Chapter

5

The Circular Problem

This chapter will cover:

1. **AccuSnap**
2. **Working with Levels**
3. **Drawing Circles using AccuSnap**
4. **The Scale Command**
5. **The SmartLine Command**
6. **The Move Command**
7. **The Copy Command**
8. **The Fillet Command**
9. **The IntelliTrim Command**
10. **The Array Command**

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.

AccuSnap

_____

Accusnap is a very powerful tool, as you learned in the rectangle problem. For the circular problem, you will need to modify the MultiSnap properties.

Right-click on the MultiSnap button on the Snap Modes toolbar and select properties. Under the MultiSnap 1 button, make sure that Intersection, Midpoint, Key point, and Nearest are checked. Exit the MultiSnap window, and make sure that MultiSnap mode 1 is selected on the Snap Modes toolbar.
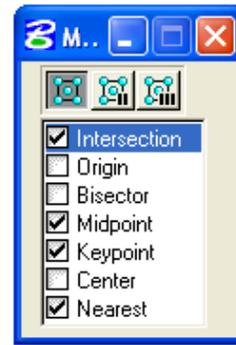

Figure 5.1 – MultiSnap 1 Properties

Working with Levels

_____

When you drew the rectangle problem, you used a single level while drawing the part and then created additional levels later for dimensioning. In the circular problem, you will create levels first.
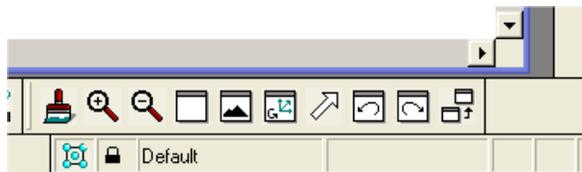

Click the Active Level Indicator, the word "Default," on the Status Bar on the bottom of the screen (Figure 5.2). The Level Manager window will appear (Figure 5.3).

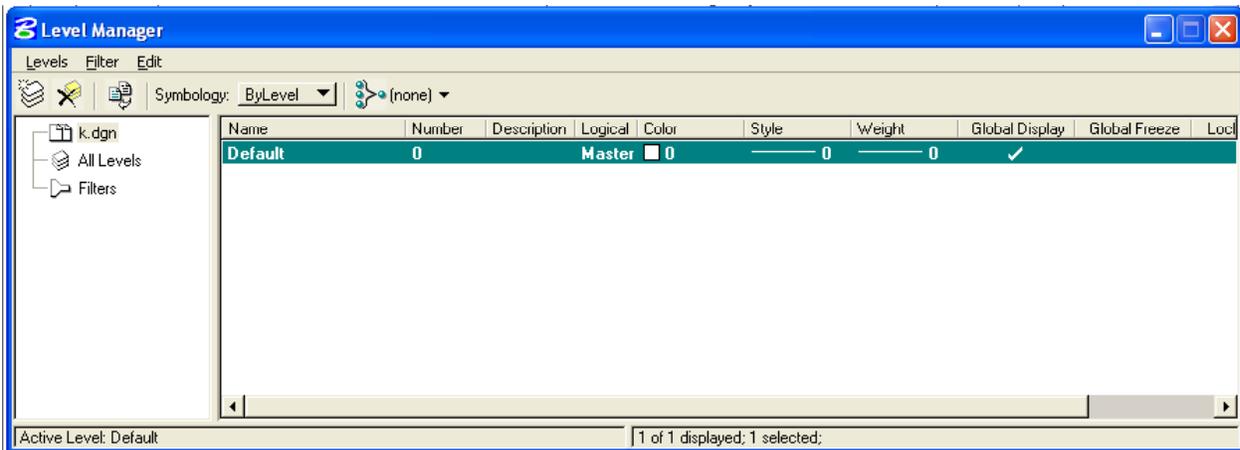**Figure 5.2 – The Active Level Indicator**

Figure 5.3 – The Level Manager Window

Click the New Level icon at the top left corner of the window three times to create three new levels. Rename the levels Center, Dimension, and Text. Change the colors of the dimensions by clicking the color box and selecting a new color. Set Dimension to red, Center to green, and Text to blue, as shown in Figure 5.4.
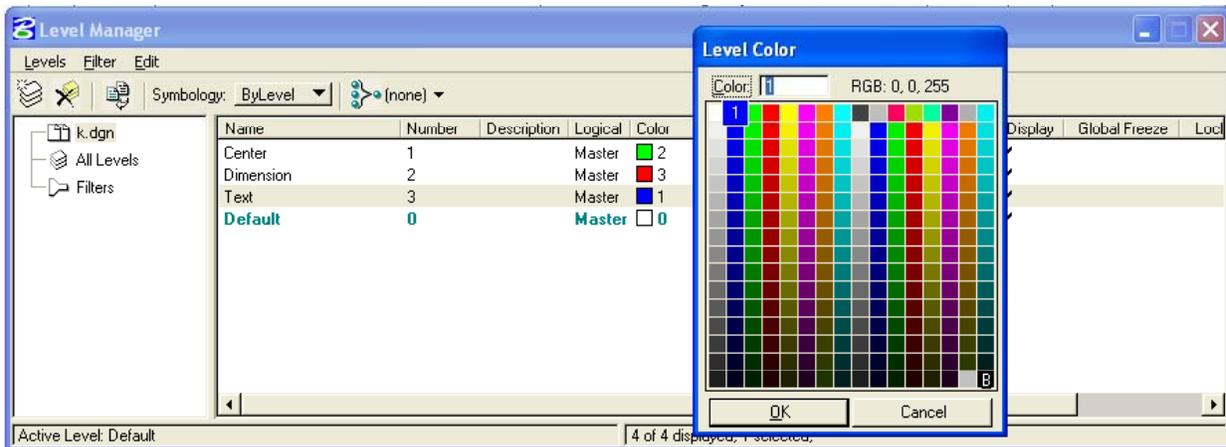

Figure 5.4 - Selecting Colors for New Layers

Change the Line Style of the Center Layer, by clicking the Style column in the center row. Select Line Style 4 from the list and click OK (Figure 5.5). You now have 4 layers, as shown in Figure 5.6.
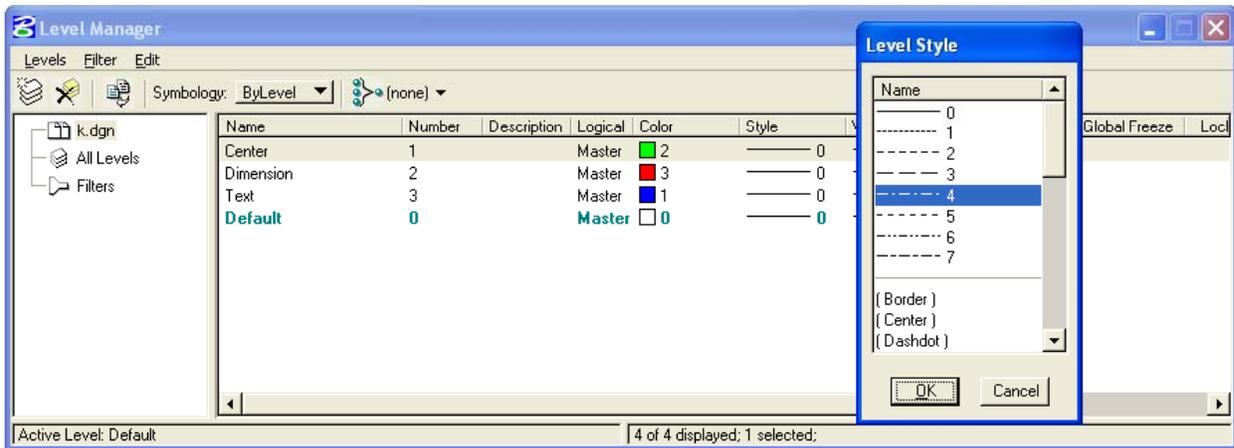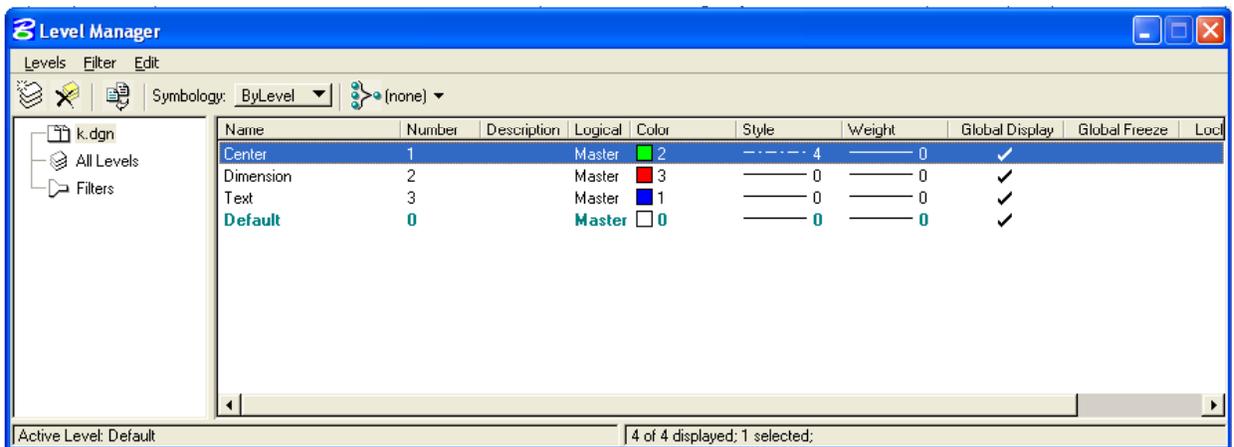
Figure 5.5 - The Line Style Window
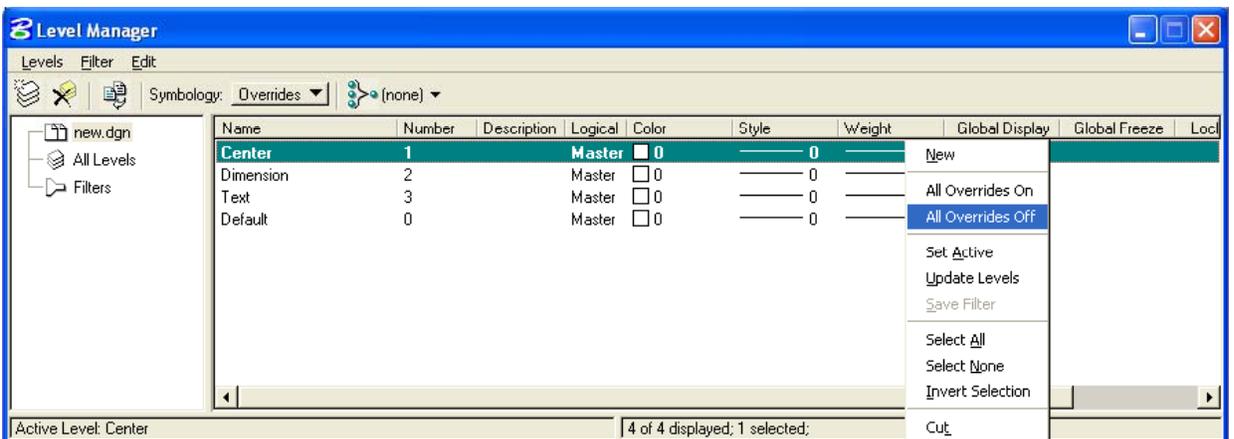

Figure 5.6 – Four Completed Layers

Figure 5.7 – Turning Off Overrides

Under the Symbology drop box, select Overrides. You will see the four levels you just created, but with different settings. Right click on the Weight column and select All Overrides Off. Do the same for the Style and Color columns. Now you will be able to see all level Symbology.

PowerDraft's default workspace displays a grid. Most CAD drafters prefer to work without a grid, as it clutters the workspace. To toggle the grid on/off, hit CTRL+B and uncheck the box beside Grid. Click Apply to save the new settings. You can also maximize the View 1 window to enlarge your workspace.

Drawing Circles using AccuSnap

_____

You need to draw five circles that all have the same center with diameters of 12, 5, 2.5, 0.75, and 0.625 units. You also need to draw a circle with a 4.4375 radius, also with the same center. Select the Default level as the active level in the Active Level combo box.

Select the Circle Command from the Ellipses/Arcs/Circles toolbar and the Place Circle window will appear. Make sure that the Method is set to Center.

Figure 5.8 – The Place Circle Window

Click in the center of the workspace and the AccuDraw compass will appear. Type "6" and Enter to create a 6 unit radius (12 unit diameter) circle (Figure 5.9). Left-click to place the circle. You can also use the Diameter box on the Place Circle window; however, using this feature does not allow you to make use of the AccuDraw tool.
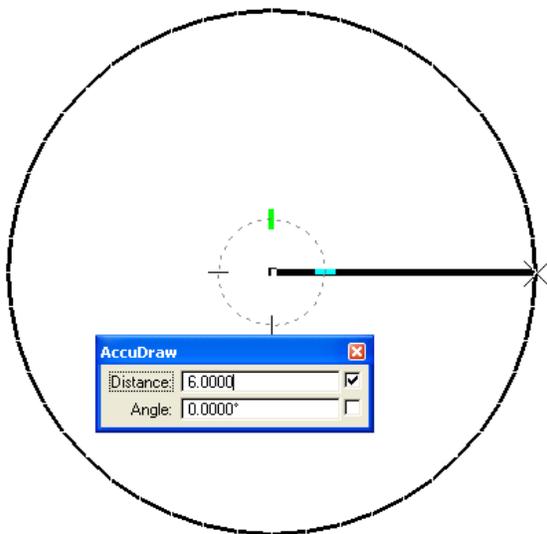
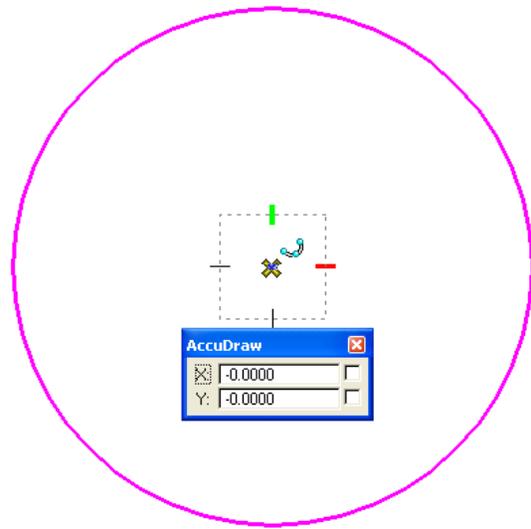**Figure 5.9 – Placing the 12-Unit Circle**   **Figure 5.10 – Selecting the Center**

The AccuDraw compass will change from round to square. Select the center of the circle you just drew. A yellow AccuSnap X should appear in the middle of the circle (Figure 5.10). Click to select that point as the center of the second circle. Type "2.5" and Enter, and then left-click to place the 5-unit diameter circle (Figure 5.11). After you place that circle, select the center once again to create the third circle. Type "2.5/2" and Enter. Notice that the AccuDraw tool automatically calculates the entry: 2.5 / 2 = 1.25 (Figure 5.12). It will create a circle with a radius of 1.25 and a diameter of 2.5. Left-click to place the circle, and select the center again for next circle.
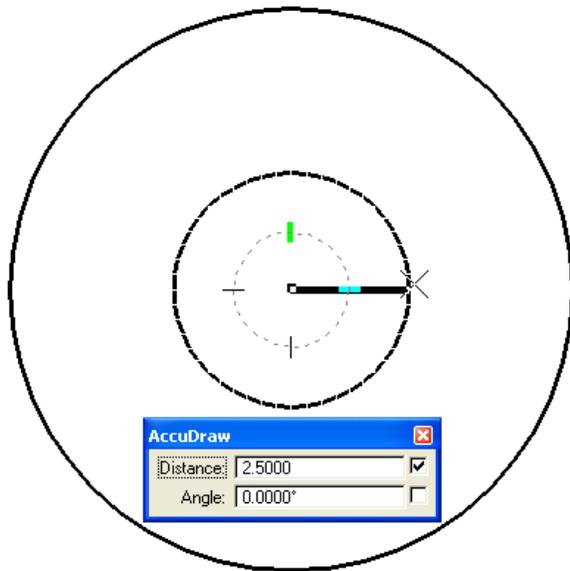

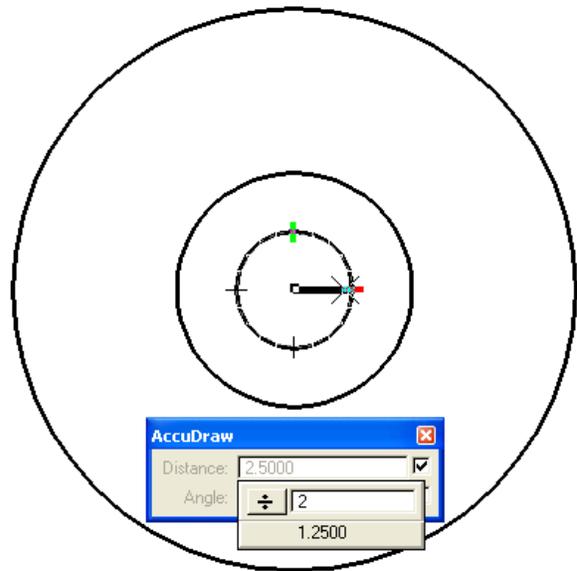**Figure 5.11 – Placing the 5-Unit Circle**

Figure 5.12 – Placing the 2.5-Unit Circle

Using the same techniques described above, place the final two circles: 0.75 and 0.625 (Figure 5.13 and 5.14). Use the AccuDraw tool to calculate the radius by dividing each diameter by 2. Finally, place the 4.4375 radius circle (Figure 5.15). All the circles should share the same centerpoint.
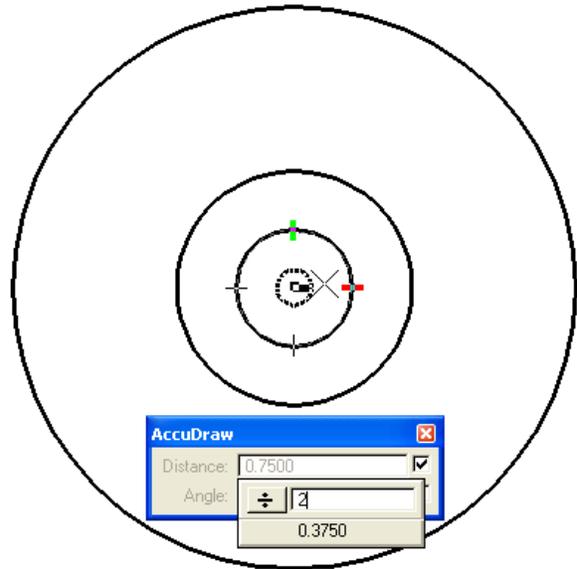

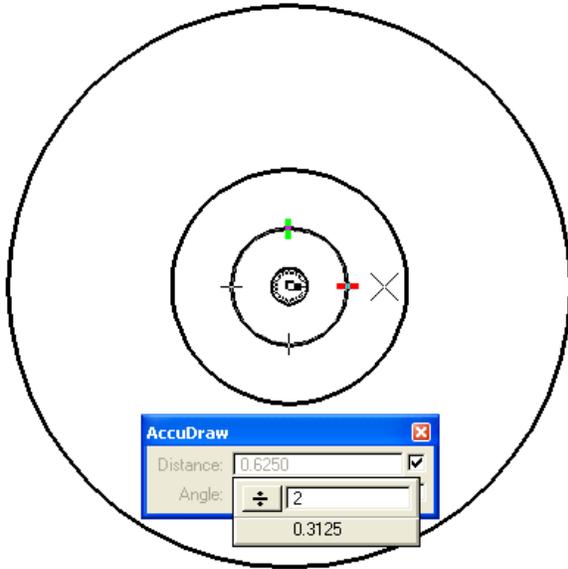**Figure 5.13 – Placing the 0.75-Unit Circle**
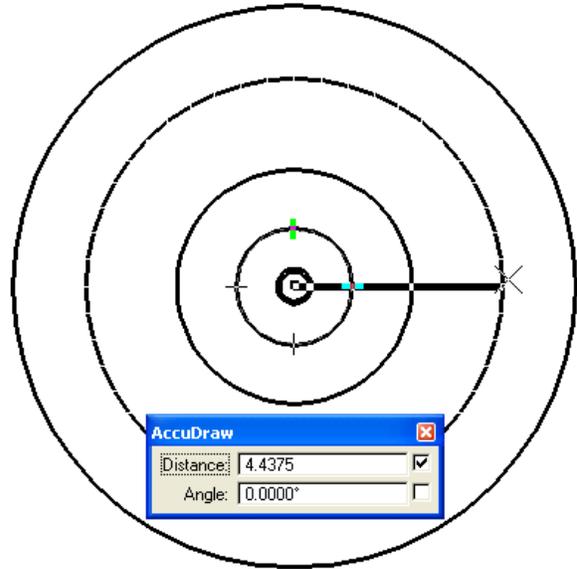
Figure 5.14 – Placing the 0.625-Unit Circle

Figure 5.15 – Placing the 4.4375-Radius Circle

Using the Scale Command to Add Another Circle

Unlike AutoCAD, PowerDraft does not have an Offset tool; however, it does have a tool that can be used, in some cases, to create the same effect. This tool is called the Scale tool.

Select it from the Manipulate toolbar and the Scale widow will appear (Figure 5.16). On the Scale widow, click the Method box and select 3 Points. Check the Proportional and Copies boxes, and make sure the number of Copies is set to 1.
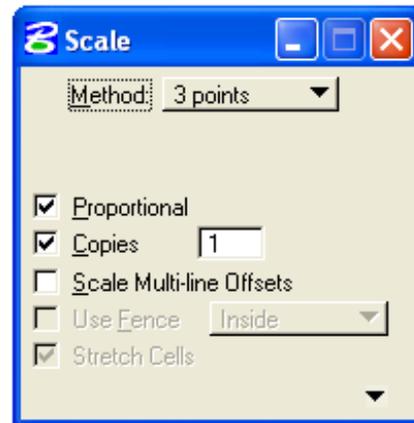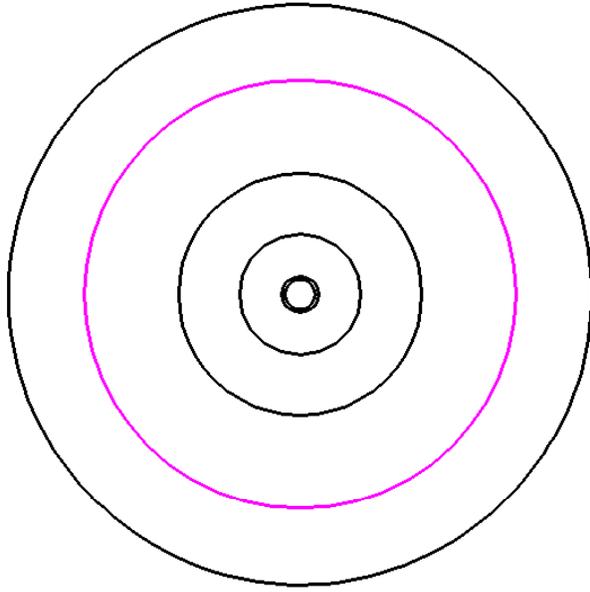

Figure 5.16 – The Scale Window

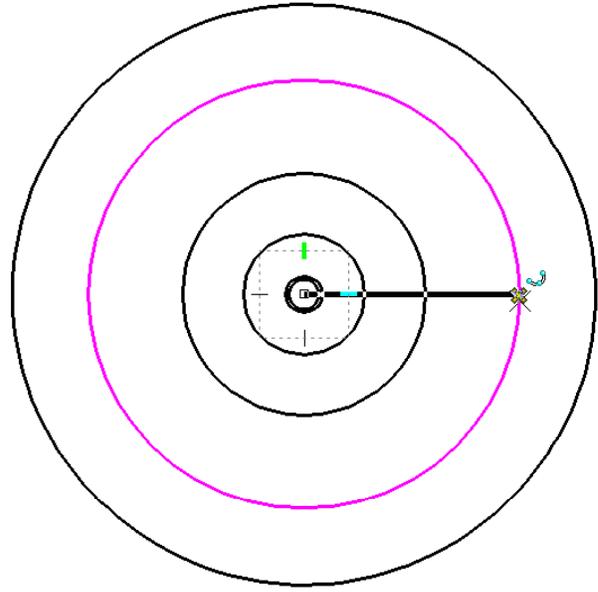**Figure 5.17 – Selecting the Circle to Scale**

Figure 5.18 – The Centerpoint and Reference Point

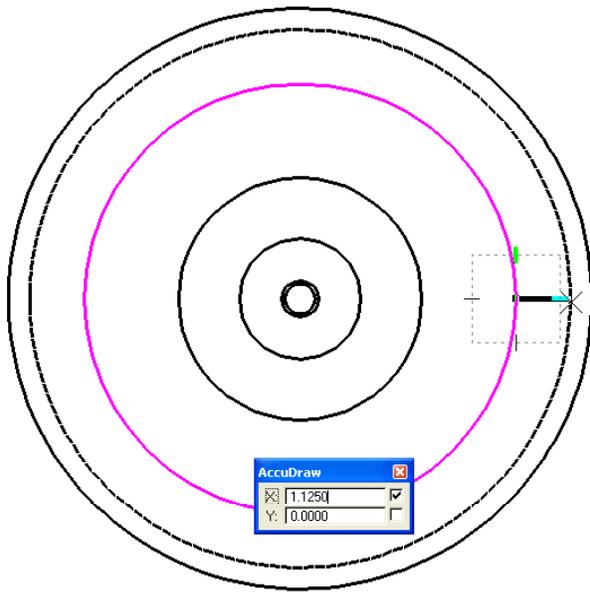The first part of the 3 Point Scale tool is to select the object you wish to scale, in this case the 4.4375 radius circle, the second largest one on the screen. Select it and it will become highlighted (Figure 5.17). Next, select the point around which to scale the new circle. Select the centerpoint of all the circles, and the AccuDraw compass will appear. Click the right quadrant of the 4.4375 circle (Figure 5.18). You should see a yellow AccuSnap X appear on the side of the circle. The AccuDraw compass will move to that point. Move the cursor away from the center, so that the new circle grows in size. Type 1.125 and hit Enter. A new circle 1.125 units larger than the 4.4375 circle will appear on the screen (Figure 5.19). Left click to place the new circle. Right click to reset the command.
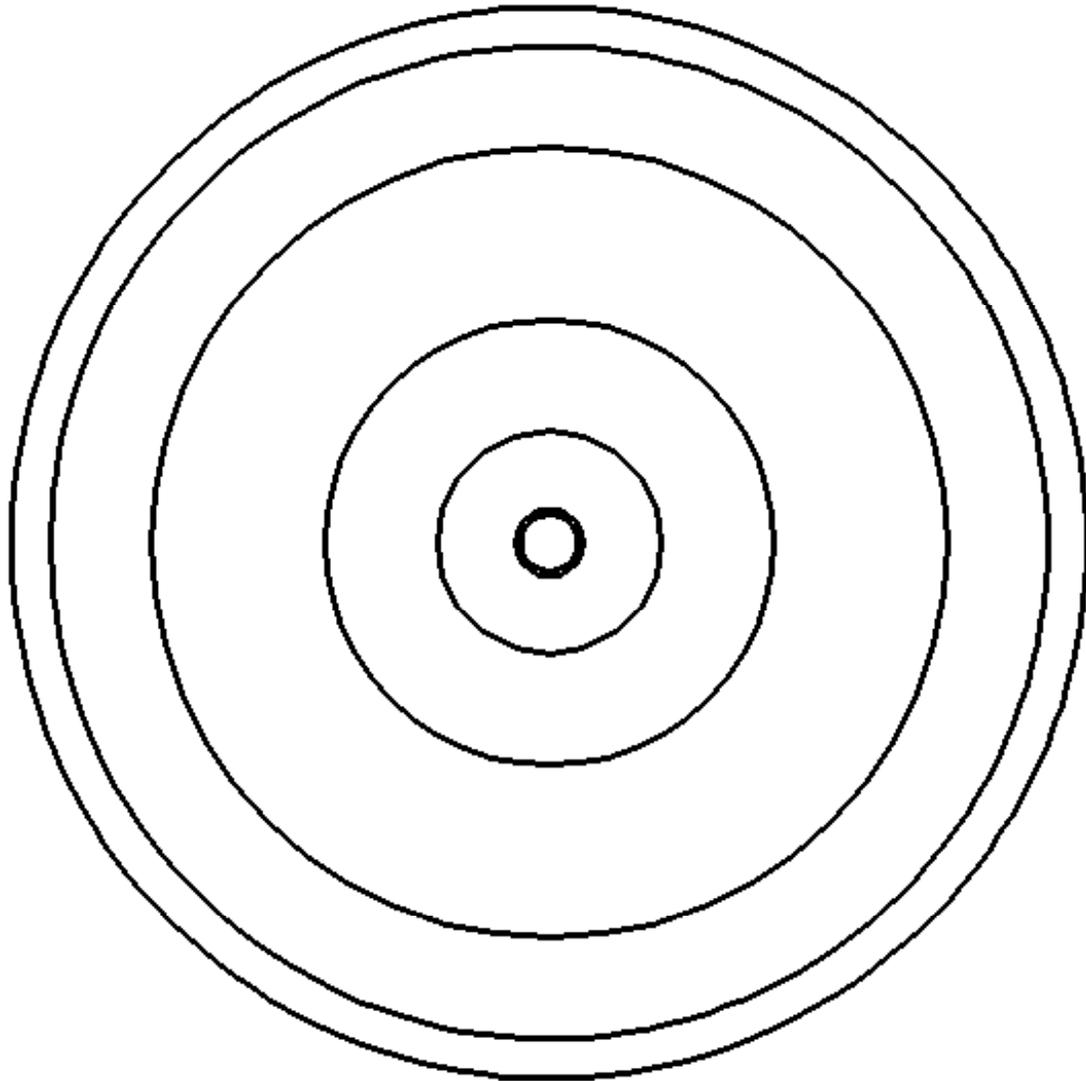

Figure 5.19 – Scaling the New Circle

Figure 5.20 – Seven Finished Circles

* World Class CAD Challenge 02-10 * - Create a New file, create the three new levels, set the MultiSnap 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 SmartLine Command

_____

As opposed to the rectangular problem, this problem is being drawn on a polar axis. Therefore, the centerlines do not need to cross in the center of the part. Rather, for the moment, the centerline will extend along the 0° point. Later you will use the array tool to create more centerlines.

You should zoom in to the part to get a better view of the next few steps. Select the Window Area tool from the 2D View Control and the cursor will become a large cross that extends off of the workspace. Start by clicking by the top left corner of the part, but making sure that the outer circle stays inside the cross lines (Figure 5.21). Drag the box down and to the right, with the bottom border just below the smallest circles, but the right border including the entire part (Figure 5.22). Notice that the box always stays the same ratio, so you may have to be creative when sizing the box. Click again to zoom in to this window.
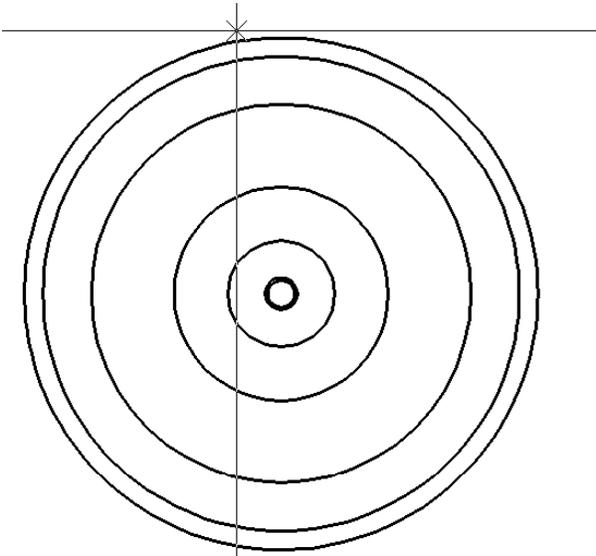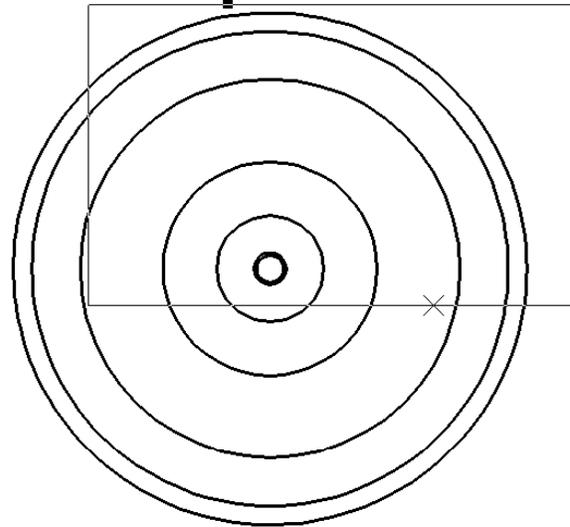
Figure 5.21 – Starting the Zoom Window

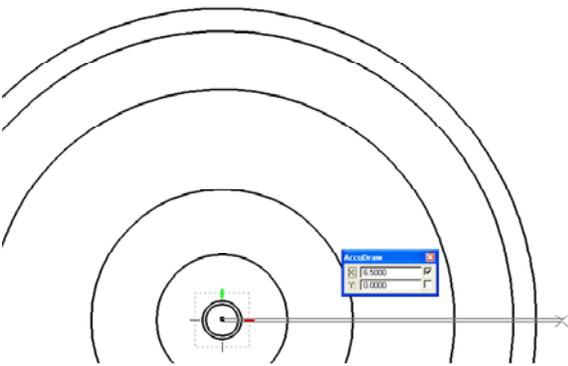**Figure 5.22 – Selecting the Window Area**

Select the SmartLine command from the Linear elements toolbar. Make sure that on the Place SmartLine window the Segment Type is set to Lines and the Vertex Type is set to Sharp. Select the center of the part as the starting point of the SmartLine and the AccuDraw compass will appear. Type "6.5" and Enter, and then click to place the centerline (Figure 5.23).

Figure 5.23 – The Horizontal Centerline

Using the Move Command

_____

You need to move the 0.625 and 0.75 circles away from the center of the part. Select the Move command from the Manipulate toolbar.

Select the center of the part by clicking the yellow AccuSnap X, and any of the objects will be initially selected. You can tell which object is currently selected by keeping the cursor on the center of the part. The selected object will turn green. Right-click to cycle through all the objects with that same centerpoint until the 0.75 circle turns green (Figure 5.24). You can then move the circle up to the north quadrant of the 2.5 diameter circle, using the AccuDraw compass to stay in line with the center of the part. A yellow Accusnap X will appear at the top of the circle (Figure 5.25). Left-click to move the 0.75 circle in its new position. Right-click to restart the command.
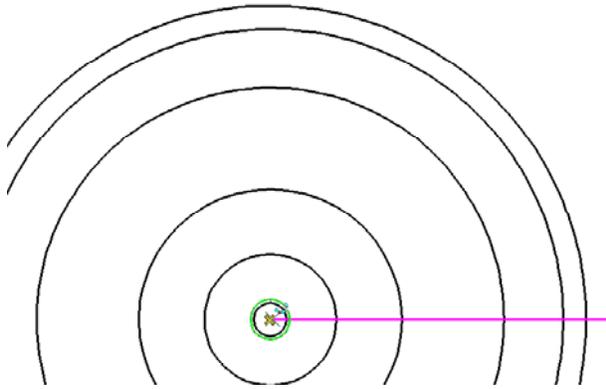

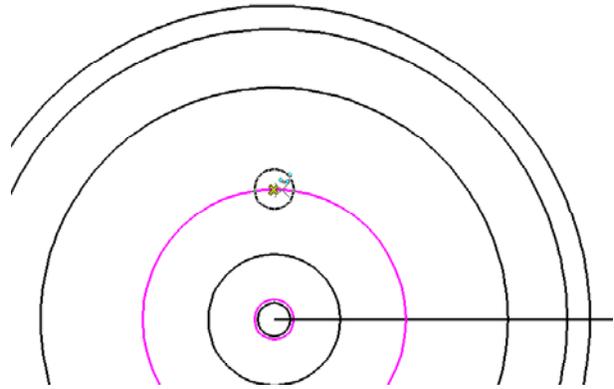**Figure 5.24 – The Selected 0.75 Circle**   **Figure 5.25 – Moving the 0.75 Circle**

To move the 0.625 circle, select it using the same technique mentioned above. Select the center of the part using the yellow Accusnap X, and an object will turn green. Right-click to cycle through all of the objects that share that center until the 0.625 circle is highlighted (Figure 5.26). Using the AccuDraw compass, move the circle straight up to the north quadrant of the 4.4375-radius circle (Figure 5.27). Left-click to move the 0.625 circle, and right click to reset the command.
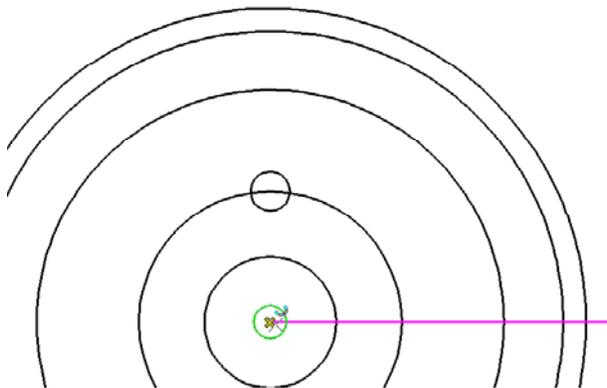
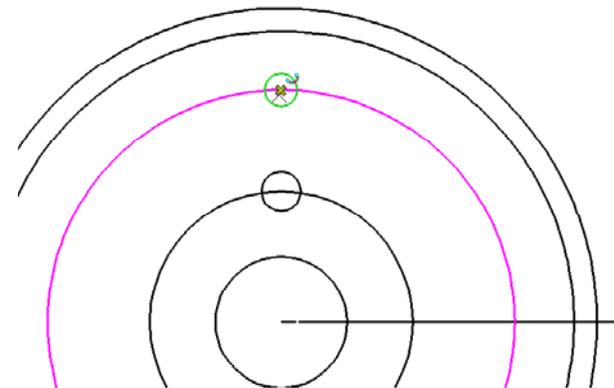

Figure 5.26 – The Selected 0.625 Circle   **Figure 5.27 – Moving the 0.625 Circle**

Using the Copy Command

_____

Now you need to place a copy the 0.625 circle above it on the north quadrant of the scaled circle. Select the Copy command from the Manipulate toolbar. Select the center of the 0.625 circle. The 4.4375-radius circle will probably be highlighted instead of the one you wish to copy. Right-click to change the selected object to the 0.625 circle, which will turn green (Figure 5.28). Using the AccuDraw compass and the yellow Accusnap X, move the new copy of the circle up to the north quadrant of the scaled circle (Figure 5.29). Left-click to place the new circle, and right-click to reset the command.
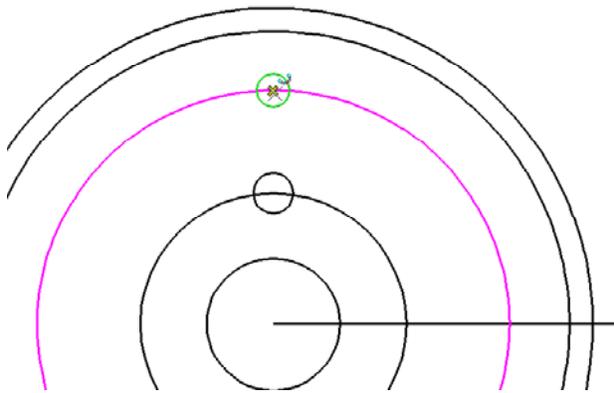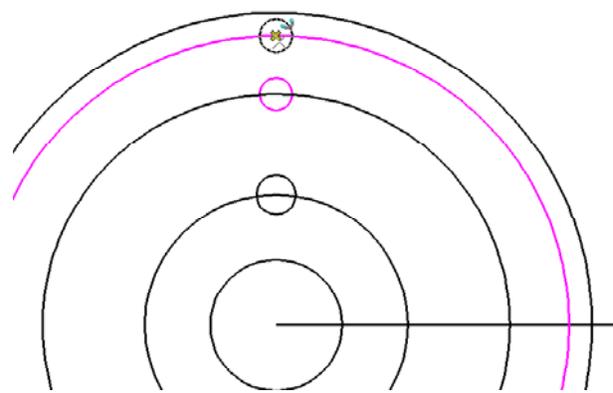
Figure 5.28 – The Selected 0.625 Circle   **Figure 5.29 – Copying the 0.625 circle**

* World Class CAD Challenge 02-11 * - Create a New file, create the three new levels, set the MultiSnap 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 SmartLine Command to Add Detail

_____

Five lines needed to be added to the drawing to help create detail and aid in the positioning of other objects. Select the SmartLine command from the Linear Elements toolbar and the Place SmartLine window will appear. Make sure that the Segment Type is set to Lines and the Vertex Type is set to Sharp.
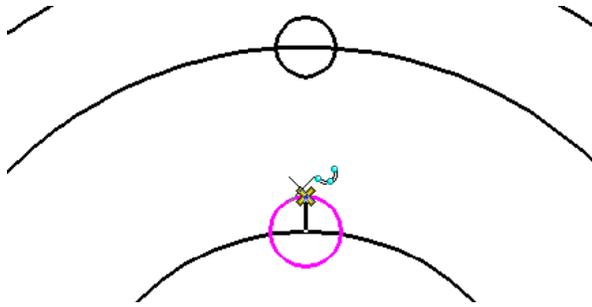
Figure 5.30 – Drawing the Center mark

Create the first line by connecting the north quadrant of the 2.5-diameter circle with the north quadrant of the 0.75 circle (Figure 5.30). This line will become a center mark later in the drawing. Right-click to reset the command.
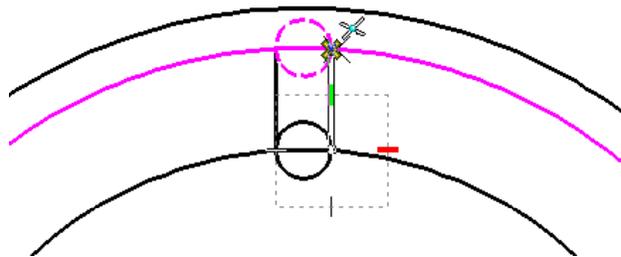
Next, connect the east and west quadrants of the two 0.625 circles by placing two vertical lines on either side (Figure 5.31). Right-click after placing the first line to reset the command before placing the second. AccuSnap should make this process very simple. Right-click again to move on to the next line.

Figure 5.31 – Drawing the Slot Lines

The next line will not have a defined length, as it will just be a construction line for now. Select the center of the upper 0.625 circle and draw a line that extends past the outer edge of the part (Figure 5.32). Right-click after placing the line to reset the command.
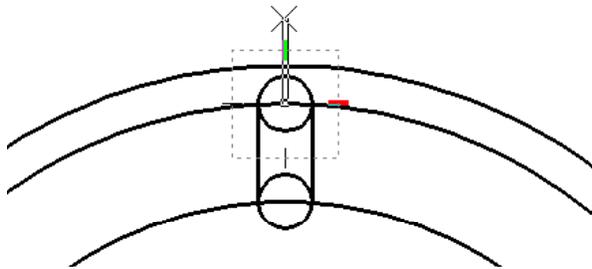

Figure 5.32 – Drawing the Centerline

Like before, the last line will not have a defined length, either. Select the west quadrant of the 2.5-diameter circle and extend the line vertically upwards (Figure 5.33). Do not exceed the 4.375-radius circle. Right-click to reset the command after you are finished.
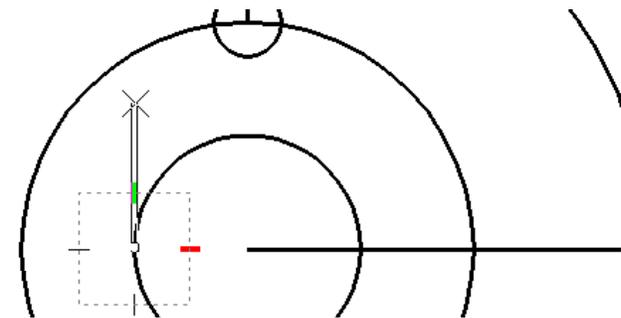

Figure 5.33 – Drawing the Keyway Line

Using the Move Command to Add Detail

Some of the lines you just placed need to be moved. Select the Move command from the Manipulate toolbar.

Select the vertical line that extends from the west quadrant of the 2.5-diameter circle and the AccuDraw compass will appear. Move the line to the right and type "2.687" and Enter (Figure 5.34). Left-click to move the line 2.687 units to the right and right-click to reset the command.
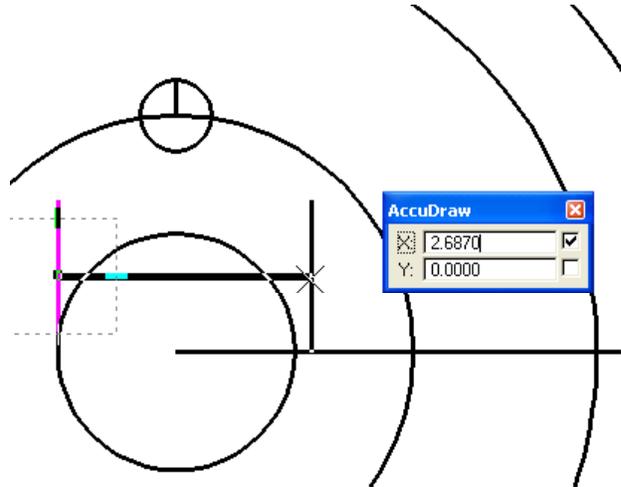
Figure 5.34 – Moving the Keyway Line

Center the line you placed in the 0.75 circle by selecting it at its midpoint. Using the yellow Accusnap X, move it to the north quadrant of the 4.375-radius circle so that it is in the center of the 0.75 circle (Figure 5.35). Left-click to place the line and right-click to reset the command.
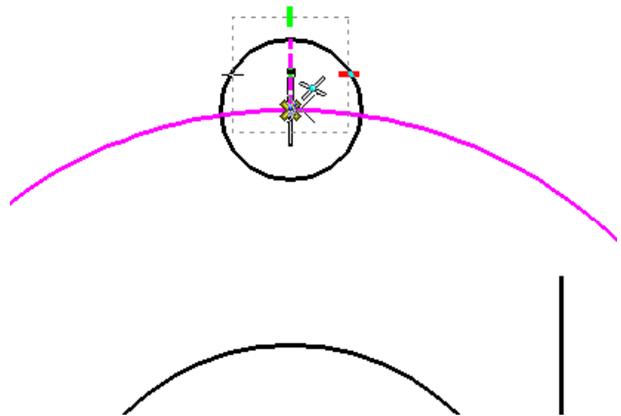
Figure 5.35 – Moving the Center Mark

Using the Copy Command to Add Detail

Select the Copy tool once again from the Manipulate toolbar. You will need to create four new lines, but luckily for you they are all the same distance from the originals. The AccuDraw feature will make this task very simple.

Select the 0º centerline that extends from the center of the part and the AccuDraw compass will appear. Drag the line upwards and type "0.1875" and Enter (Figure 5.36). Left-click to place new line 0.1875 units above the centerline. Right-click to reset the command.
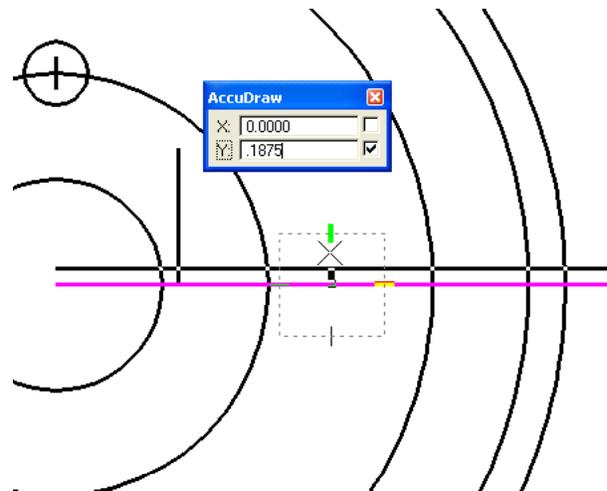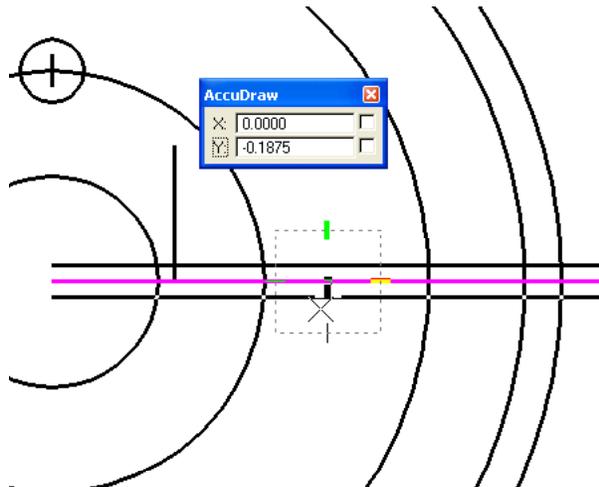

Figure 5.36 – Copying the Centerline

Believe it or not, that was the hard part. Select the centerline again, but this time drag the line straight down. When the cursor approaches 0.1875 units away from the AccuDraw compass, the line should lock into place (Figure 5.37). AccuDraw remembers the last entry and will automatically snap to that distance until a new one is entered. In this case, 0.1875 is the distance you want, so left-click to place the line and right-click to reset the command.

Figure 5.37 – Copying the Centerline

Use the same strategy to place new lines 0.1875 units on either side of the vertical centerline extending from the center of the top 0.625 circle. Select the original line and drag it to the right until it snaps 0.1875 units away (Figure 5.38). Left-click to place the line and right-click to reset the command. Do the same on the left side of the original line to finish the Copy command (Figure 5.39).
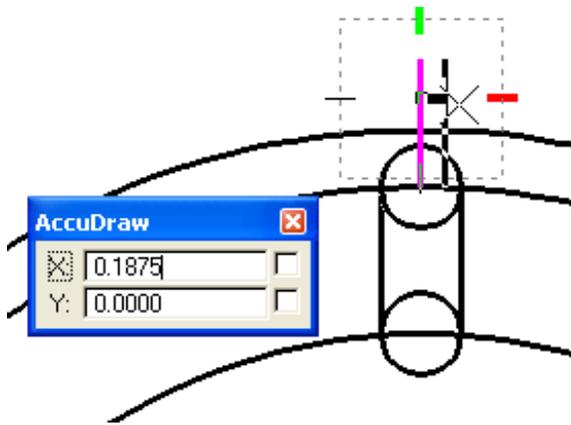
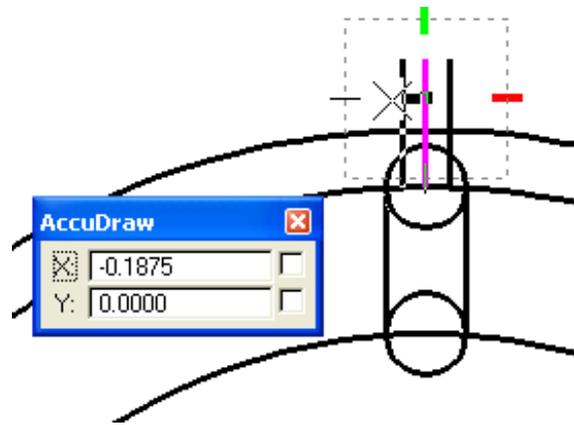**Figure 5.38 – Copying the Centerline**   **Figure 5.39 – Copying the Centerline**

* World Class CAD Challenge 02-12 * - Create a New file, create the three new levels, set the MultiSnap 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

_____

The Fillet command can be used to create sharp as well as rounded edges. For this next action you will use the Construct Circular Fillet tool with a 0º radius.

Select the Construct Circular Fillet command from the Modify toolbar. Type "0" for the Radius on the Construct Circular Fillet window (Figure 5.40). Make sure Truncate is set to Both.

Figure 5.40 – The Fillet Window
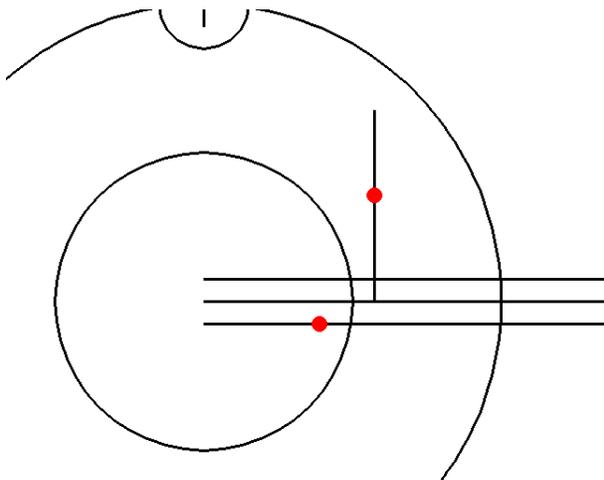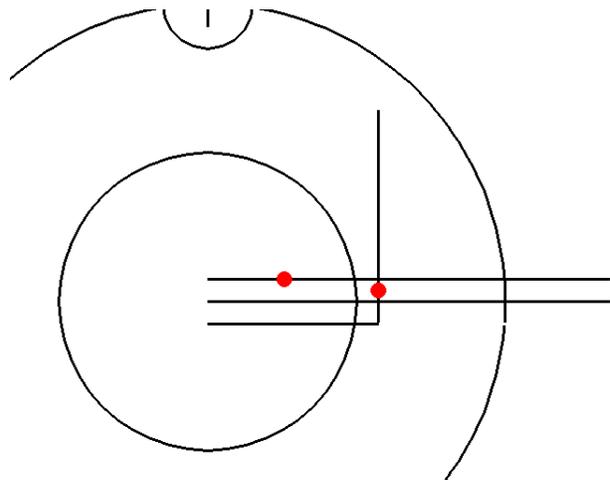

Figure 5.41 – The First Fillet   **Figure 5.42 – The Second Fillet**

5-16

To fillet the first corner, select the bottom 0º centerline (close to the center of the part; remember to select the objects to be filleted near the portion of it you wish to remain on the drawing) and then the vertical line you just moved 2.687 units to the right (see red dots in figure 5.41). The vertical line will extend to meet the bottom line, and the bottom line will be trimmed to meet the vertical line. Left-click to apply the fillet.

Select the vertical line below the upper centerline to start the second fillet. Next, click the top centerline on the left side of the vertical line (see red dots in figure 5.42). The two lines will meet and trim past their intersection. Left-click to apply the fillet.

Using the IntelliTrim Command to Erase Construction Lines

Select the IntelliTrim command from the Modify toolbar. You have used this tool before, but only in Advanced mode. For the first trim, you will use Quick mode. Select Quick as the Mode from the IntelliTrim window.

First you must select the element to trim to, in this case, the 2.5-diameter circle. Now you can draw lines crossing over the objects you wish to trim. Click below the bottom horizontal line and bring the line up so that it crosses only that line (Figure 5.43). The line will then be trimmed to the circle. Draw another line crossing the top horizontal line and it too will be trimmed to the circle (Figure 5.44). Right-click if you need to reset the command.
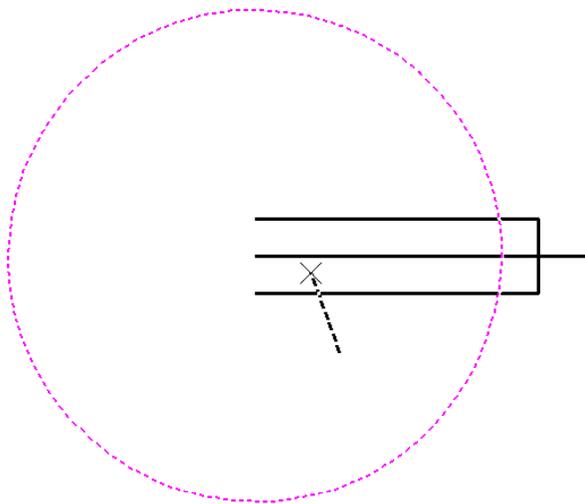
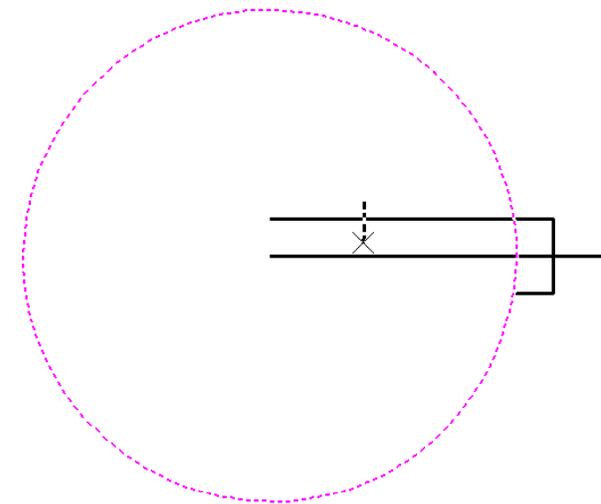**Figure 5.43 – Trimming the Bottom Line**   **Figure 5.44 – Trimming the Top Line**

If you feel more comfortable using the Advanced mode in this situation, feel free. If you have been practicing, you may even decide to use the Element Selection tool and shorten the two lines using handles. You should at this point be developing a personal drawing style based on your tool preferences and efficiency. Be aware, however, that every tool has a purpose, and it is worthwhile to familiarize yourself with every command even if you find another tool more useful.

For the next trim you will need to switch to Advanced mode on the IntelliTrim window. Make sure the tool is set to Select Cutting Elements (Figure 5.45).
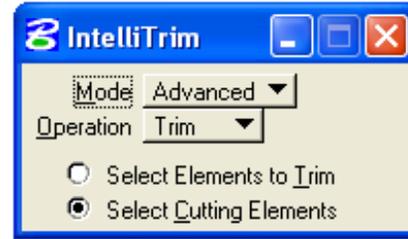
First, identify the cutting elements by selecting the top and bottom horizontal lines (Figure 5.46). Right-click to continue the command. Next, select the element to be trimmed, in this case, the circle (Figure 5.47). After selecting it, right click to move on. Finally, select a point near the portion of the element you wish to keep. Click near the center of the part (Figure 5.48). You should see the part of the circle between the two lines disappear. Right-click once more to apply the trim.
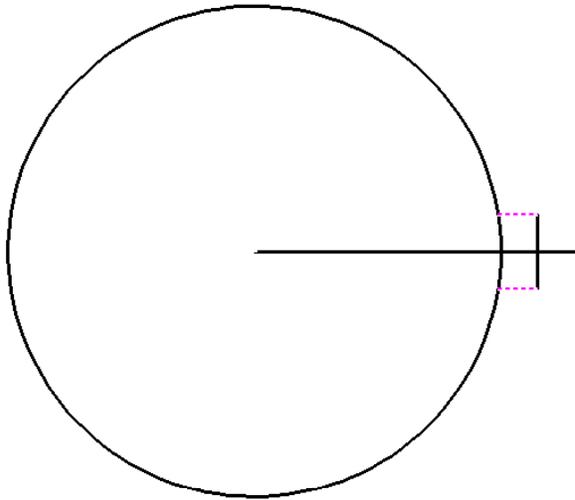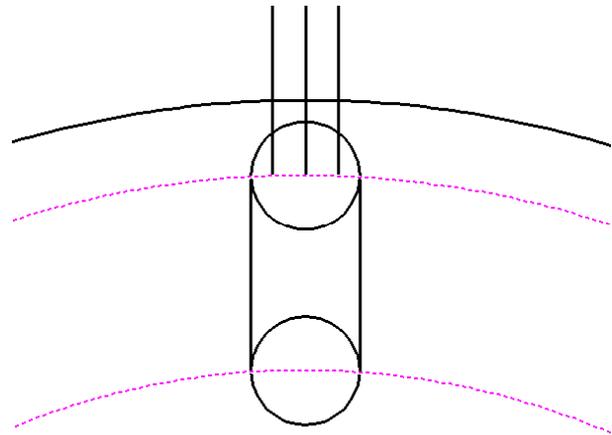

Figure 5.46 – Selecting Cutting Elements
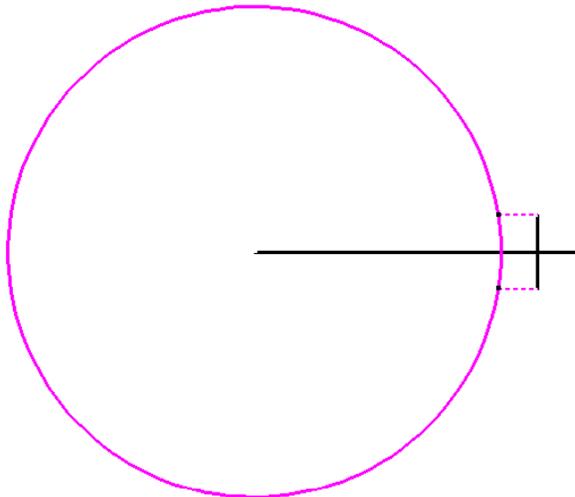
Figure 5.47 – Selecting Elements to Trim
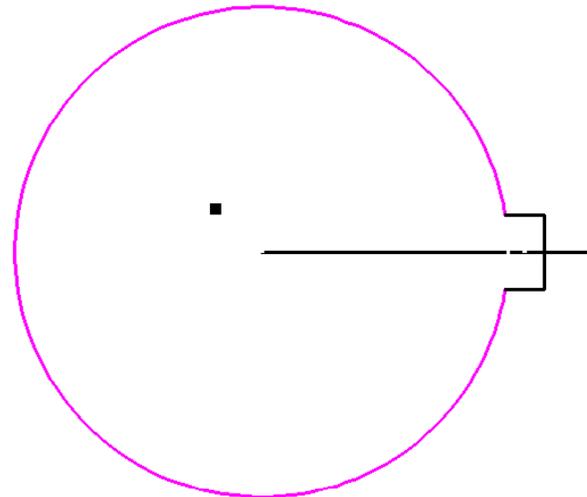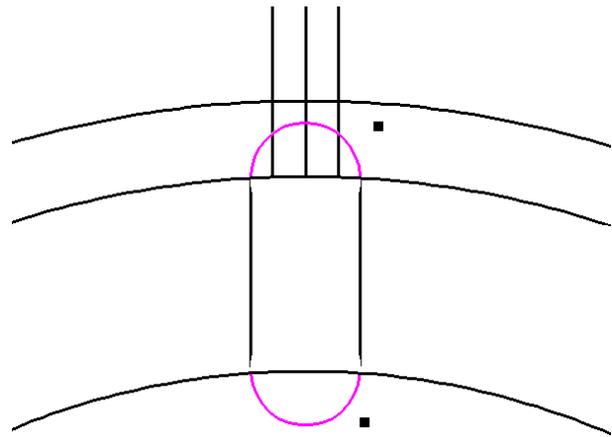

Figure 5.48 – Selecting Portions to Keep

The slot at the top of the part is almost finished, but many of the construction elements need to be trimmed away. Use the scroll button on the mouse to zoom into the slot. Remain in the Advanced mode of the IntelliTrim tool for these next three trims.

Begin the first trim by selecting the 4.4375-radius circle and the scaled circle as the cutting elements (Figure 5.49). Right-click to move on to selecting the elements to trim, which are the two 0.625 circles (Figure 5.50). Select them and right-click again to go on to the third part of the IntelliTrim command. Finally, place points near the portions of the trimmed elements you want to remain, in this case, the top and bottom of the slot (Figure 5.51). Right-click once more to apply the trim.

Figure 5.49 – Selecting Cutting Elements
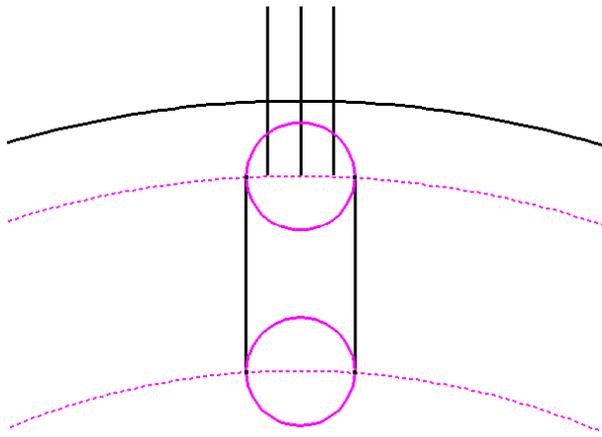
Figure 5.50 – Selecting Elements to Trim

Figure 5.51 – Selecting Portions to Keep

For the second trim, select the 12-unit circle and the remaining hemisphere of the top 0.625 circle as the cutting elements (Figure 5.52). Select the horizontal lines to the left and right of the centerline to be trimmed (Figure 5.53). Place a point in-between the 12-unit circle and the 0.625 circle to designate the parts of those lines you wish to keep (Figure 5.54). Right-click to apply the trim.
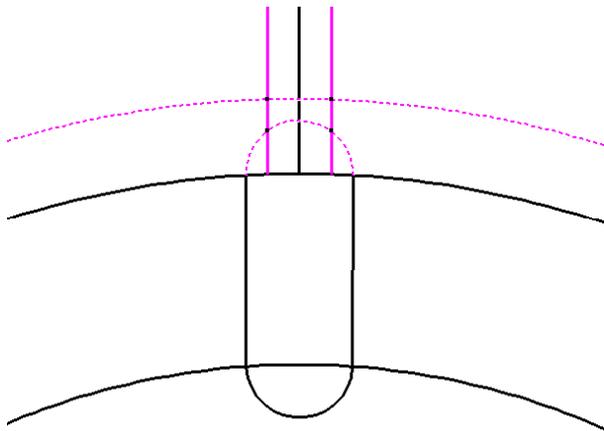

Figure 5.52 – Selecting Cutting Elements

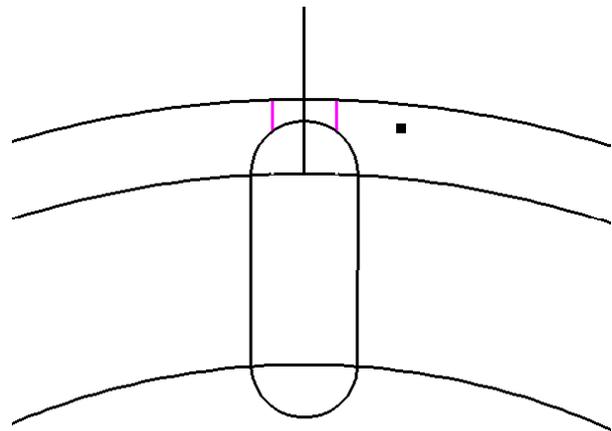**Figure 5.53 – Selecting Elements to Trim**

Figure 5.54 – Selecting Portions to Keep

For the last trim, reverse the cutting elements and the elements you to be trimmed. Select the two horizontal lines you just trimmed as the cutting elements (Figure 5.55). The 12-unit circle and the top 0.625 hemisphere will be the elements to be trimmed (Figure 5.56). Place a point to the left or right of the slot to retain the outer circle and the top of the slot (Figure 5.57). Right-click to apply the trim.
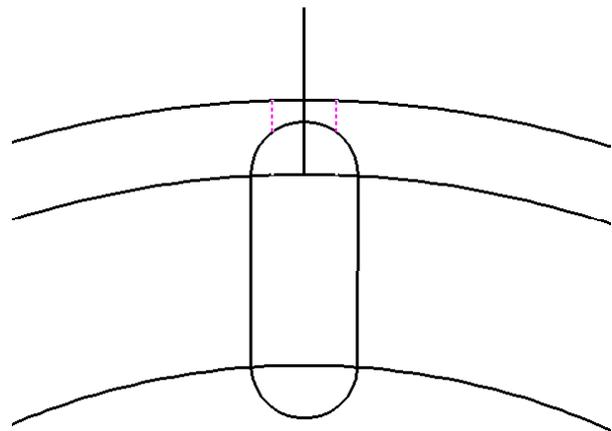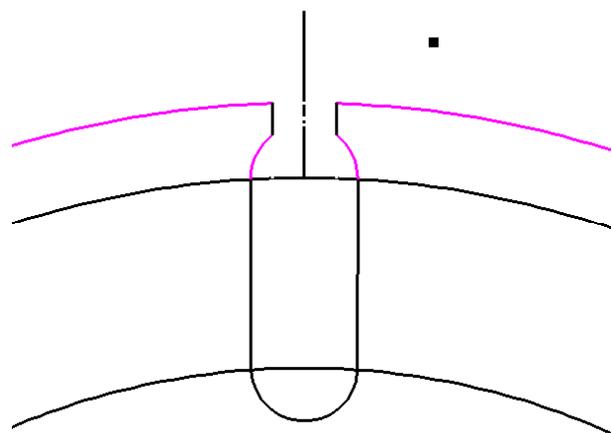

Figure 5.55 – Selecting Cutting Elements
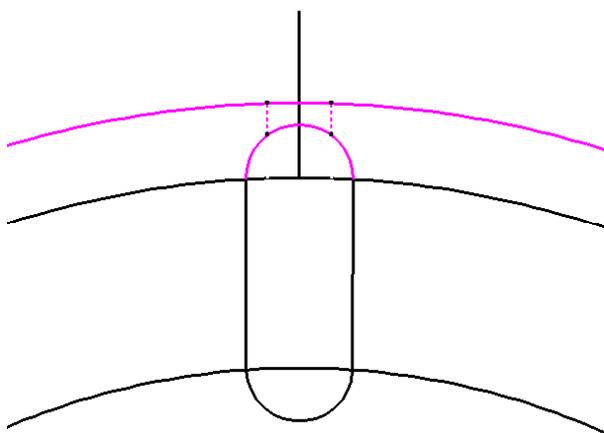
Figure 5.56 – Selecting Elements to Trim

Figure 5.57 – Selecting Portions to Keep
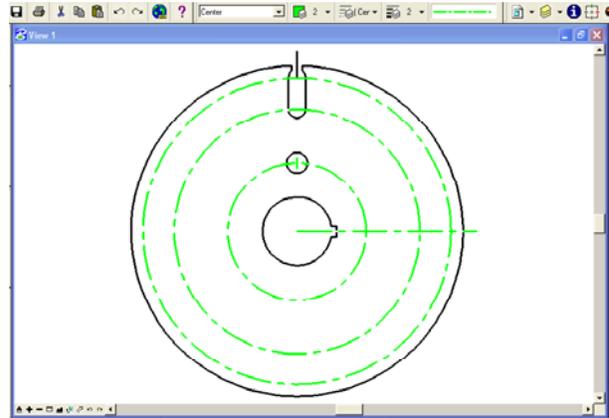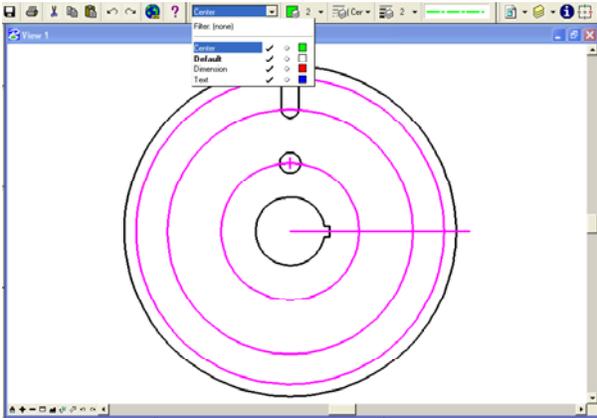
Changing an Object's Level

Figure 5.58 – The Active Layer Combo Box   Figure 5.59 – The Center Level

Using the Element Selection tool from the Element Selection toolbar, hold CTRL and select the scaled circle, the 4.4375-raidus circle, and the 5-unit circle. While still holding CTRL, select the 0º centerline and the 0.75 circle's centerline. Click the Active Level combo box and select the Center level (Figure 5.58). The circles and lines will turn green and become dashed, just as you set the Symbology for the Center level (Figure 5.59).

Using the Array Command

The rectangular problem introduced the Array command by incorporating a rectangular array. There are four arrays to create in the circular problem, but they are polar rather than rectangular.

Select the Construct Array command from the Manipulate toolbar and the Construct Array window will appear. Change the Array Type to Polar. For Items enter "6," and for the DeltaAngle type "360/6" and Enter. 60º should appear in the box. Finally, check the Rotate Items box (Figure 5.60).
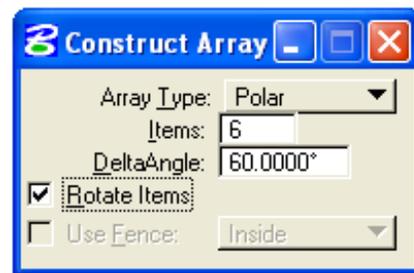

Figure 5.60 – The Construct Array Window

This array will include the 0.75 circle and its centerline. Click the 0.75 circle and then click the center of the drawing (Figure 5.61). Five new circles will appear encircling the center of the part. To array the centerline, click it and then the center of the drawing, just like before (Figure 5.62). Five new lines will appear, each marking the center of a 0.75 circle.
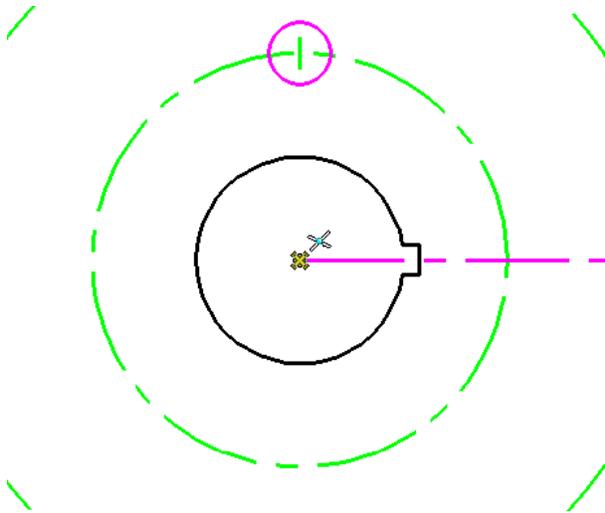
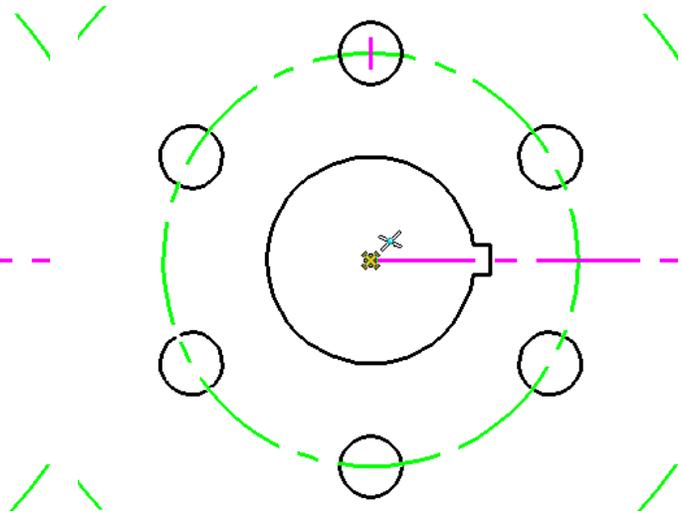**Figure 5.61 – Arraying the 0.75 Circle**

Figure 5.62 – Arraying the Centerline

Deleting Elements

_____

Before continuing with the Array command, you need to remove three lines from the drawing. Select the Element Selection tool from the Element Selection toolbar. Holding CTRL, click the two vertical centerlines from the top and bottom 0.75 circles and also the vertical construction line above the slotted hole (Figure 5.63). Hit DELETE to remove these elements from the drawing (Figure 5.64).
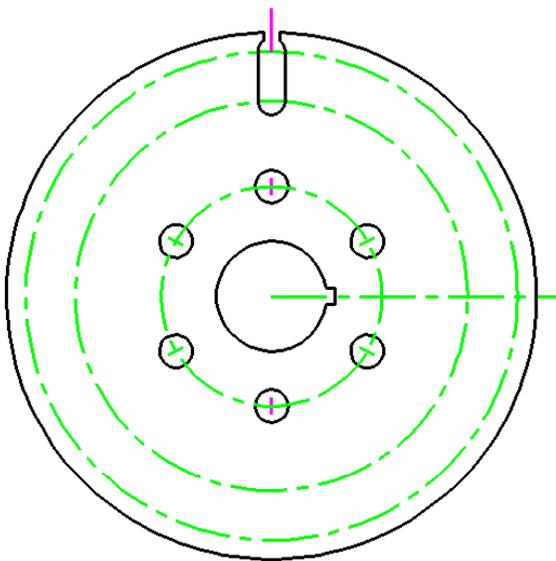

Figure 5.63 – Selecting Three Lines
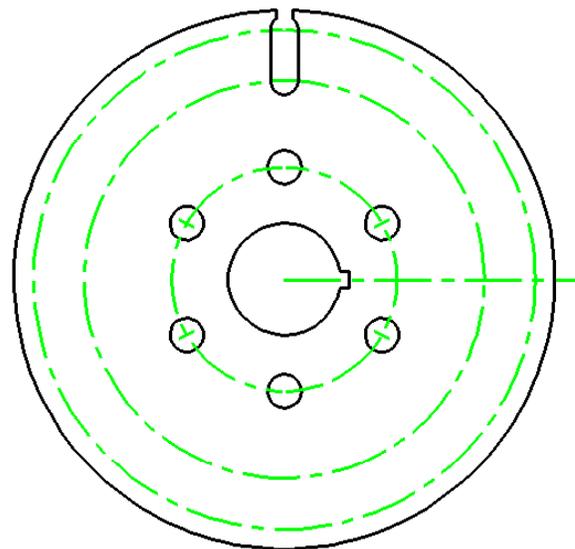
Figure 5.64 – Three Lines Deleted

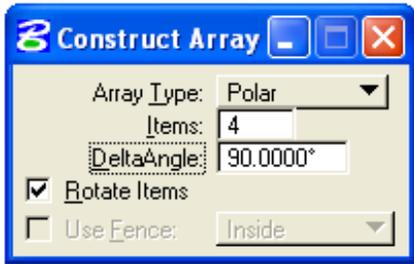# Using the Array Command to Create More Circular Patterns

Figure 5.65 – The Construct Array Window

Select the Construct Array tool again from the Manipulate toolbar. This time, change the number of Items to 4 and the DeltaAngle to 90º on the Construct Array window (Figure 5.65). Select the 0º centerline as the object to array, and then select the center of the part as the point of rotation (Figure 5.66). Three new lines will appear, creating a cross-hair look on the drawing (Figure 5.67).
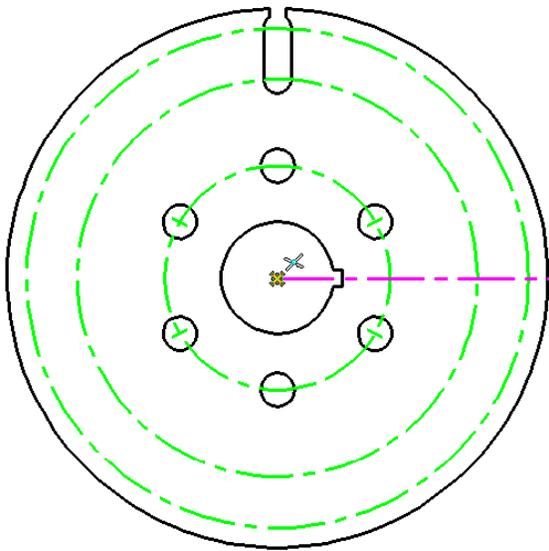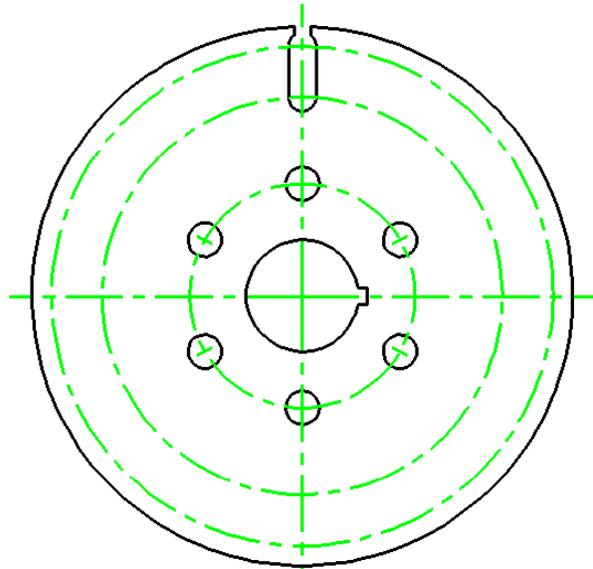

Figure 5.66 – Selecting the 0º Line

Figure 5.67 – The Arrayed Centerline

The third array involves more than just one object, so instead of starting in the Construct Array command, select the Element Selection tool from the Element Selection toolbar. Select the slotted hole by clicking and dragging to create a selection window over the area (Figure 5.68). Hold CTRL and also select the vertical centerline you created in the last array (Figure 5.69). Now select the Construct Array command from the Manipulate toolbar.
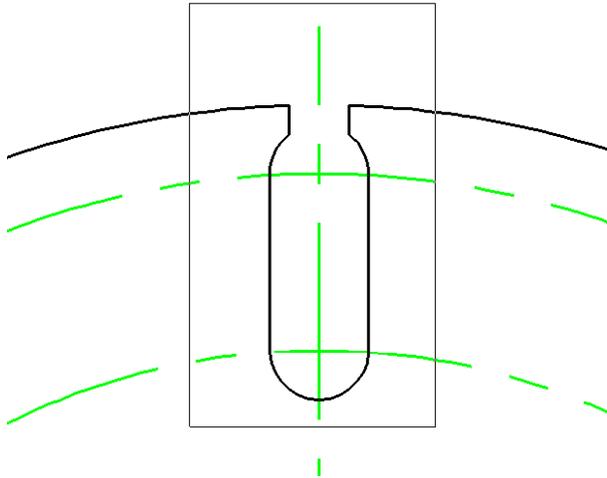
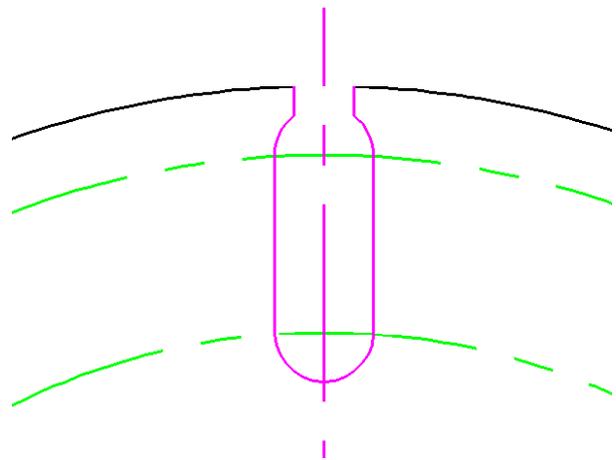**Figure 5.68 – Creating a Selection Window**

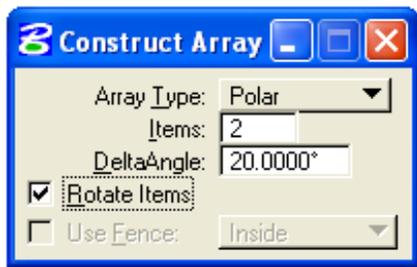**Figure 5.69 – The Selected Elements**

Figure 5.70 – The Construct Array Window

Enter 2 for the number of Items and 20º for the DeltaAngle (Figure 5.70). The slotted hole and centerline should still be highlighted. Select the center of the point twice to place the new slot and centerline (Figure 5.71). You may need to zoom in to the center of the drawing to make sure you selection is accurate. If not, and the drawing is inexact even to a very minute degree, the entire drawing will be flawed.
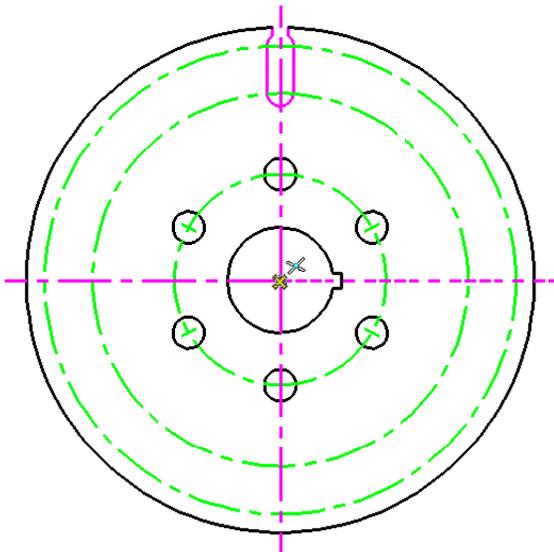

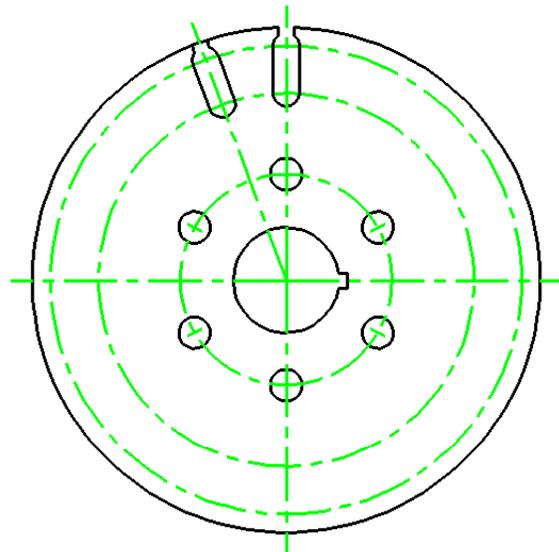**Figure 5.71 – Selecting the Center**

Figure 5.72 – The Arrayed Slot and Centerline

To check the accuracy of your array, use the scroll button on your mouse to zoom into different areas of the drawing. For example, if you zoom into the newly arrayed slot and find that the edge of the slot extends past the 12-unit circle, you have made a mistake (Figure 5.73). The new slot should directly intersect with the circle, no farther or shorter (Figure 5.74). The farther you zoom into an intersection, the more evident an inaccurate placement of an object will become.
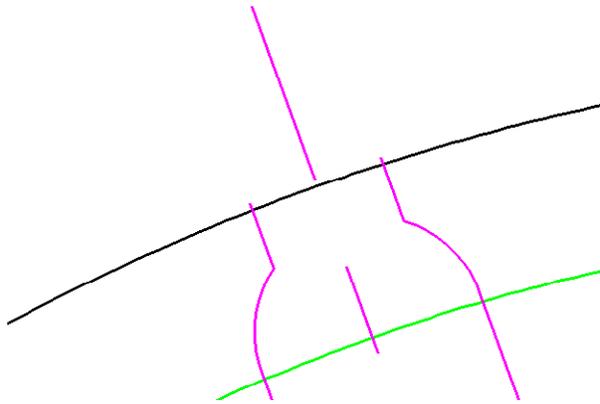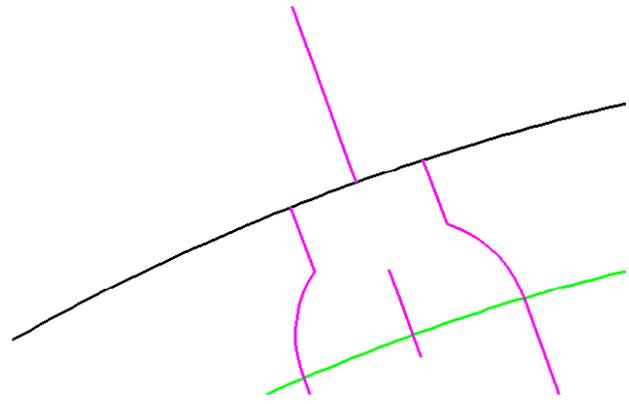

Figure 5.73 – An Inaccurate Placement   **Figure 5.74 – An Accurate Placement**

Using the IntelliTrim and Element Selection Commands to Remove Arcs and Lines

Before making the final array, you need to prep the object to be arrayed. Select the IntelliTrim tool from the Modify toolbar. Change the Mode to Quick in the IntelliTrim widow. Select the top right line of the newly created slot as the element to trim to. Now draw a line crossing the object you wish to trim, in this case the 12-unit circle any point beyond the second slot (Figure 5.75). If done correctly, the trim will erase the entire circle, leaving only a small arc connecting the two slots (Figure 5.76).
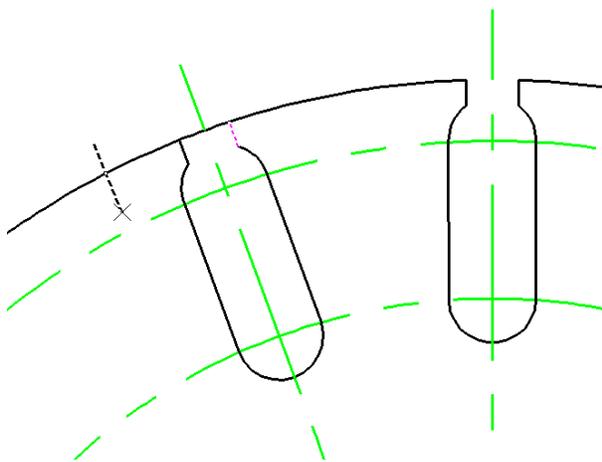

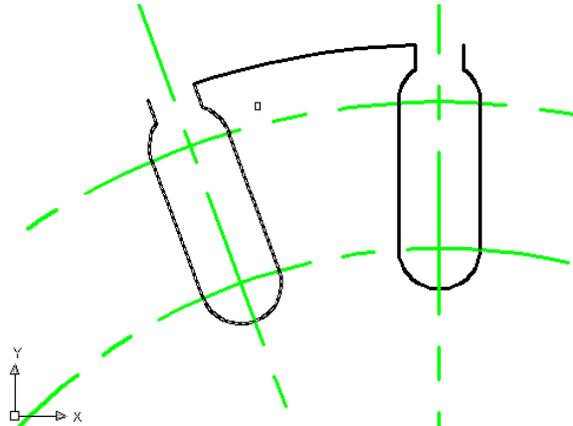**Figure 5.75 – Trimming the 12-Unit Circle**   **Figure 5.76 – The Completed Trim**

Using the Element Selection tool from the Element Selection toolbar, select the entire second slot by clicking and dragging a selection window around it (Figure 5.77). Hit DELETE to remove the slot. Only the original slot, the section of arc from the 12-unit circle, and the two vertical centerlines should remain at the top of the drawing (Figure 5.78).
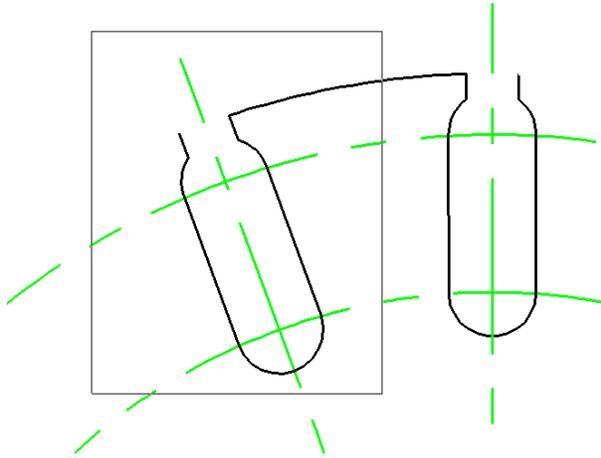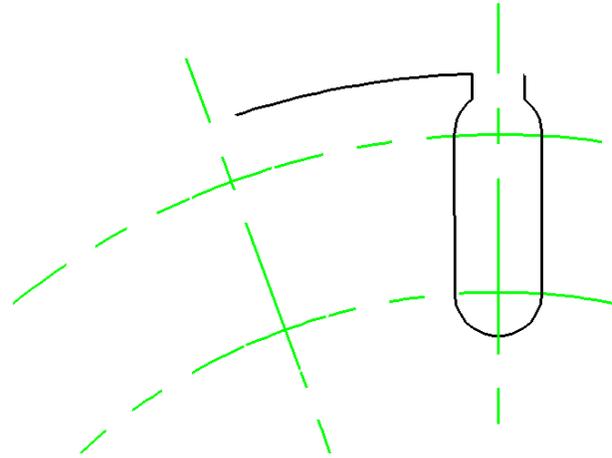

Figure 5.77 – The Selection Window   **Figure 5.78 – The Remaining Slot**

Using the Array Command to Create More Circular Patterns

While remaining in the Element Selection command, select the slot and the arc by clicking and dragging a selection window over the area. The slot and arc will become highlighted. Now select the Construct Array tool from the Manipulate toolbar. In the Construct Array window, type 18 for the number of Items and 20º for the DeltaAngle. Make sure the Rotate Items box has been checked, as it has been for all of the prior polar arrays. Select the center of the drawing twice, and 17 new slots and arcs will appear, creating a complete perimeter of the drawing.
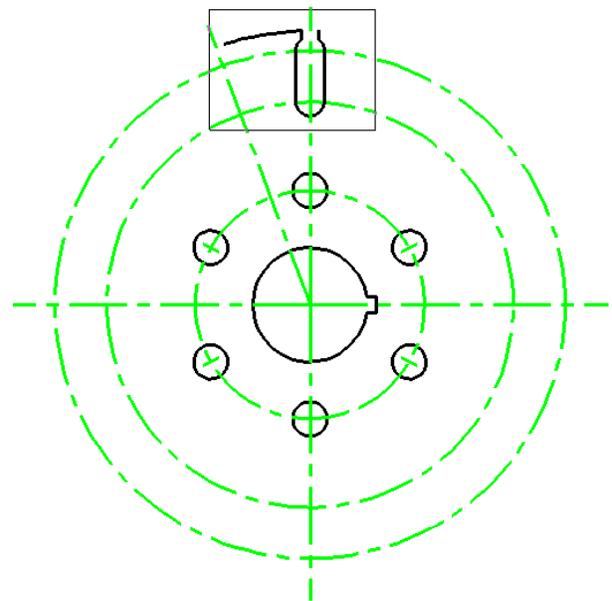

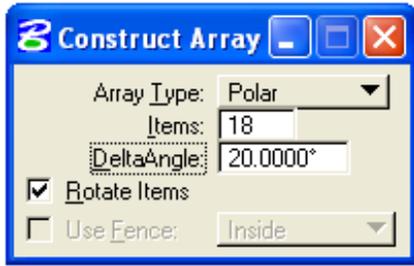**Figure 5.79 – The Selection Window**
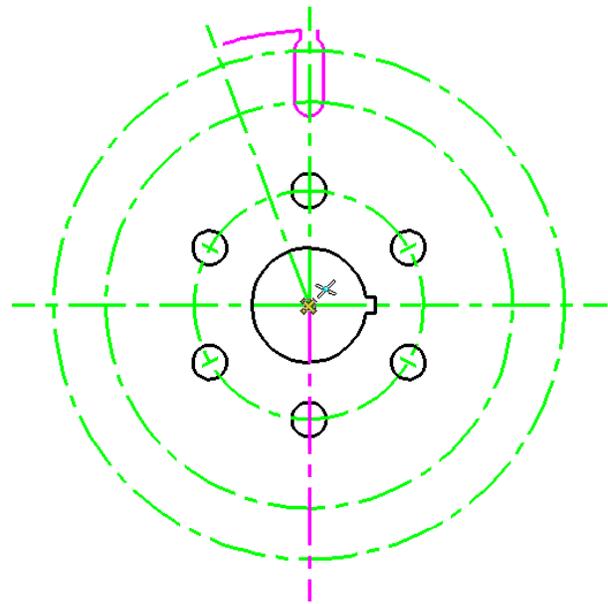
Figure 5.80 – The Construct Array Window

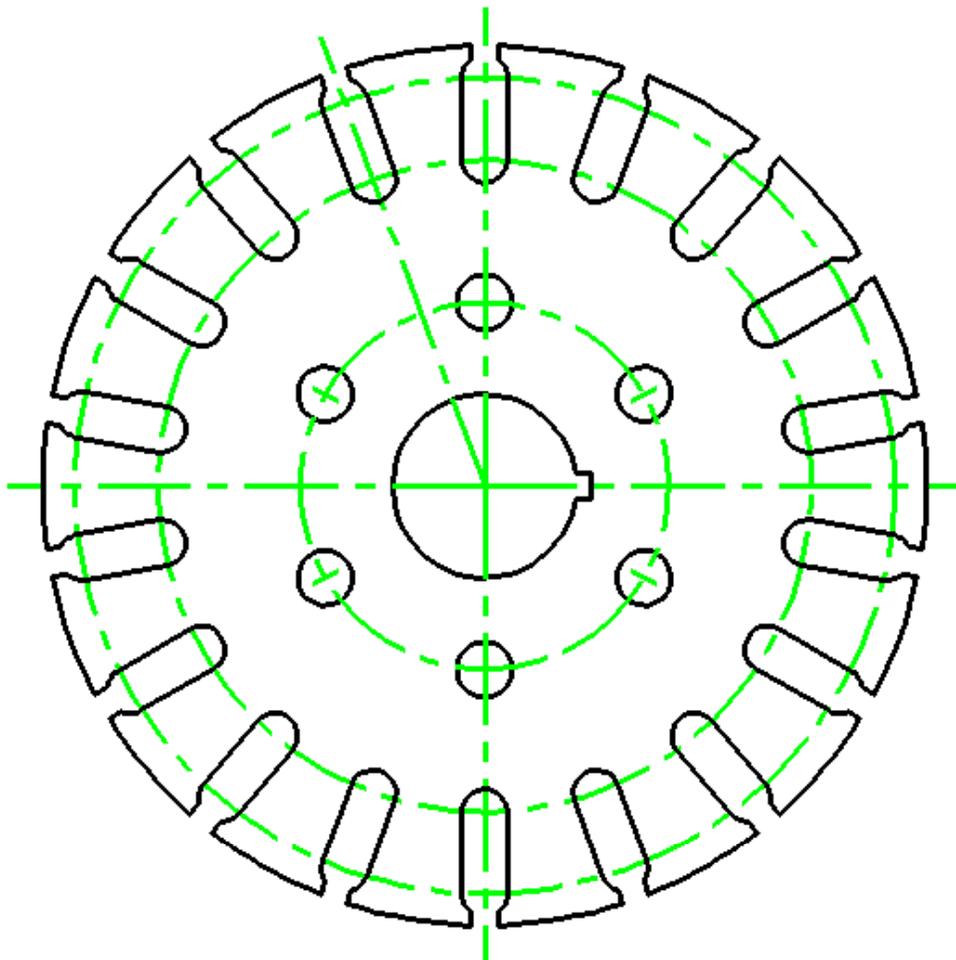**Figure 5.81 – Selecting the Center**

Figure 5.827 – The Finished Circular Problem

* World Class CAD Challenge 02-13 * - Create a New file, create the three new levels, set the MultiSnap 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.