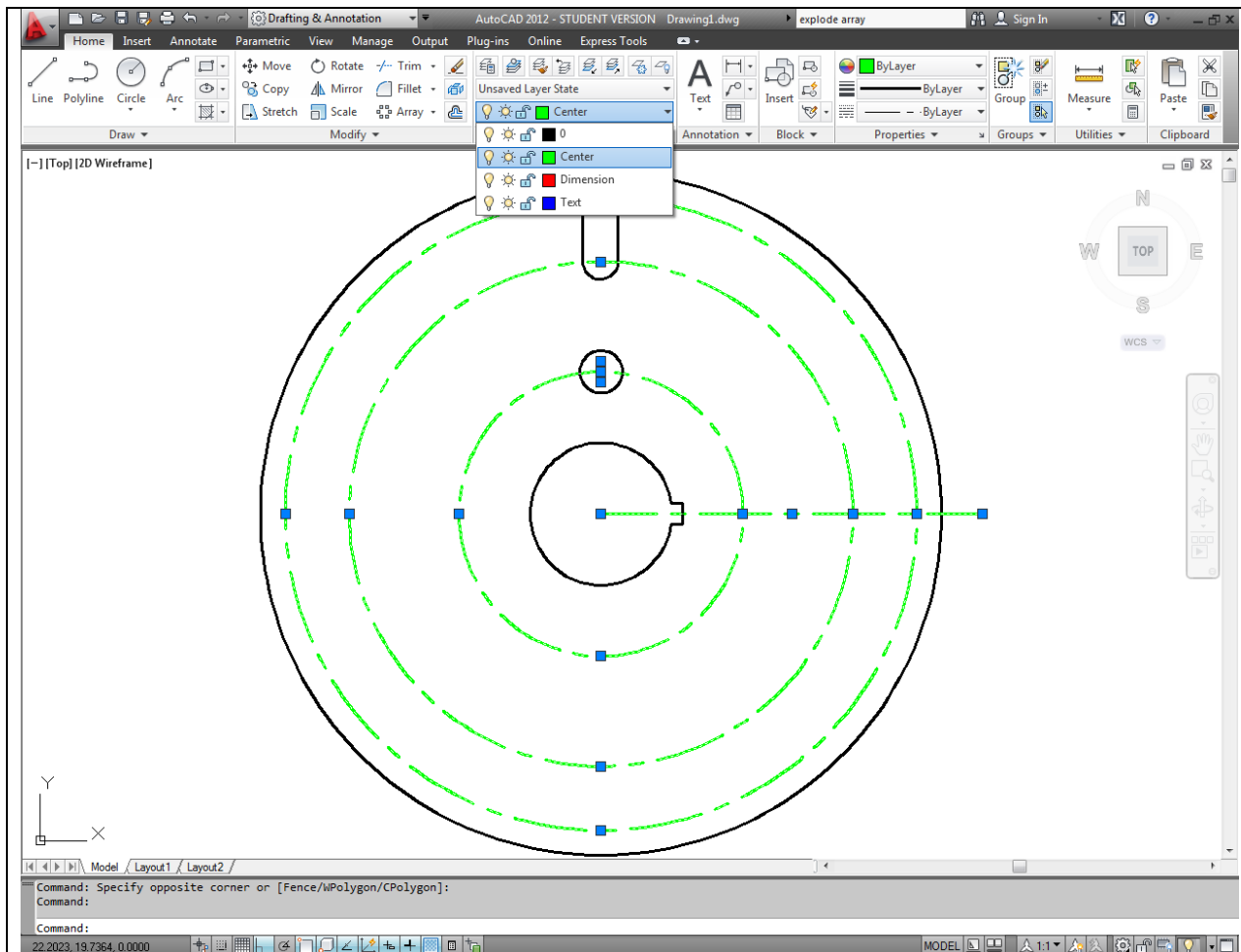


Figure 5.35 – Objects place on the Center layer

You may have already noticed that drawing entities, such as lines, arcs, and circles, can be selected prior to choosing an AutoCAD tool. To place certain line and circle entities on the Center layer, make your selections 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, 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

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 not only exposed to the rectangular array, but also the new



windowless version of the array command present in AutoCAD 2012. Now you will perform four separate polar arrays, the first of which will involve 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 unnecessary 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. You will also want to toggle the ortho mode off (F8 on the keyboard) and change the Osnap settings so that only the Center Osnap is toggled on.

Select the Polar Array tool on the Modify panel, select the 0.75 diameter circle and its centerline, and hit ENTER. Now you will be prompted to select the centerpoint of the array, so select the center Osnap point of the part (Figure 5.36). Next you need to enter the number of items in the array. You can either use the cursor to drag a series of circles around the center of the array so that there are 6 showing, including the original you selected to begin the array command, and select with the left mouse button. Or, you can type 6 at the keyboard and hit ENTER to specify the number of items (Figure 5.37). To specify the angle to fill, type 360 and hit ENTER. Finally, hit ENTER once more or select eXit from the dynamic input menu to exit the polar array command (Figure 5.38). Unlike the rectangular version of the array command, the polar array in this case required more data to be entered at the keyboard. The only Osnap point used was to select the center of the array.

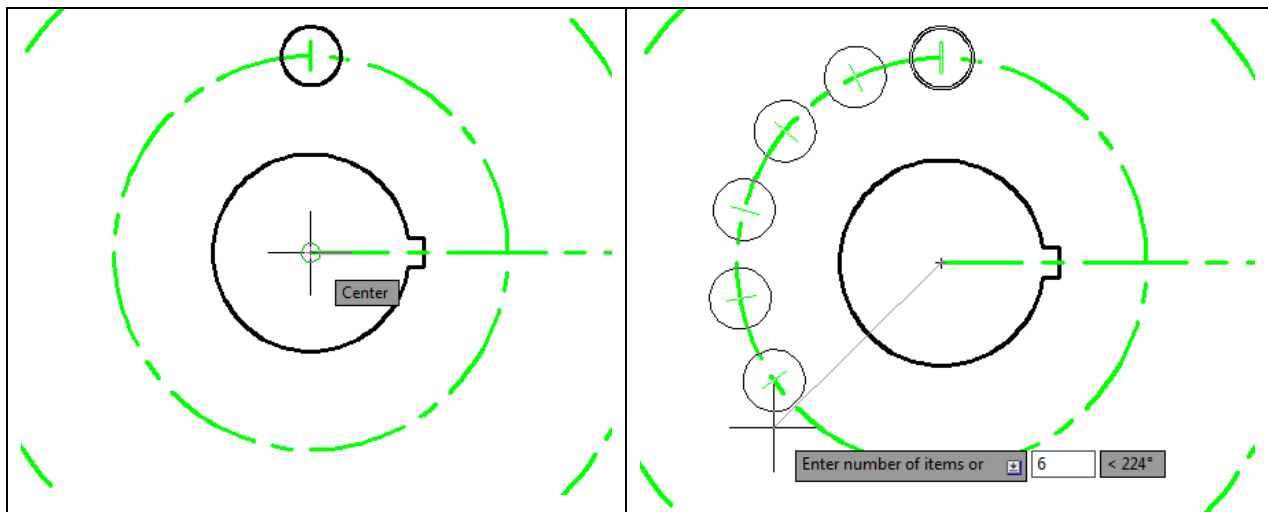


Figure 5.36 – The Center of the array

Figure 5.37 – Specifying 6 items

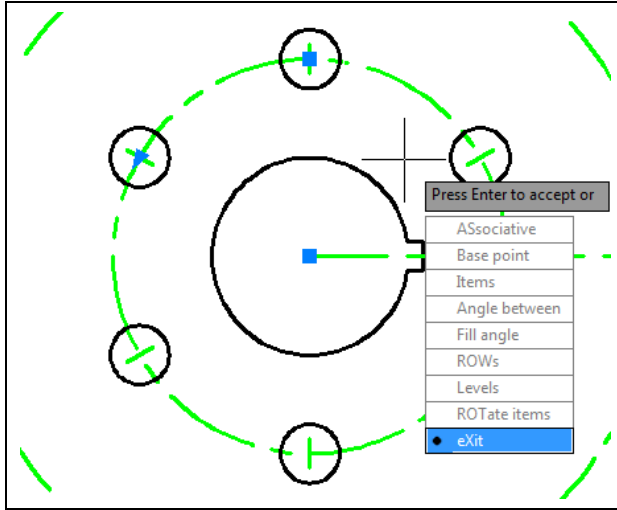


Figure 5.38 – The finished array

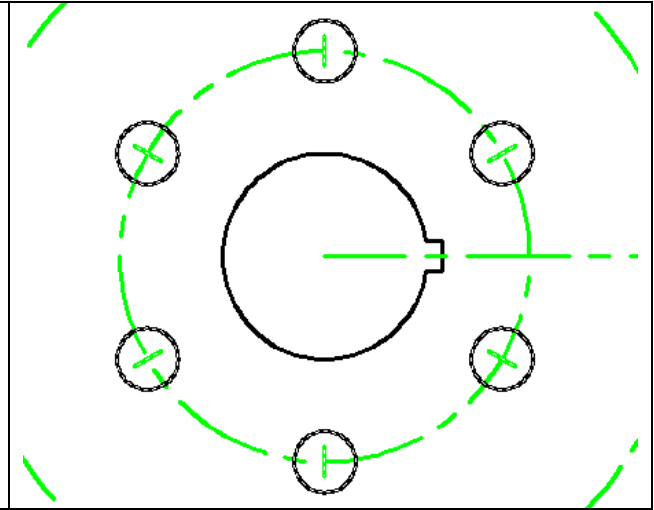


Figure 5.39 – Exploding the array

After completing your first polar array, type “explode” at the command line and hit ENTER. You will need to separate the array into its individual entities in order to remove two of the centerlines that will be overlapped by your second array. Select one of the entities in the array and hit ENTER to explode the array (Figure 5.39). Now you will be able to select each entity individually.

Using the Erase Command to remove three lines

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.

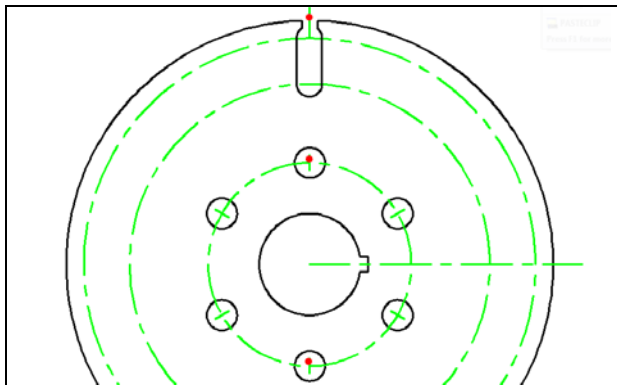


Figure 5.40 – Erasing three lines

Select the Erase tool on the Modify panel, and select the three entities on the part as shown in Figure 5.40. 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.

Using the Array Command to create more circular patterns

Select the Polar Array tool on the Modify panel. Select the main centerline on the display screen and hit ENTER. Then select the center Osnap as the center of the array (Figure 5.41). Type 4 and ENTER for the number of items in the array, and then type 360 and ENTER for the angle to fill (Figure 5.42). You can also space the centerlines visually by select the center of the bottom 0.75 circle as shown in Figure 5.43. Finally, hit ENTER once more to exit the array command. You will see four centerlines, each extending out through a different quadrant of the circle. You will need to explode this array as well by selecting the array with the cursor and then typing “explode” at the keyboard and hitting ENTER (Figure 5.44). You will need to select the top centerline individually in your next polar array.

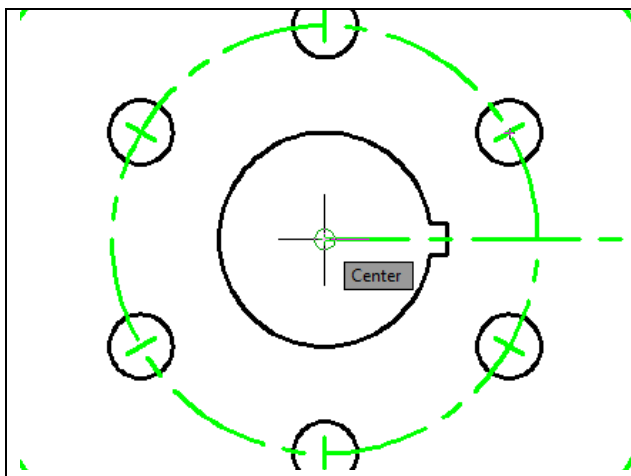


Figure 5.41 – Selecting the center

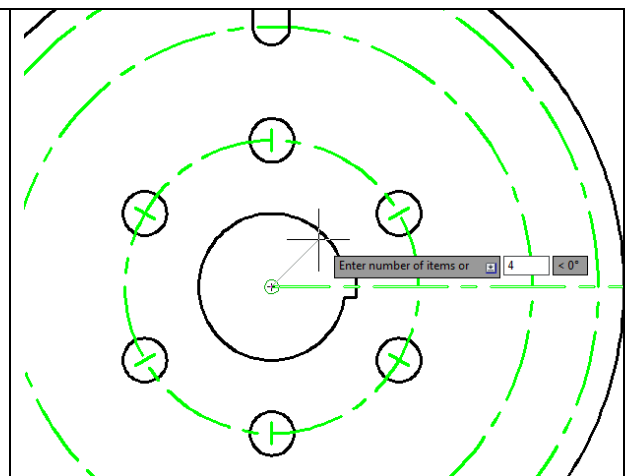


Figure 5.42 – Specifying four entities

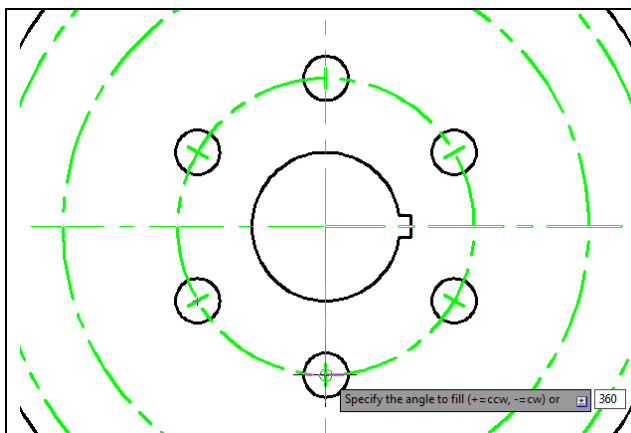


Figure 5.43 – Specifying the angle to fill

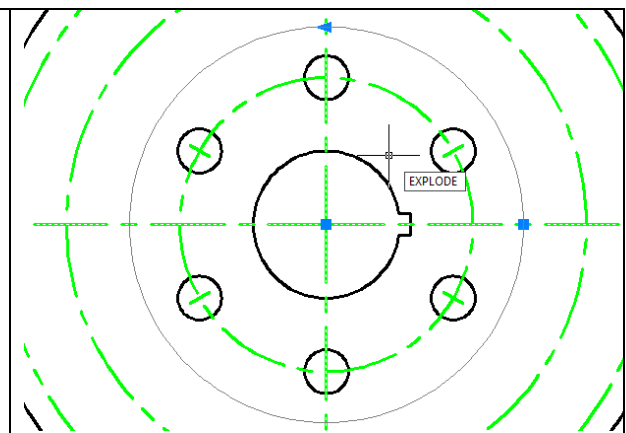


Figure 5.44 – Exploding the array

For your third polar array select the Polar Array command again from the Modify panel. Using a selection window select the entire detail of the slot (Figure 5.45). Also select the top main centerline and hit ENTER to accept the selection set. Select the center Osnap as the center of the array. Drag the cursor to the left so that only one additional entity is shown as part of the

array and left-click (Figure 5.46), or type “2” and ENTER to specify the number of items. Then type “20” at the keyboard and ENTER to specify the spacing angle (Figure 5.47). Finally, hit ENTER once more to exit the array command. You will see the slot detail and the centerline have been copied and rotated 20 degrees counterclockwise. Explode this array as well by selecting the Explode command from the Modify panel (Figure 5.48). You will need to edit individual parts of the array before finalizing the part.

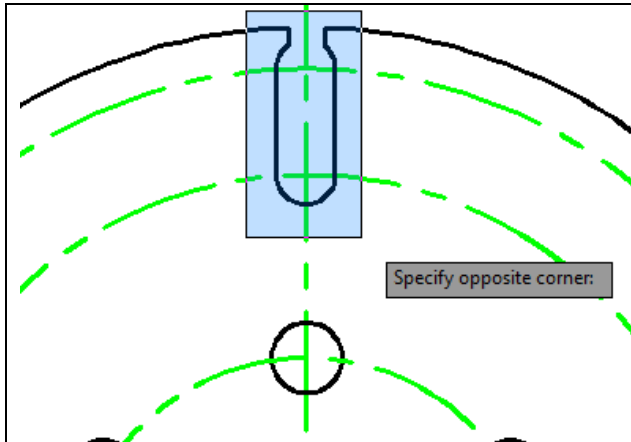


Figure 5.45 – Exploding the array

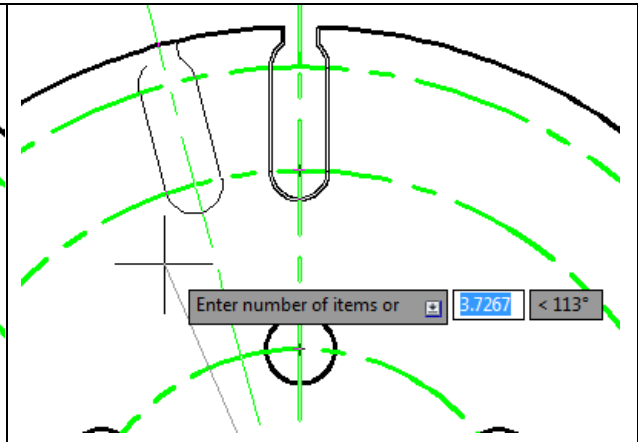


Figure 5.46 – Exploding the array

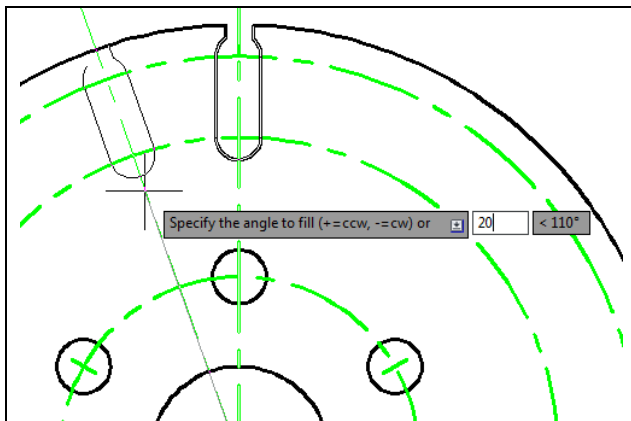


Figure 5.47 – Exploding the array

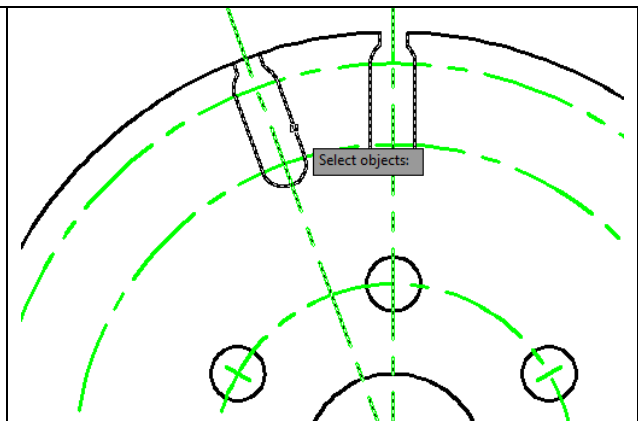


Figure 5.48 – Exploding the array

Using the Trim and Erase Commands 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 5.49. Hit 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 entire slot with a selection window as shown in Figure 5.50 and hit Delete on the keyboard to erase the detail.

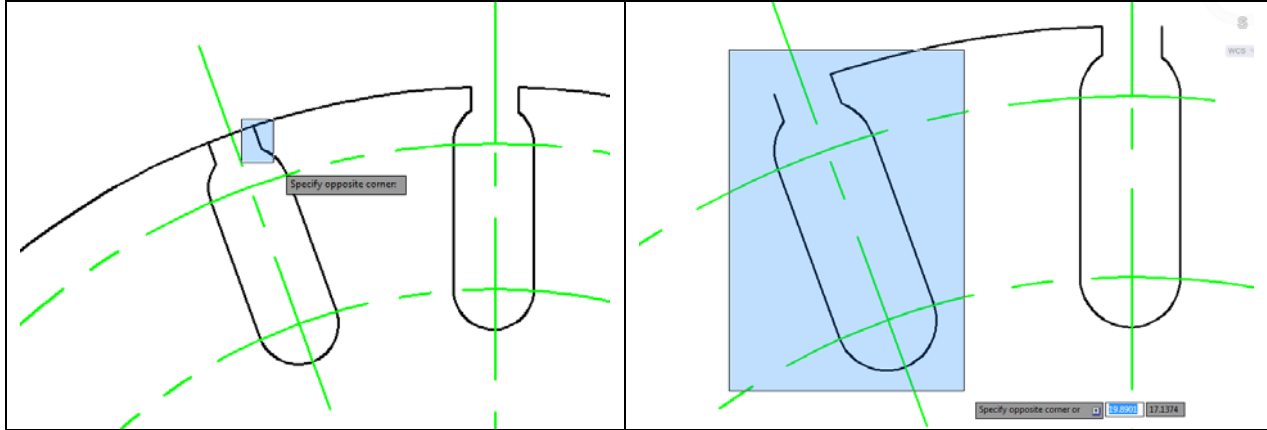


Figure 5.49 – Trimming the perimeter

Figure 5.50 – 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

The final array will create a detail around the outside of the circular part that would take a significantly longer amount of time to create if you had to draw each slot individually. With the Array command, you will create it in a matter of seconds. Select the Polar Array tool once more from the Modify panel. Use a selection window to select all of the entities in the slot detail (Figure 5.51), and then also select the small arc to the left of the slot. Hit ENTER to continue, next selecting the center Osnap as the center of the polar array. Type “18” at the keyboard and hit enter to specify 18 entities in the array (Figure 5.52), and then type “360” and ENTER to specify the angle to fill (Figure 5.53). Hit ENTER once more to exit the array command (Figure 5.54). You do not need to explode this array because it is the last detail you will add to the main part of the circular problem.

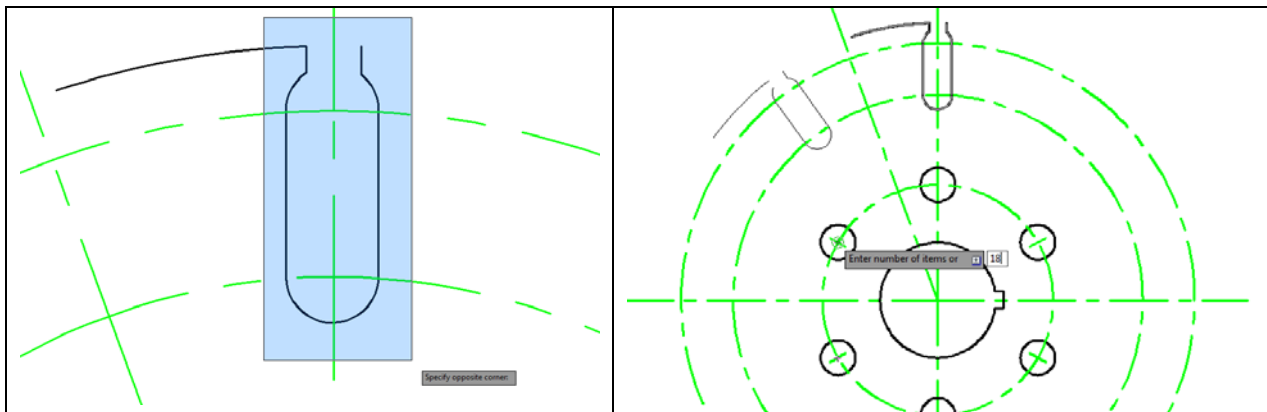


Figure 5.51 – Selecting for the final array

Figure 5.52 – Specifying 18 entities

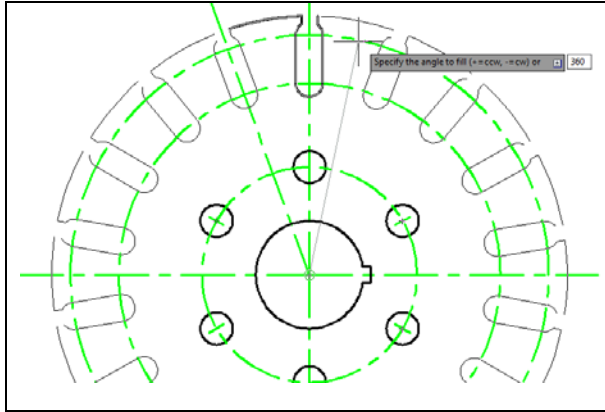


Figure 5.53 – Specifying 360 degrees

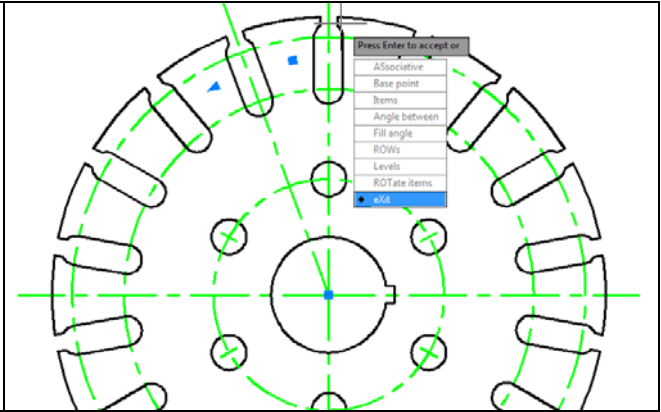


Figure 5.54 – Exiting the Polar Array tool

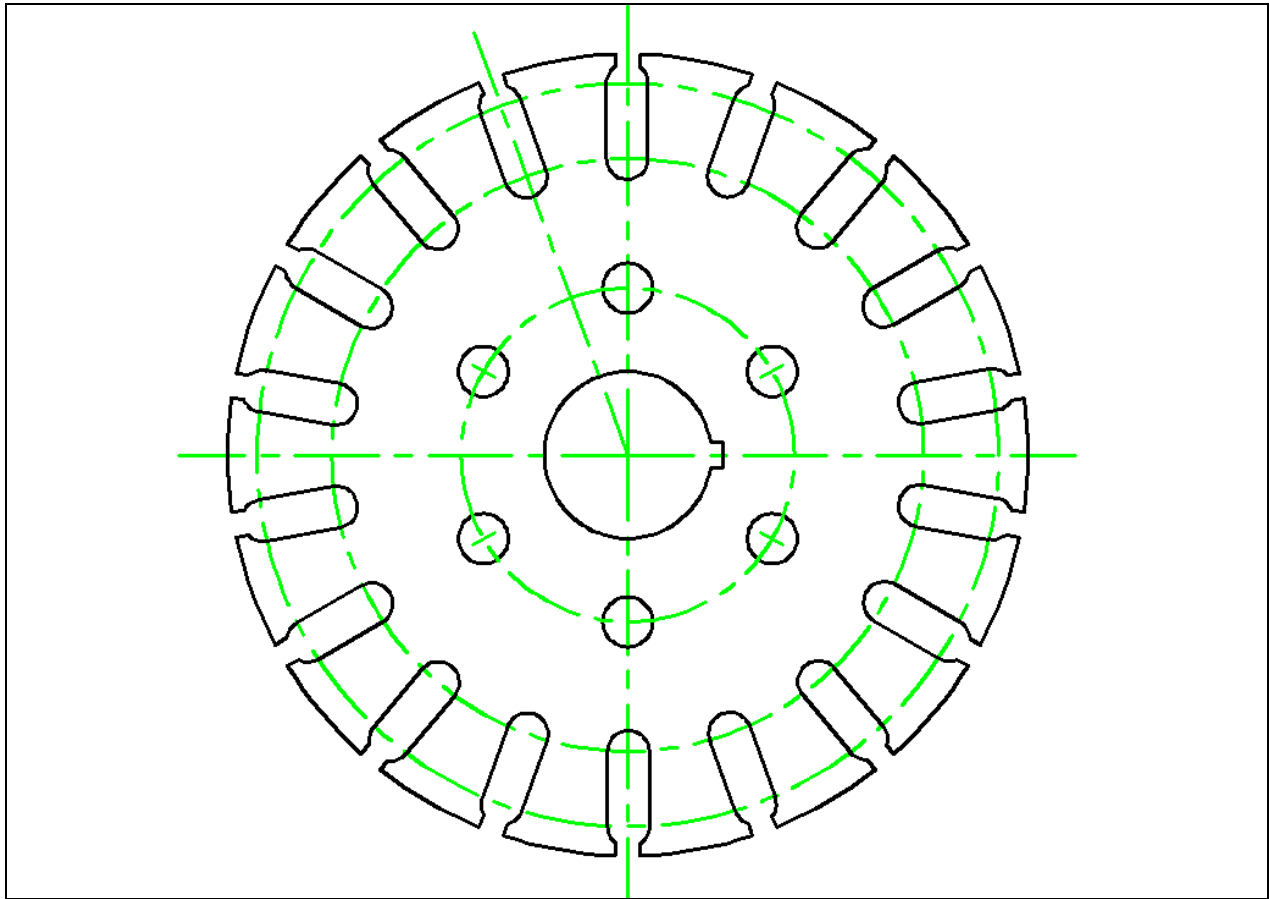


Figure 5.55 – The finished Circular problem

*** 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 in less than 5 minutes to maintain your World Class ranking.**