

Chapter 6

Dimensioning the Circular Problem

This chapter will cover:

1. Trimming Centerlines with the IntelliTrim Tool
2. Dimensioning Styles
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

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 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 6.1), and the other between the north and northeast 0.75 holes (Figure 6.2).

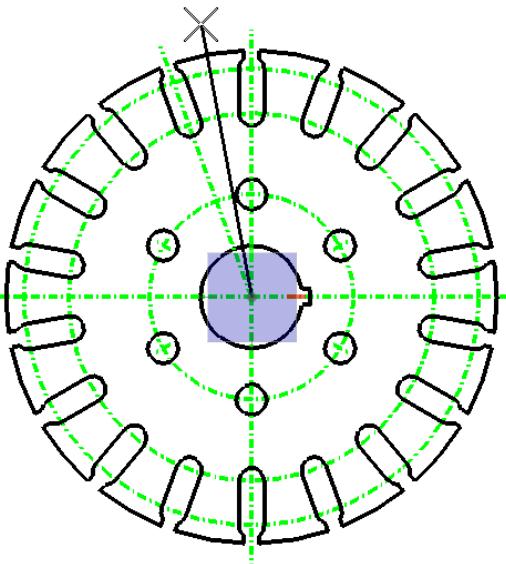


Figure 6.1– The First Cutting Line

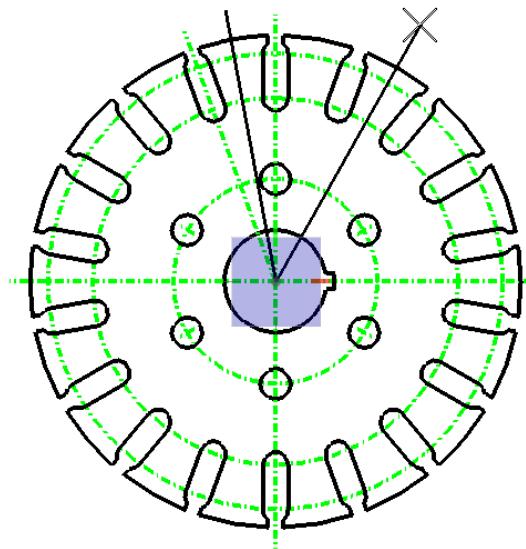


Figure 6.2 – The Second Cutting Line

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 6.3). Select the two construction lines you just drew to be the cutting elements and right-click to continue (Figure 6.4). Select the two outer circles on the center level to be the elements to be trimmed and right-click again (Figure 6.5). Finally, place a point inside the two cutting lines to keep only the part of the two circles between them (Figure 6.6). Right-click to apply the trim.

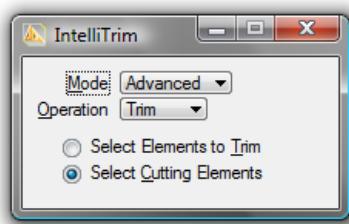


Figure 6.3 – The IntelliTrim Window

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 6.7).

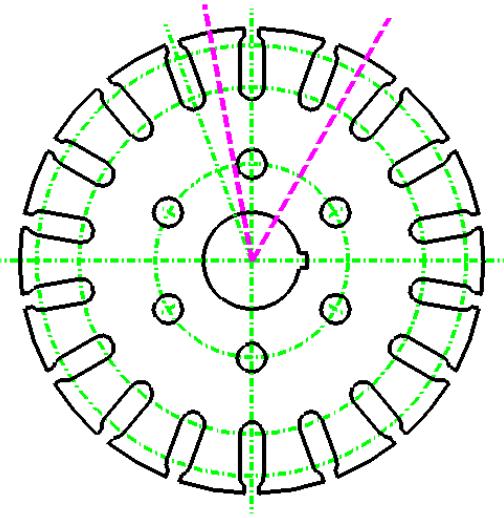


Figure 6.4 – The Cutting Lines

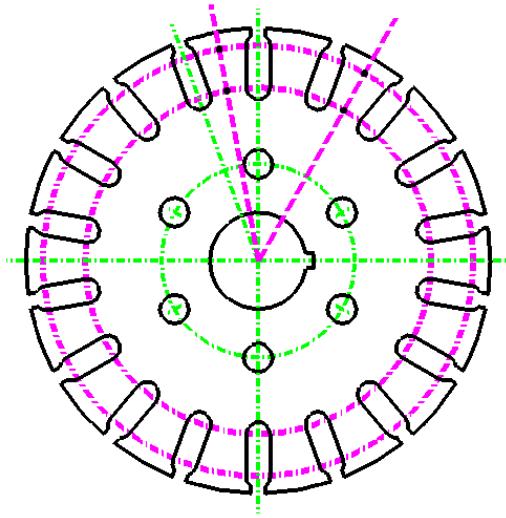


Figure 6.5 – The Lines to Trim

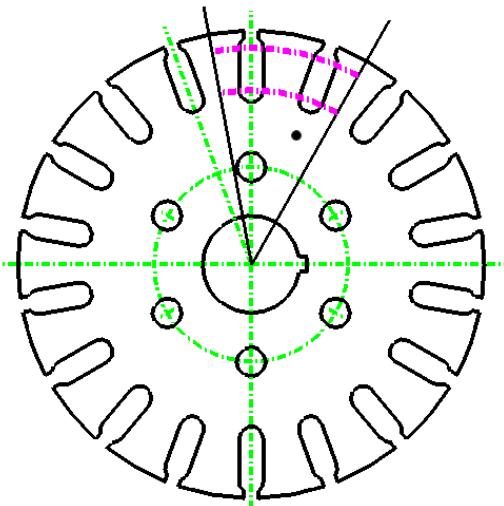


Figure 6.6 – The Portions to Keep

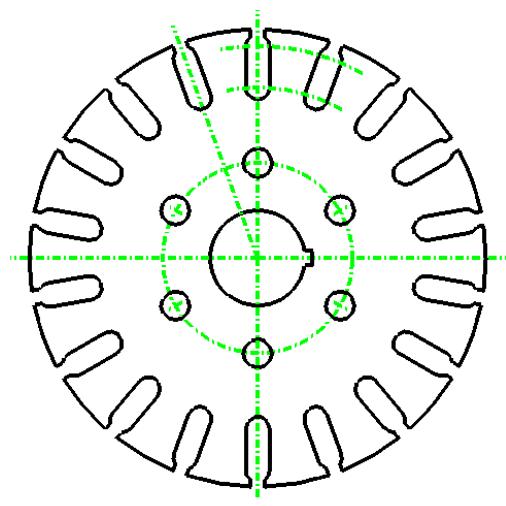


Figure 6.7 – The Cutting Lines Removed

Configuring Dimension Styles

Select the Radial Dimension tool from the Radial Dimension toolbar.



Before placing radius on the drawing, you need to modify the dimension settings. Click on the magnifying glass beside the style menu access the Dimension Styles window. Click the Create Dimensions Style button, and create a new style called Standard (Figure 6.9).

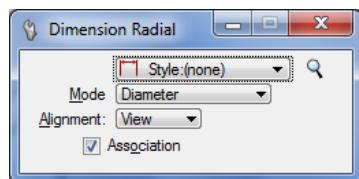


Figure 6.8 – The Place Center Mark Window

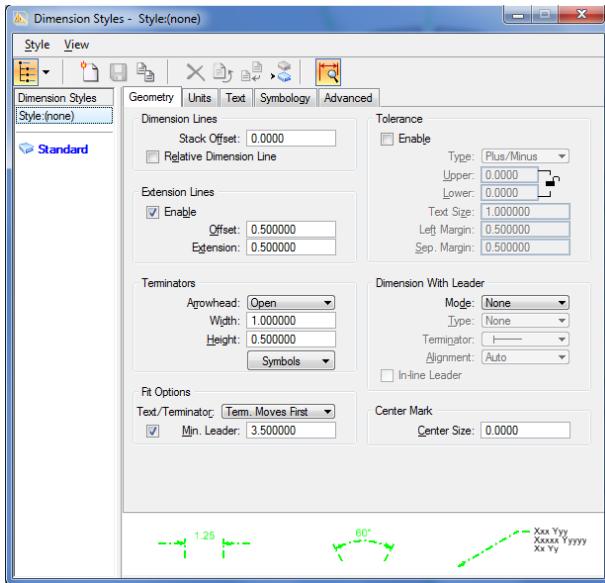


Figure 6.9 – Creating a New Style

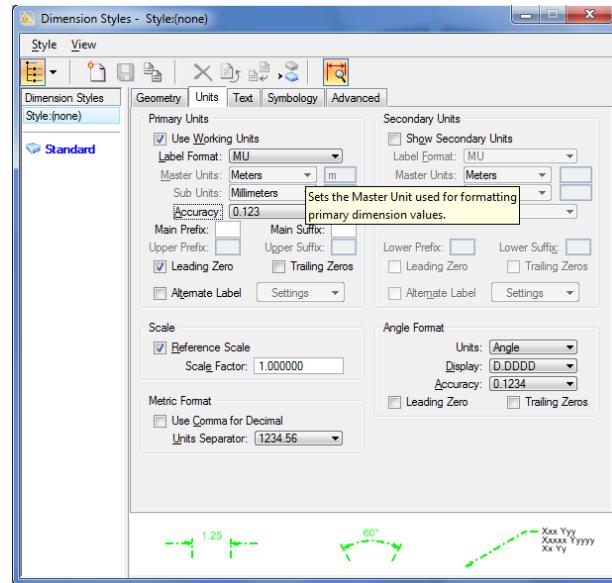


Figure 6.10 – Changing the Accuracy

Under Units, change the Accuracy to 0.123 (Figure 6.10). Also, check the Leading Zeros and Trailing Zeros boxes in the Primary Units area.

Next, you need to create a text style. Click the Place Text tool from the Text toolbox on the Task dialog box and the Text Editor – Word Processor window will appear, as well as the Place Text window. On the Place Text window, click the magnifying glass beside the Text Style drop menu. The Text Style menu will appear. Click the New button and create a new text style named Standard (Figure 6.11).

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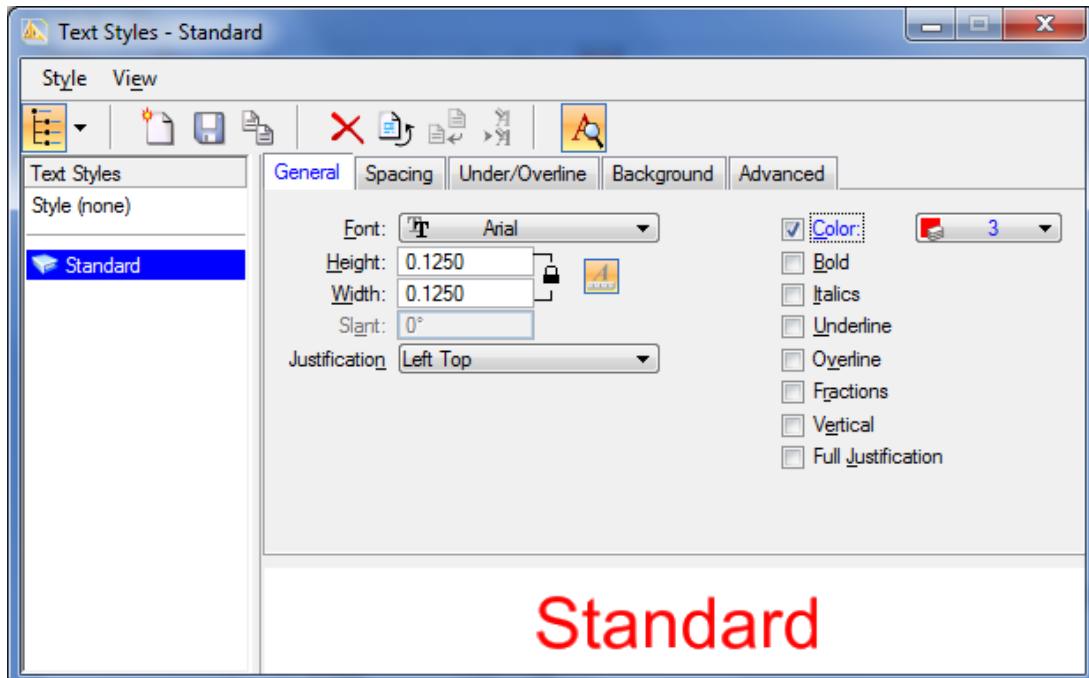


Figure 6.11 – Creating a New Text Style

Change the Text Height to 0.125, and the width will automatically change to match the height (Figure 6.14). Under Spacing, change the Line Spacing to 0.0500. Change the color to ByLevel. Click Save to save your new text style. You will use this style when placing dimensions and notes. Close the Text Styles window to continue configuring the Dimension Style.

Under Text, select the Text Style drop menu and select the Standard text style you just created (Figure 6.12). In the Format area, change the Orientation to Horizontal. Click the save style button at the top of the window when you are finished. Now you are ready to place the radius dimensions.

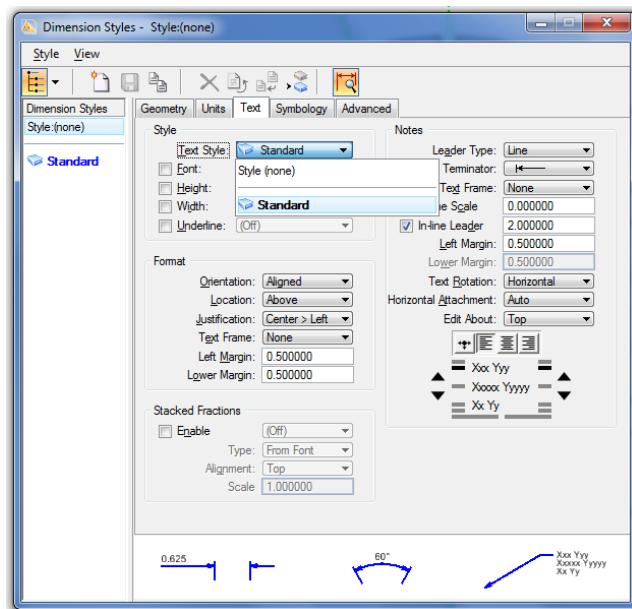


Figure 6.12 – Changing the Accuracy

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 6.13). 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 6.14). This will ensure you will not encounter any layout problems.

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.

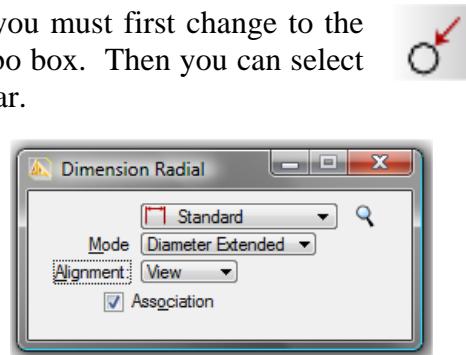


Figure 6.13 – The Dimension Radial Window

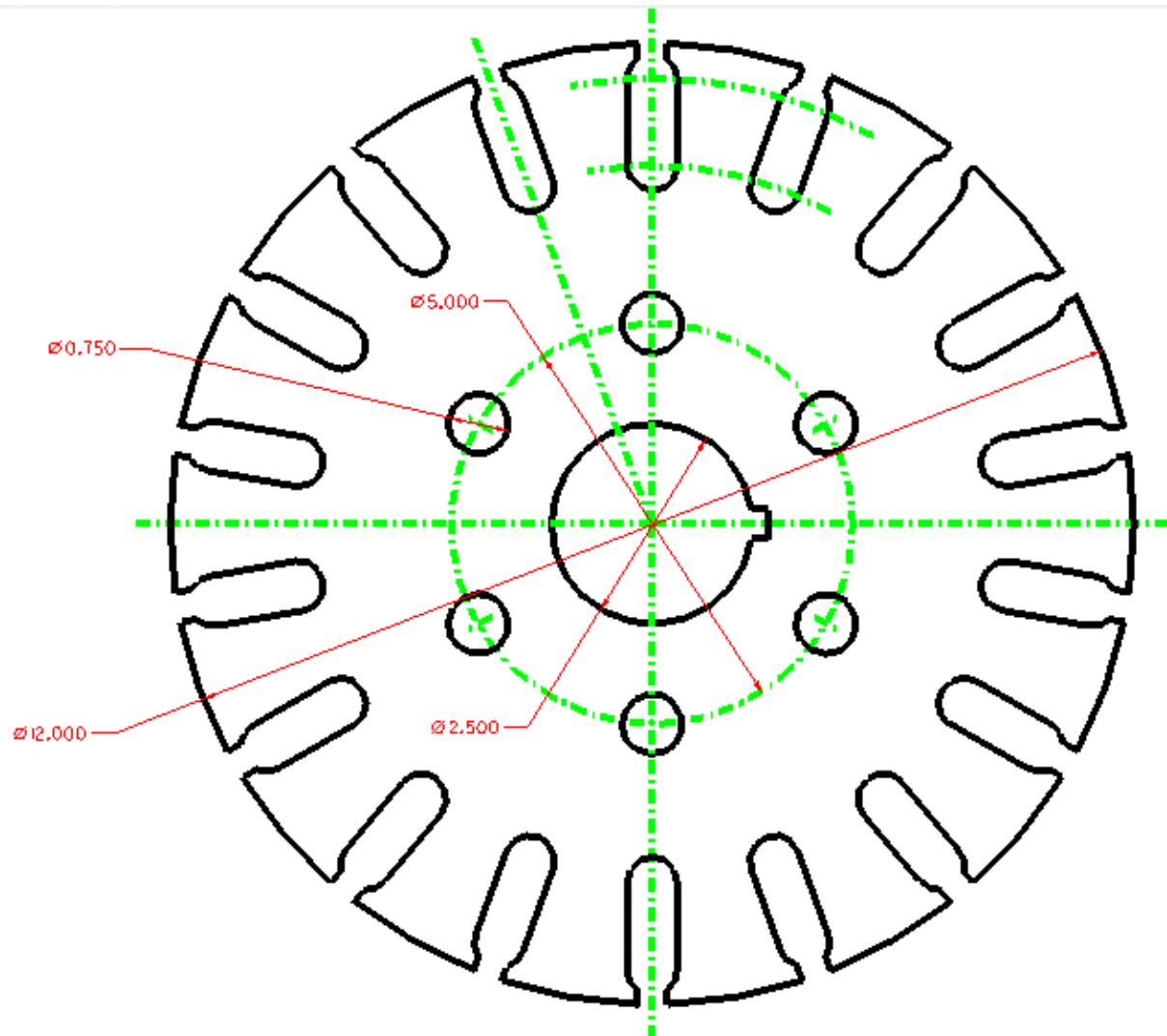


Figure 6.14 – The Diameter Dimensions

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 6.15), and the arc of the slotted hole (Figure 6.14).

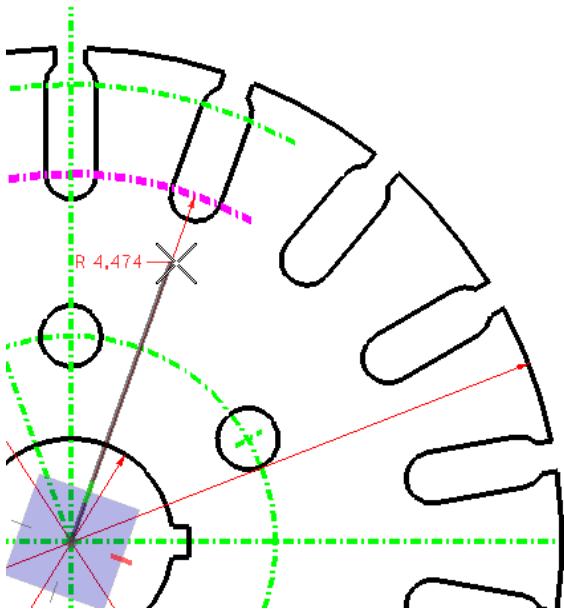


Figure 6.15 – The Radius of the Inner Center Arc

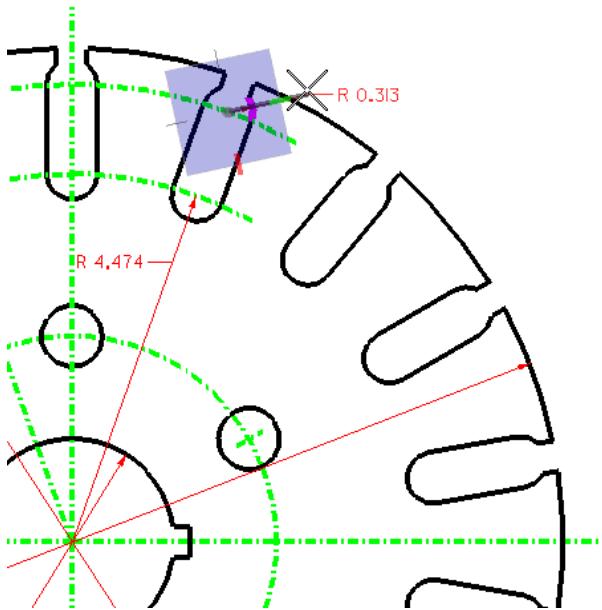


Figure 6.16 – The Radius of the Slotted Hole

Using Linear Dimension Tools

Select the Dimension Linear tool from the Dimensioning toolbar and the Linear Dimensioning 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. Click the west quadrant of the center hole to finish set the length of the dimension (Figure 6.17). 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. Pull the dimension down below the keyhole and left-click to place it (Figure 6.18). If you are having trouble with accuracy, try selecting the Intersection Snap from Snap Mode toolbar. Right-click twice to reset the command.

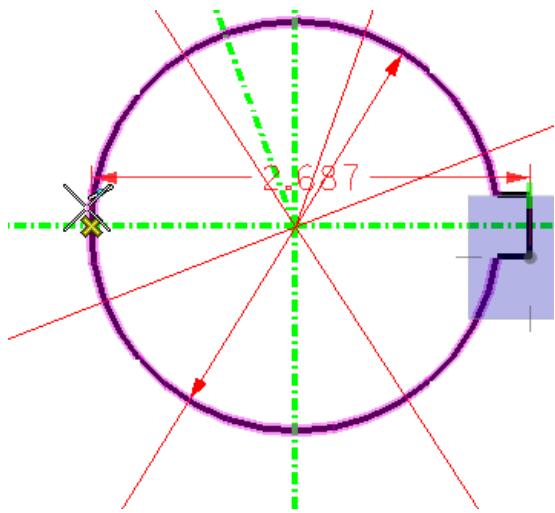


Figure 6.17 – Selecting the Corner of the Key

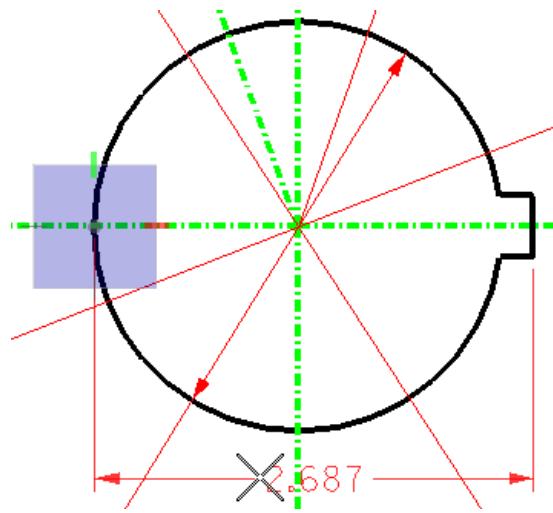


Figure 6.18 – Placing the 2.687 Dimension

Using the same technique, dimension the key of center hole, except this time changing the Location to Semi-Automatic in the Linear Dimensioning window. Select the top corner of the key, and then the bottom of the key to set the dimension to 0.375 (Figure 6.19). Pull the dimension to the right so that it is unobstructed by other elements (Figure 6.20). 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 6.21).

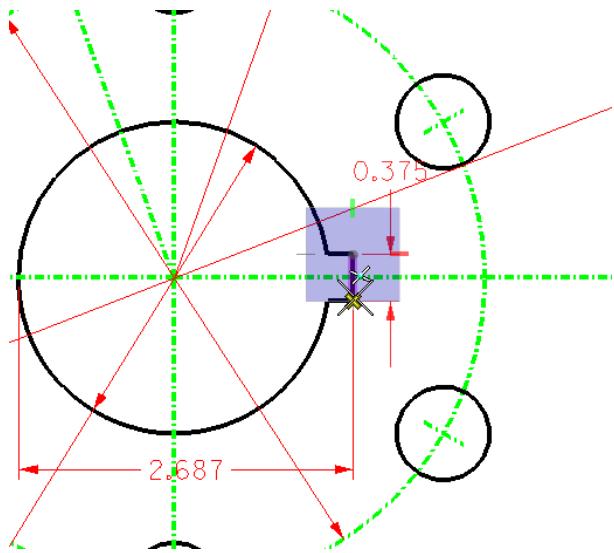


Figure 6.19 – Selecting the Top Corner of the Key

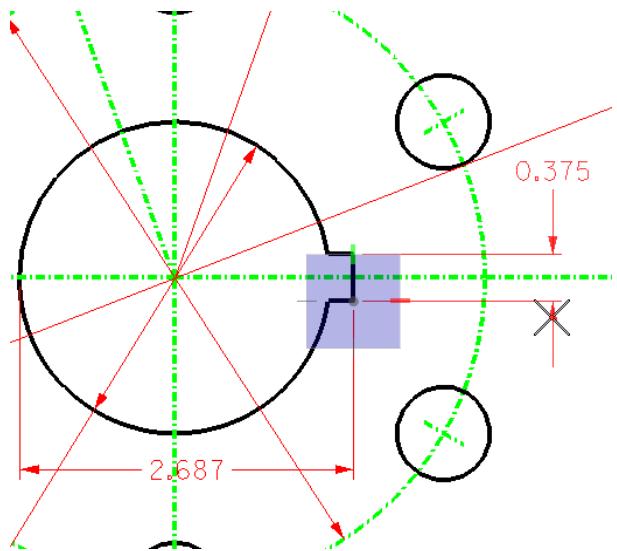


Figure 6.20 – Placing the Length of the Dimension

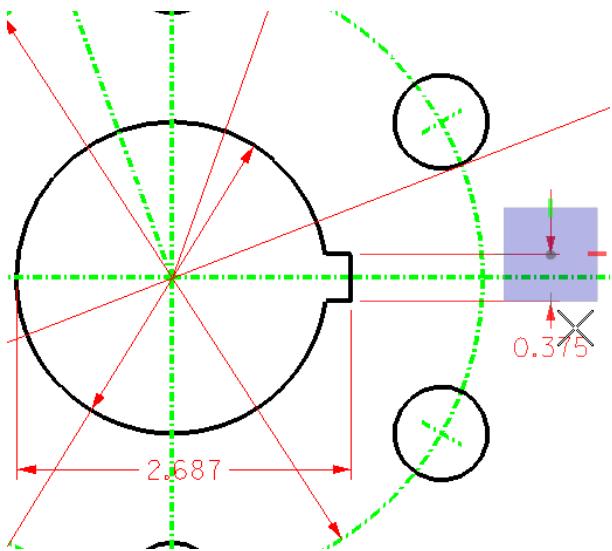


Figure 6.21 – Placing the Text

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 6.22).

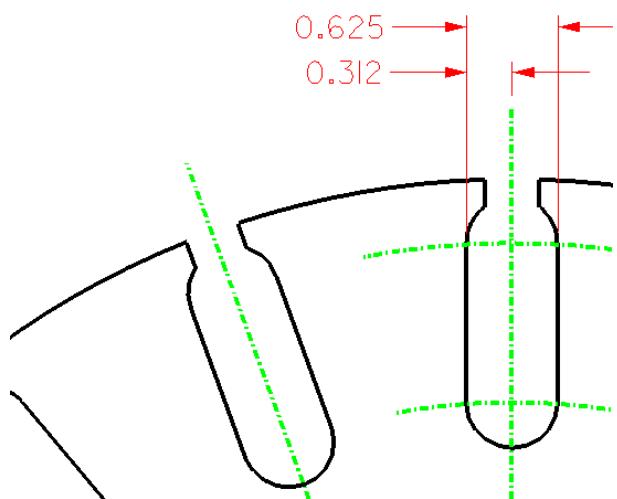


Figure 6.22 – The 0.625 and 0.312 Dimensions

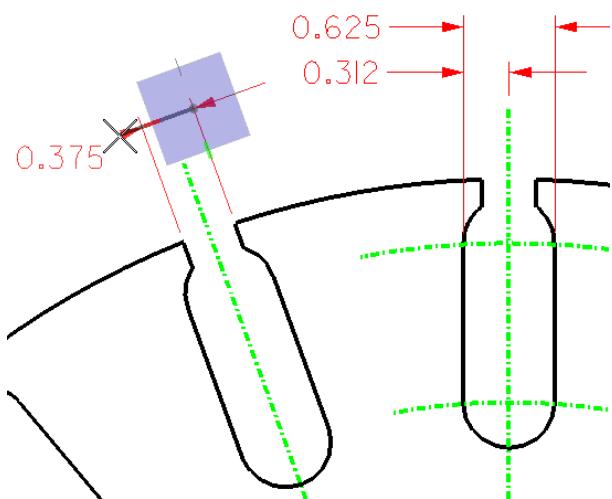


Figure 6.23 – The 0.375 Dimension

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 6.23).

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 6.24). Finally, place the dimension in position next to the 4.438 radius (Figure 6.25).

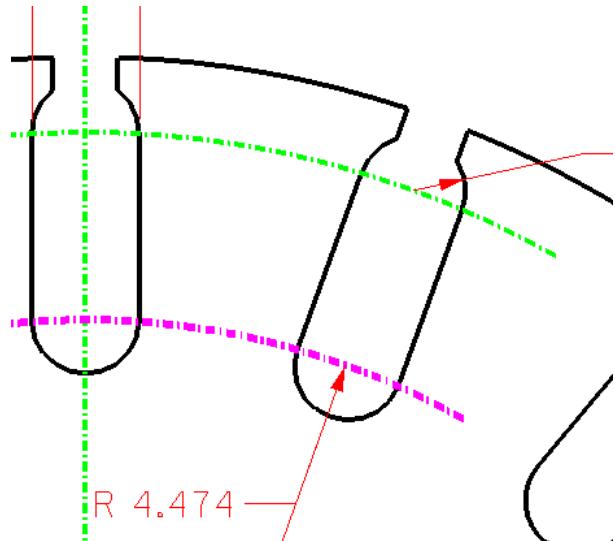


Figure 6.24 – Selecting the Inner Center Arc

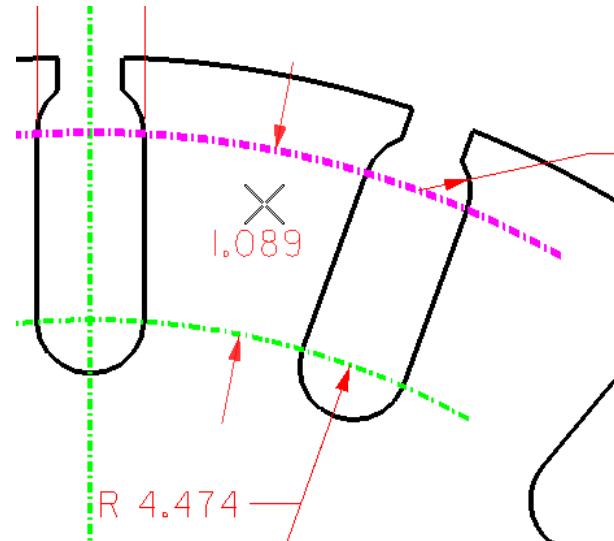


Figure 6.25 – Placing the Dimension

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 (Figure 6.26). When the AccuDraw compass appears, select the center of the part to define the center of the angle (Figure 6.27). Next, , select the top of the top centerline to set a 60° dimension. Finally, pull the dimension to the outside of the part so that it does not interfere with any other dimensions (Figure 2.28). 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 6.29). Now right-click twice to reset the command.

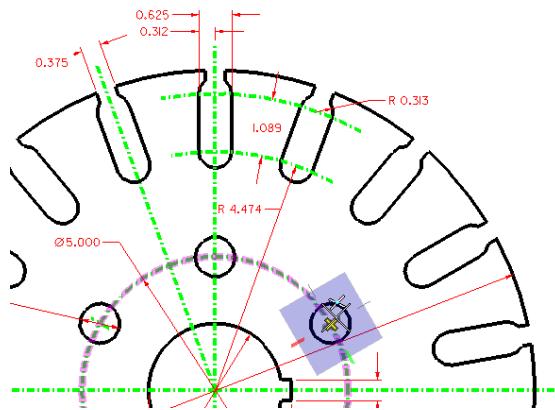


Figure 6.26 – Setting the Extension Line Length

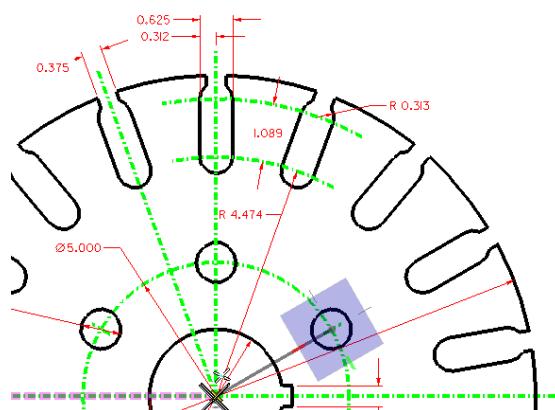


Figure 6.27 – Selecting the Center

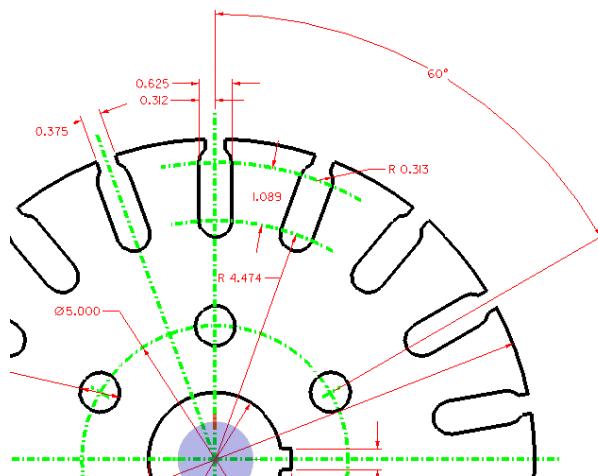


Figure 6.28 – The 60° Dimension

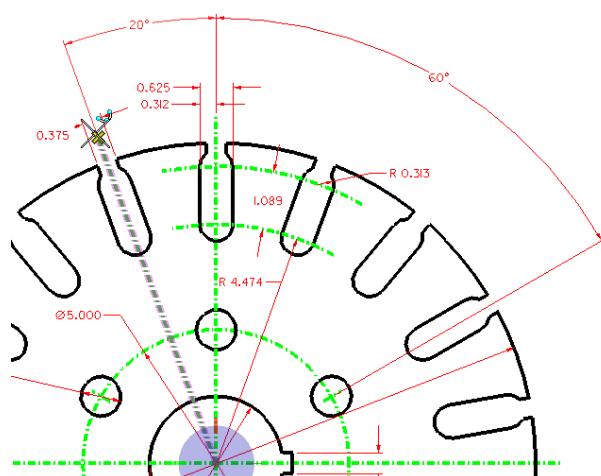


Figure 6.29 – The 20° Dimension

* 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 6.30).

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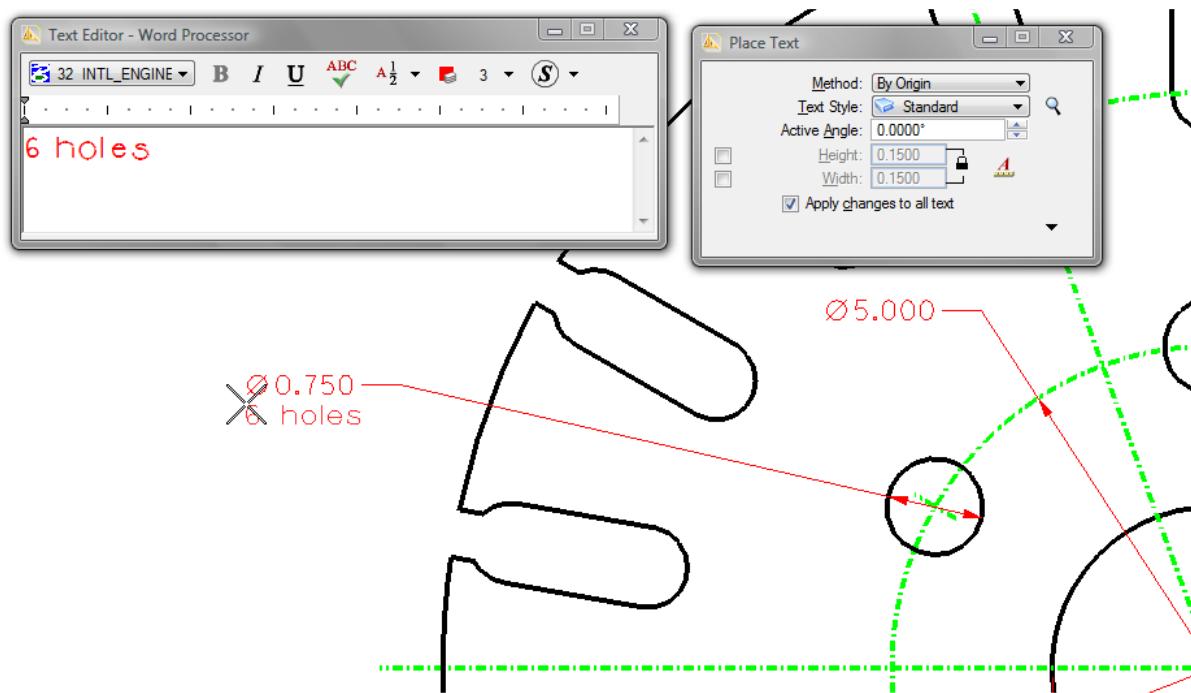


Figure 6.30 – Placing Text

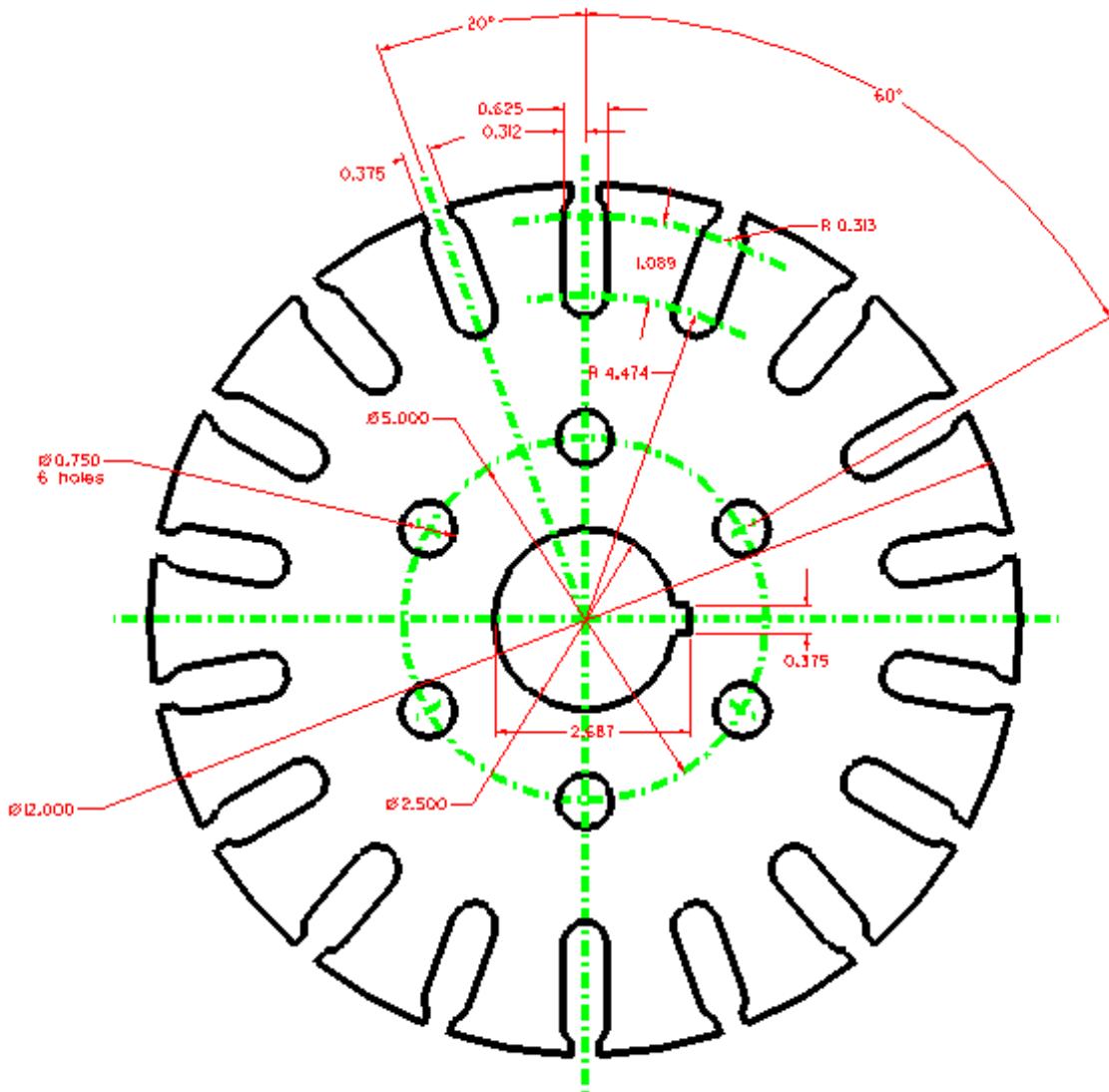


Figure 6.31 – The Dimensioned Circular Problem

* 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.