

Chapter

6

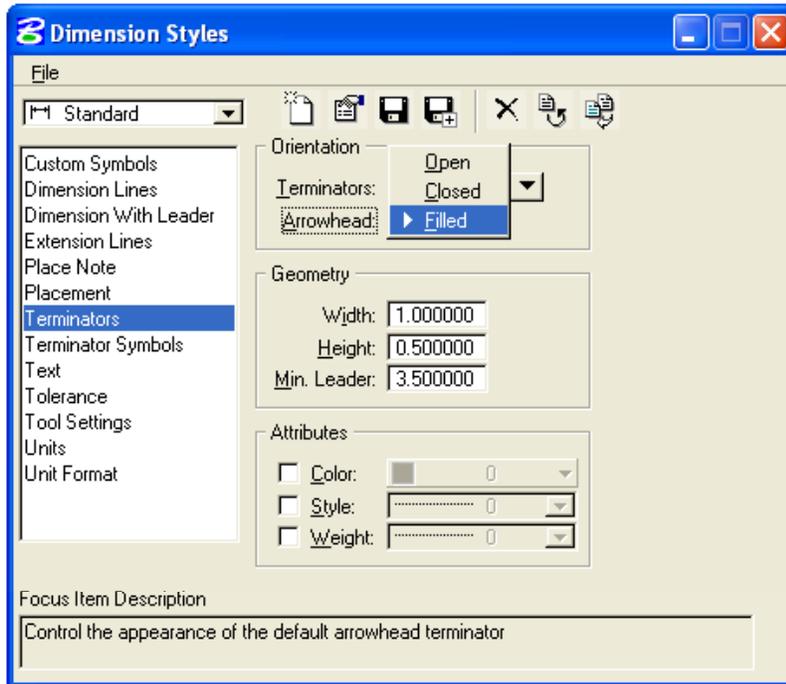
Dimensioning the Circular Problem

This chapter will cover:

- 1. Dimensioning Styles**
- 2. Trimming Centerlines with the IntelliTrim Tool**
- 3. The Radial Dimension Tool**
- 4. Linear Dimension Tools**
- 5. The Arc Distance Tool**
- 6. The Dimension Angle Size Tool**
- 7. The Place Text Tool**

Configuring Dimension Styles

From the Menu bar, click Element and then Dimension Styles. The Dimension Styles window will appear on the screen. It should have the “Standard” dimension style loaded in the Active Style drop box. You created this style in the rectangle problem. You will use this same style when dimensioning the circular problem, but with a few changes.



Under Terminators, change the Arrowhead to Filled (Figure 6.1). Under Units, change the Accuracy to 0.123. Under Unit Format, change the Accuracy for the Angle Format to 0.1 and check the Leading Zero box.

Figure 6.1 – Changing the Arrowhead to Filled

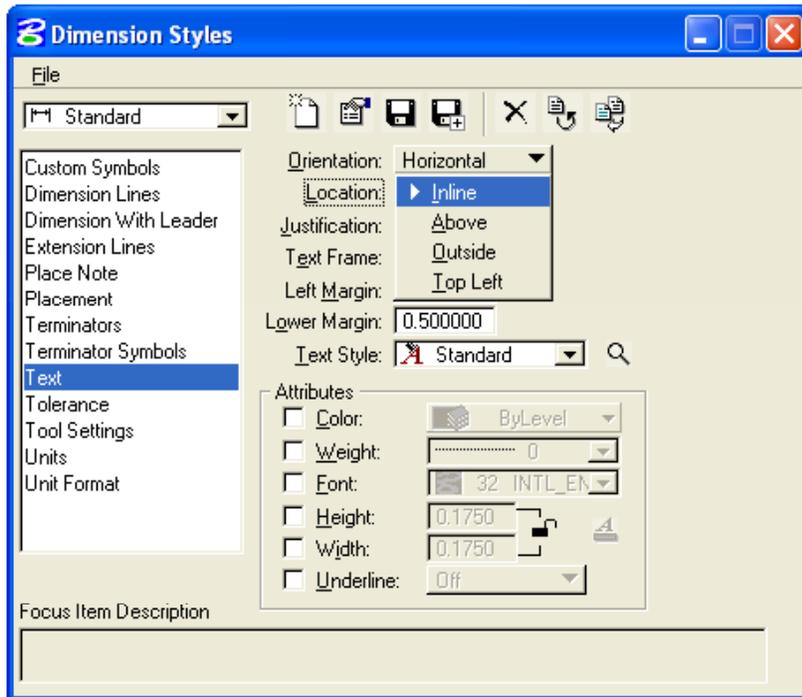


Figure 6.2 – Changing the Location to Inline

Under Text, change the Location to Inline (Figure 6.2). Next, select the magnifying glass next to the Text Style drop box and the Text Styles window will appear. If the Standard text style you created during the rectangle problem is not present, create a new text style named Standard. Change the Height to 0.15 and the font to 32 INTL_ENGINEERING (Figure 6.3). Save the text style and close the Text Styles window. Select Standard to be the Active Text Style in the Text Style drop box on the Dimension Styles window. Save the Standard dimension style and then exit the Dimension Styles window.

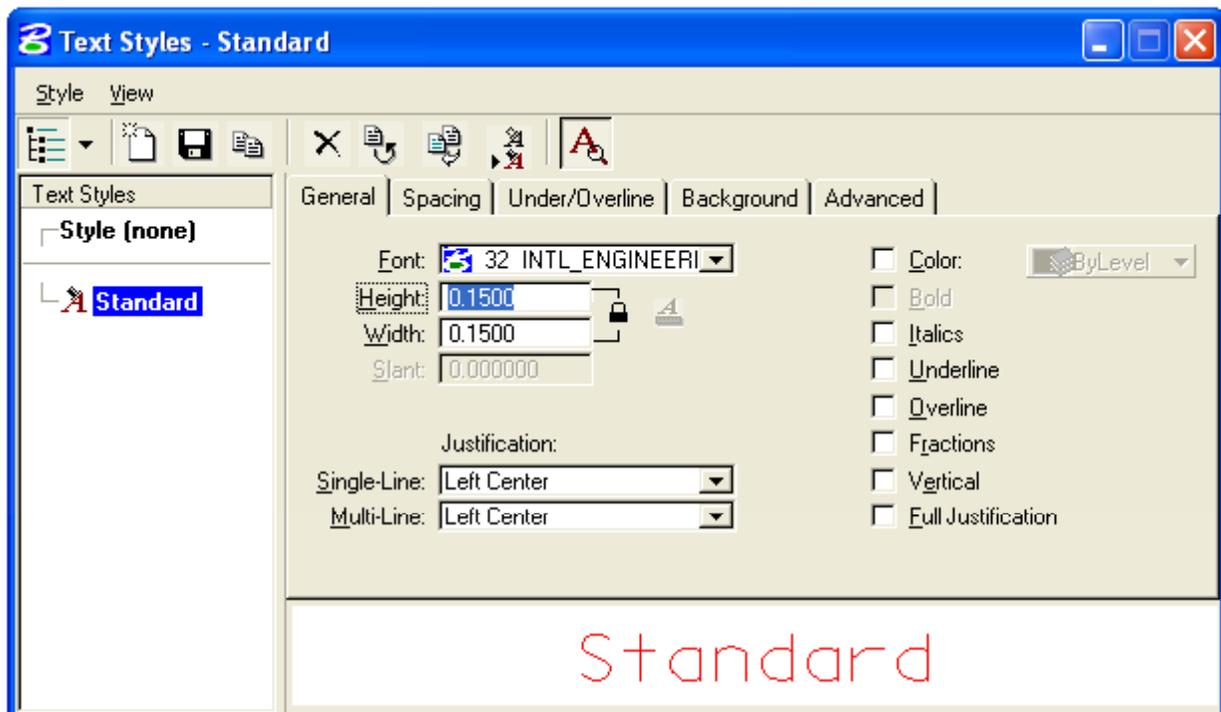


Figure 6.3 – The Standard Text Style

Trimming Centerlines with the IntelliTrim Tool

You could use the Partial Delete tool from the Modify toolbar to shorten the two polar centerlines on the outside of the circular problem, but chances are that to achieve accuracy and neatness you should trim both at once. The Partial Delete command only allows you to work on one element at a time.



Try this: Using the SmartLine tool from the Linear Elements toolbar, create two construction lines both extending from the center of the part. The exact length and angle of these lines is not specified, so use your own judgment. Draw the first extending out between the two upper centerlines (Figure 6.4), and the other between the north and northeast 0.75 holes (Figure 6.5).

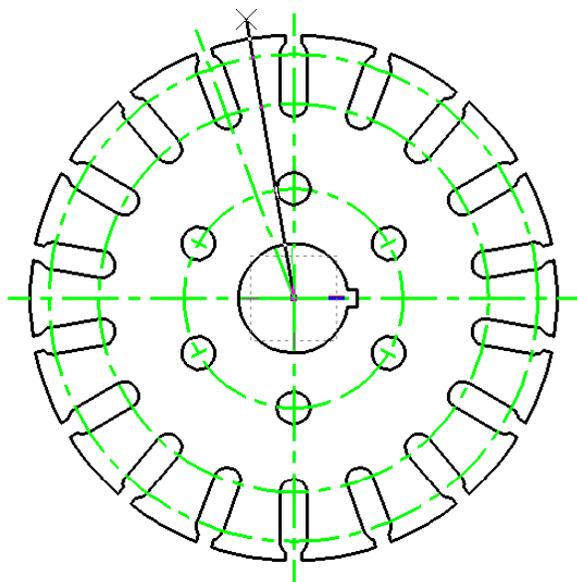


Figure 6.4 – The First Cutting Line

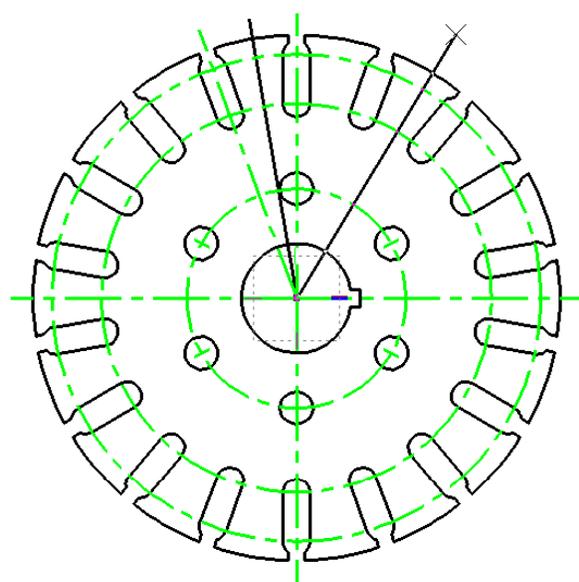


Figure 6.5 – The Second Cutting Line

Select the IntelliTrim tool from the Manipulate toolbar. Make sure the Mode is set to Advanced and that Select Cutting Elements is selected on the IntelliTrim window (Figure 6.6). Select the two construction lines you just drew to be the cutting elements and right-click to continue (Figure 6.7). Select the two outer circles on the center level to be the elements to be trimmed and right-click again (Figure 6.8). Finally, place a point inside the two cutting lines to keep only the part of the two circles between them (Figure 6.9). Right-click to apply the trim.



Figure 6.6 – The IntelliTrim Window

Now the centerlines still define the center of the arcs in the slotted holes, but do not clutter the drawing with unnecessary details. Use the Element Selection tool from the Element Selecting toolbar to select and delete the two cutting lines (Figure 6.10).

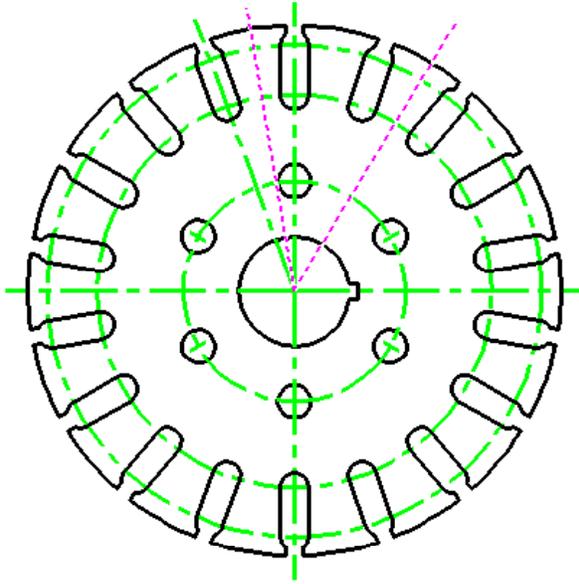


Figure 6.7 – The Cutting Lines

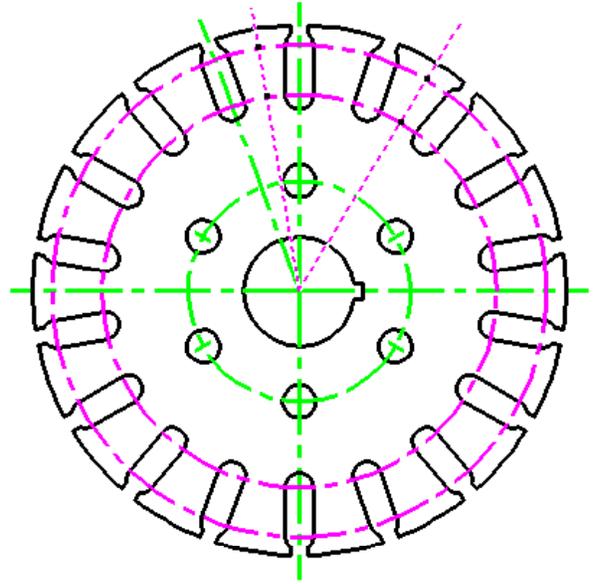


Figure 6.8 – The Lines to Trim

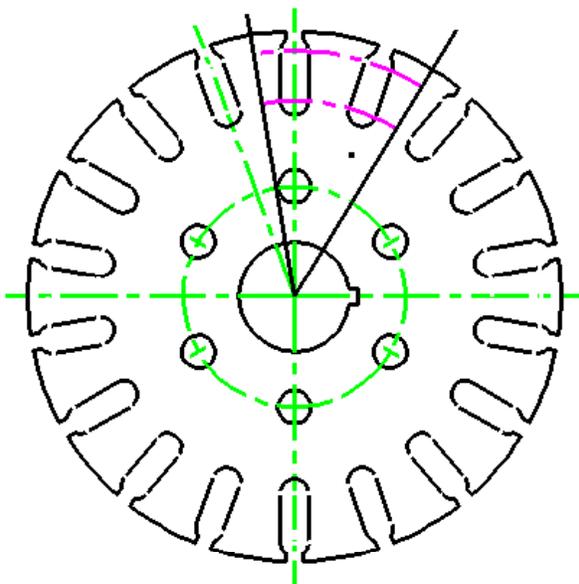


Figure 6.9 – The Portions to Keep

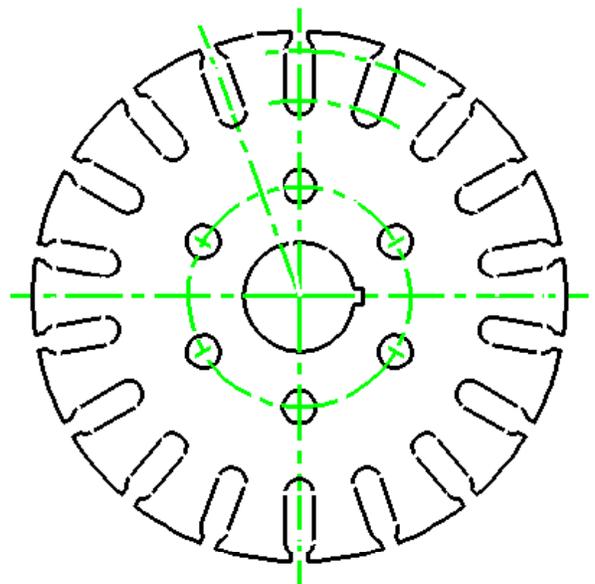


Figure 6.10 – The Cutting Lines Removed

Using the Radial Dimension Tool

Before you can begin dimensioning the circular problem, you must first change to the Dimension level by selecting it from the Active Level combo box. Then you can select the Dimension Radial tool from the Radial Dimension toolbar.



The Dimension Radial tool allows you to place an assortment of dimensions without switching tools. In the Dimension Radial window, make sure that Standard is selected in the Active level drop box. Change the Mode to Diameter Extended and the Alignment to View (Figure 6.11). Now you can place four diameter dimensions: the 12, 5, 2.5, and 0.75 unit circles. Try to place the dimensions in similar positions to what you see in the examples (Figure 6.12). This will ensure you will not encounter any layout problems.

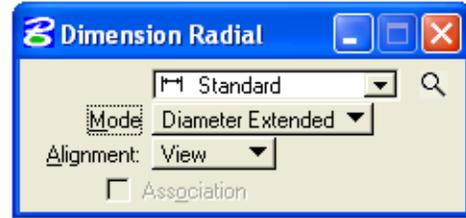


Figure 6.11 – The Dimension Radial Window

To place a diameter dimension, click the circular element you would like to dimension, and then select a point to place the text. These should be relatively simple.

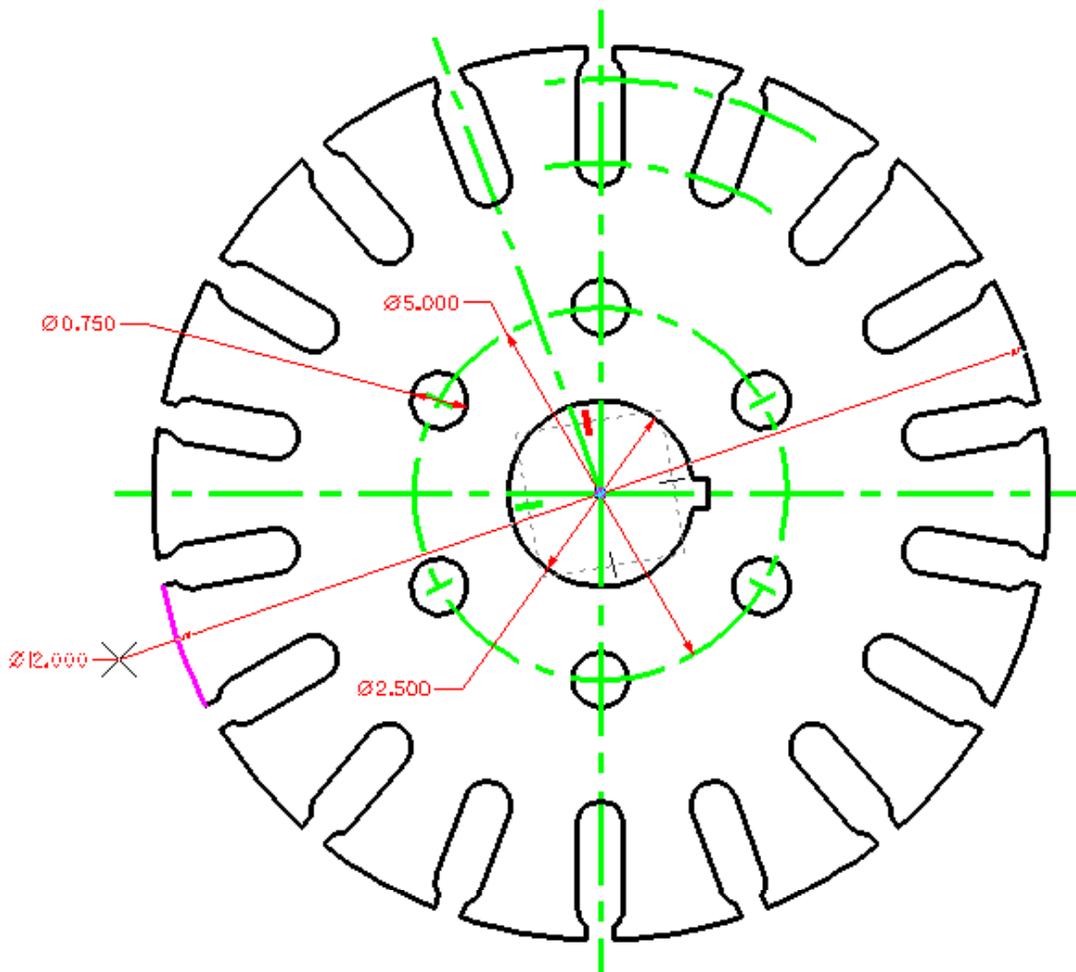


Figure 6.12 – The Diameter Dimensions

Change the Mode from Diameter Extended to Radius Extended on the Dimension Radial window. Now you can dimension two more objects: the inner center-arc you trimmed earlier in this chapter (Figure 6.13), and the arc of the slotted hole (Figure 6.14).

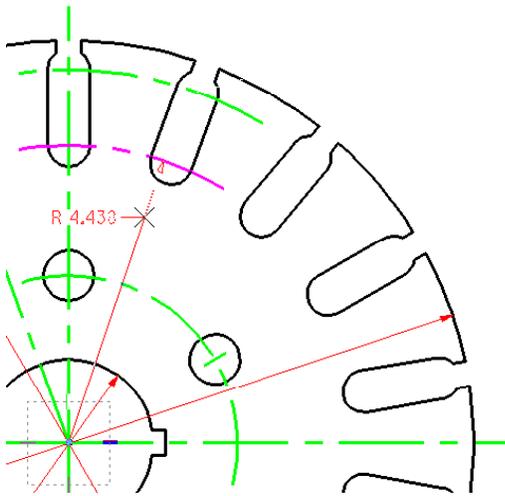


Figure 6.13 – The Radius of the Inner Center Arc

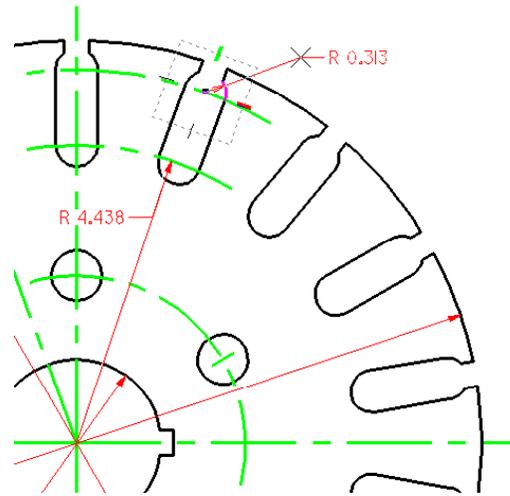


Figure 6.14 – The Radius of the Slotted Hole

Using Linear Dimension Tools

Select the Dimension Size with Arrows tool from the Linear Dimension toolbar and the Dimension Size with Arrows window will appear. Once again, make sure the Standard dimension style is selected, and change the Alignment to View.



Place the first linear dimension on the center hole. Select the bottom corner of the key slot as the first point of the dimension, and the AccuDraw compass will appear. The next point will determine the length of the extension line, so pull the cursor down past the bottom of the hole and click (Figure 6.15). Click the west quadrant of the center hole to finish placing the dimension (Figure 6.16). The length should be 2.687; if it is not, hit CTRL+Z to Undo the dimension and try again, zooming in if needed to select the exact points. If you are having trouble with accuracy, try selecting the Intersection Snap from Snap Mode toolbar. Right-click twice to reset the command.

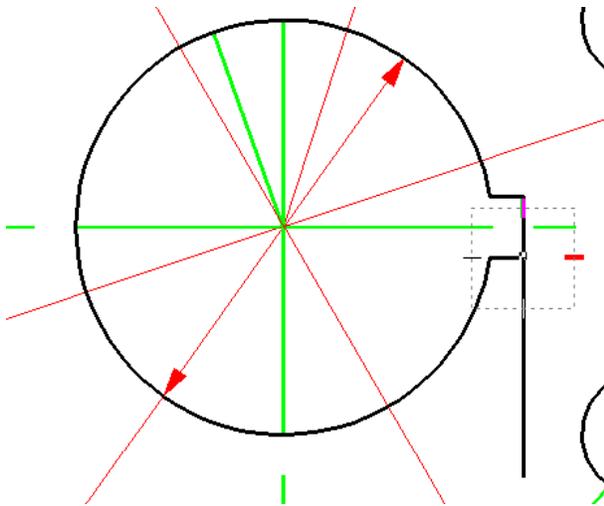


Figure 6.15 – Selecting the Corner of the Key

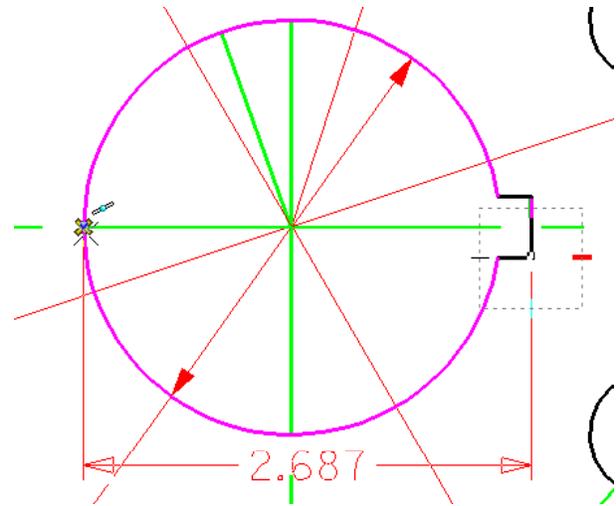


Figure 6.16 – Placing the 2.687 Dimension

Using the same technique, dimension the key of center hole. Select the top corner of the key, and then click again to the right to set the extension line distance (Figure 6.17). Click the bottom of the key to set the length of the dimension (Figure 6.18). Because the text is larger than the space in between the extension lines, you will need to make one more selection to place the text. Click anywhere in the general area to place the number and right-click twice to reset the command (Figure 6.19).

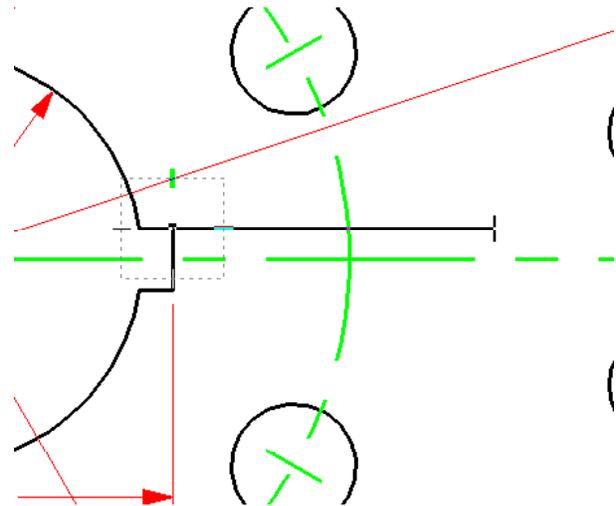


Figure 6.17 – Selecting the Top Corner of the Key

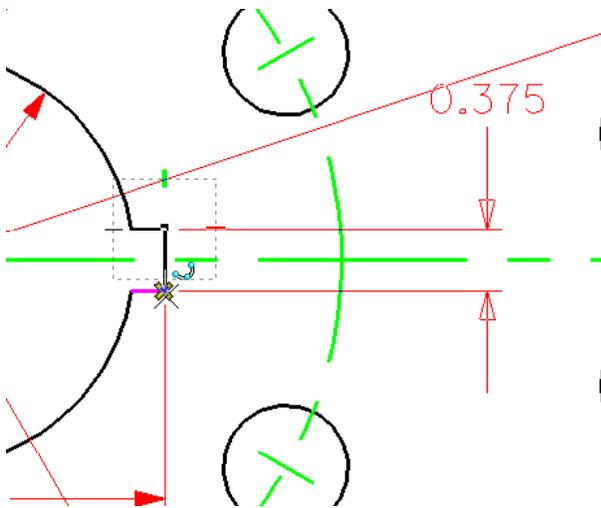


Figure 6.18 – Placing the Length of the Dimension

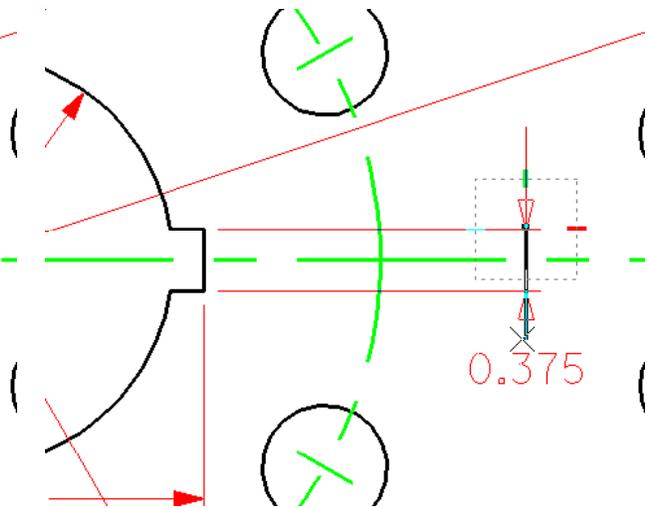


Figure 6.19 – Placing the Text

Place two linear dimensions on the top slotted hole: one showing the distance between the two inside edges of the hole, and the other showing the distance between the centerline and the edge (Figure 6.20).

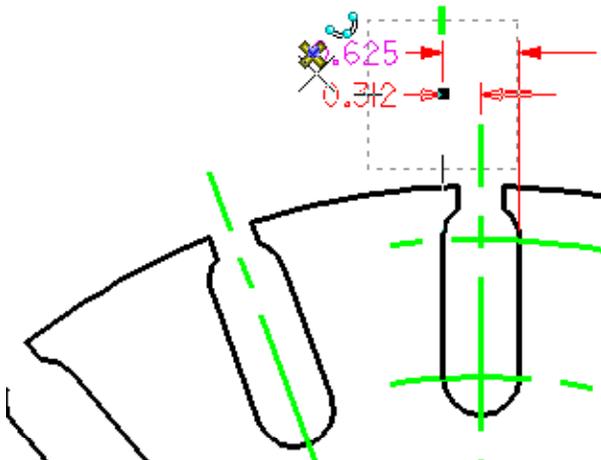


Figure 6.20 – The 0.625 and 0.312 Dimensions

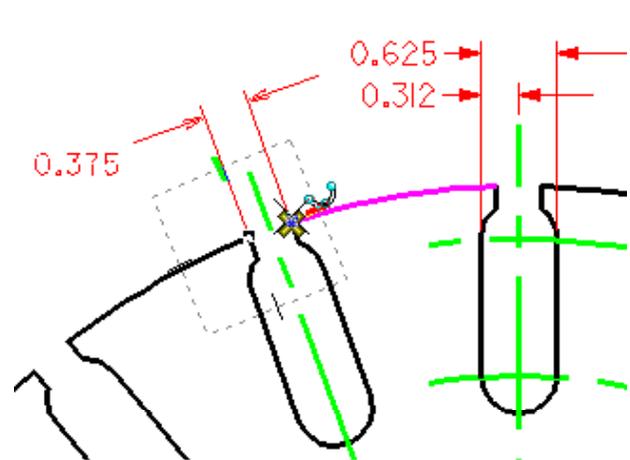


Figure 6.21 – The 0.375 Dimension

Before placing the final linear dimension, change the Alignment to True on the Dimension Size with Arrows window. Place a dimension showing the width of the mouth on the slotted hole to the left of the top hole. Because the Alignment is set to True, the dimension will be parallel with the opening (Figure 6.21).

Using the Arc Distance Tool

Select the Dimension Arc Distance tool from the Radial Dimension toolbar. Select the inner center arc and then the outer center arc (Figure 6.22). Finally, place the dimension in position next to the 4.438 radius (Figure 6.23).

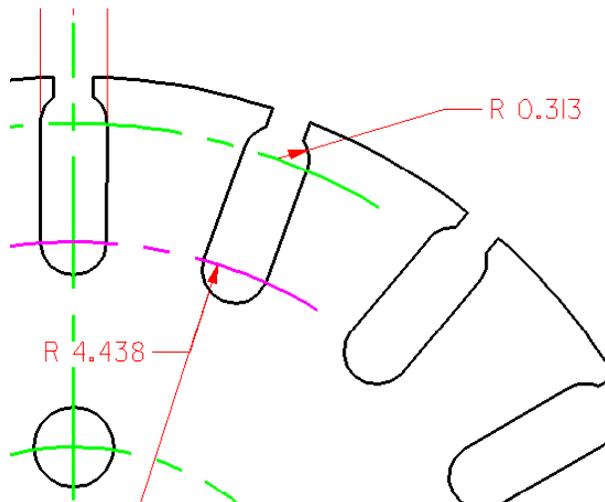


Figure 6.22 – Selecting the Inner Center Arc

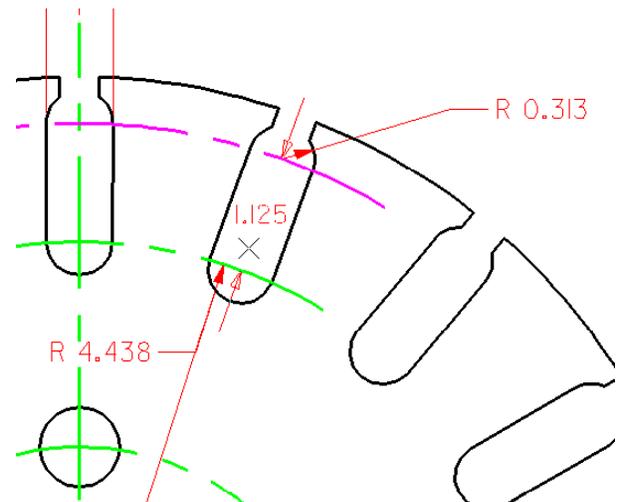


Figure 6.23 – Placing the Dimension

Using the Dimension Angle Size Tool

The last two dimensions to place on the circular problem are the angular dimensions. Select the Dimension Angle Size from the Angular Dimension toolbar.



Select the top of the center mark in the northeast 0.75 hole. When the AccuDraw compass appears, select a point extending directly in line with the centerline outside the part; this will be the length of the extension line (Figure 6.24). Next, select the center of the part to define the center of the angle (Figure 6.25). Finally, select the top of the top centerline to place a 60° dimension (Figure 6.26). Do not right-click yet. To place the other dimension, click the top of the next centerline. You should have 20° dimension right next to the 60° one you just placed (Figure 6.27). Now right-click twice to reset the command.

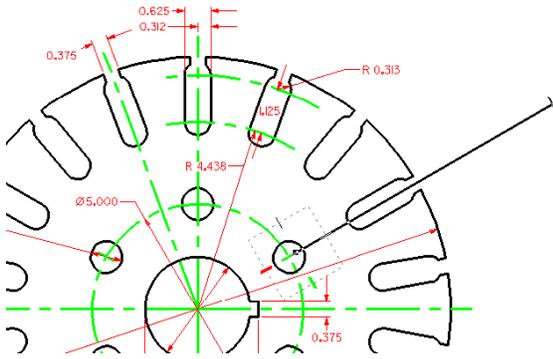


Figure 6.24 – Setting the Extension Line Length

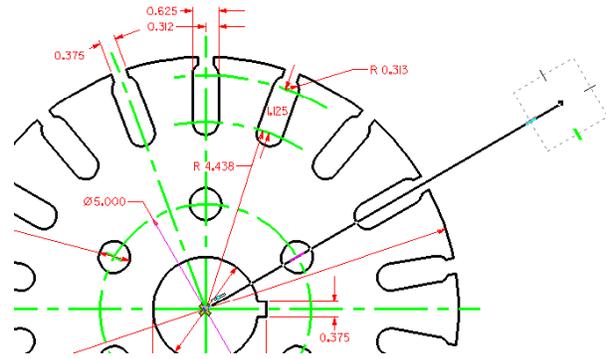


Figure 6.25 – Selecting the Center

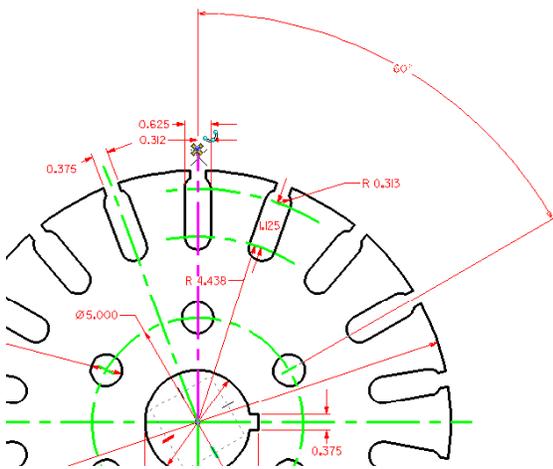


Figure 6.26 – The 60° Dimension

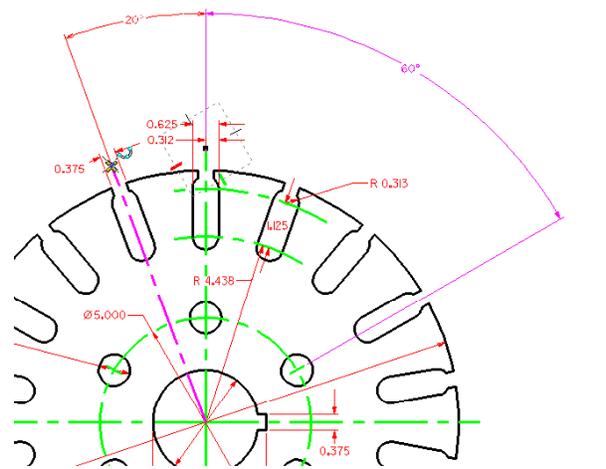


Figure 6.27 – The 20° Dimension

*** World Class CAD Challenge 02-14 * - Create a New file, create the three new levels, set the AccuSnap settings, draw the seven circles, move and copy the two circles, add lines for the keyway and slotted holes, do the four arrays, and add the 14 dimensions in 9 minutes. Continue this drill four times, each time completing the drawing under 9 minutes to maintain your World Class ranking.**

Using the Place Text Tool to Add a Note to a Dimension

Select the Place Text tool from the Text toolbar and the Text Editor – Word Processor will appear. In the Place Text window, the Standard text style should be selected. Also, make sure the Active Angle is set to 0.0. Type “6 holes” into the Text Editor – Word Processor window, and then place the text on the drawing directly below the 0.750 dimension (Figure 6.28).



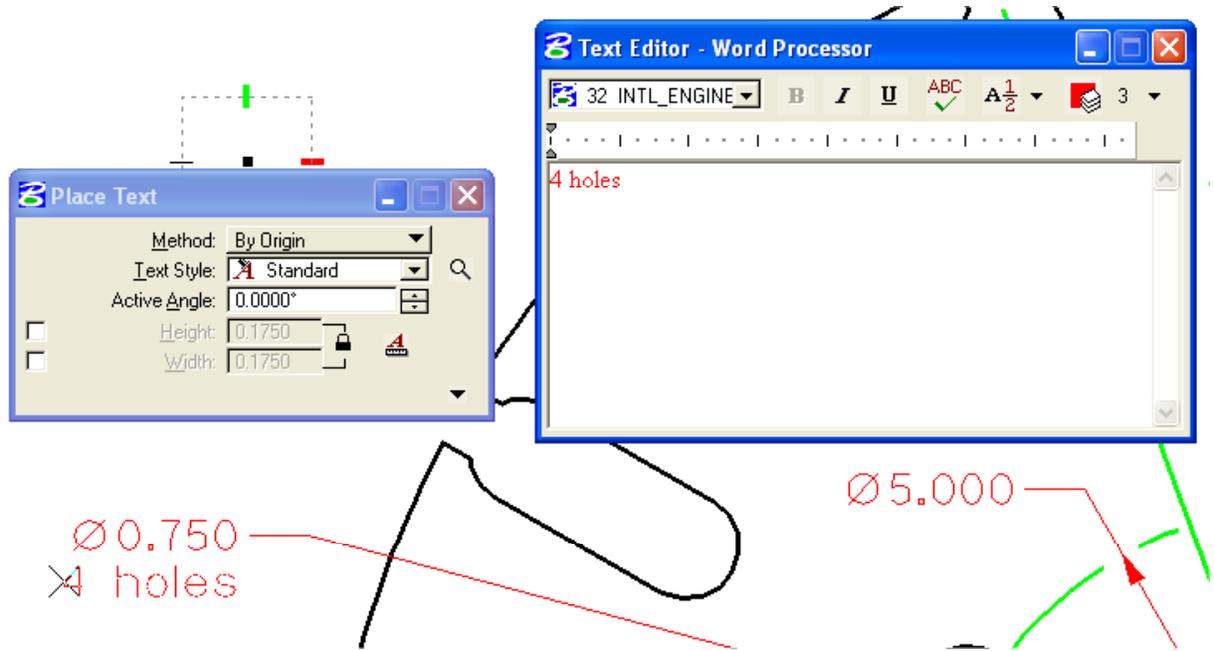


Figure 6.28 – Placing Text

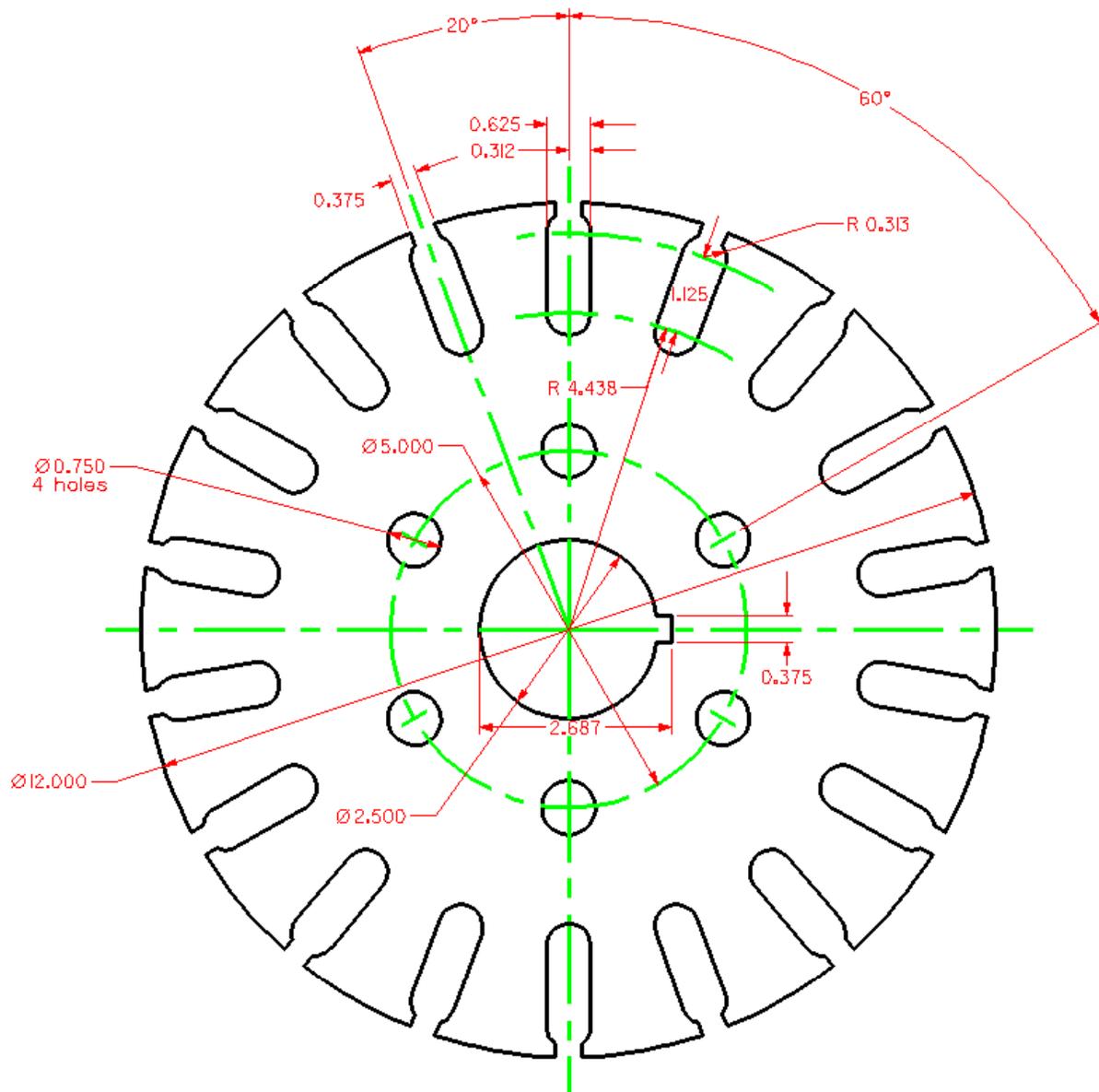


Figure 6.29 – The Dimensioned Circular Problem

*** World Class CAD Challenge 02-15 * - Create a New file, create the three new levels, set the AccuSnap settings, draw the seven circles, move and copy the two circles, add lines for the keyway and slotted holes, do the four arrays, add the 14 dimensions and the text in 10 minutes. Continue this drill four times, each time completing the drawing under 10 minutes to maintain your World Class ranking.**