Chapter

6

Dimensioning the Circular Problem

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

- **The Dimensioning Style Window**
- **Trimming Centerlines with the Trim Tool**
- **Using the Diameter Dimension Tool**
- **Using the Linear Dimension Tool**
- **Using the Radius Dimension Tool**
- **Using the Aligned Dimension Tool**
- **Using the Angular Dimension Tool**
- **More Linear Dimensions**
- **Another Aligned Dimension**
- **Another Radius Dimension**
- **Using the Properties Tool to Modify a Dimension Variable**
- **Using the Edit Text Tool to Add a Note to a Dimension**

A Basic Discussion of the Dimensioning Style Window
_____
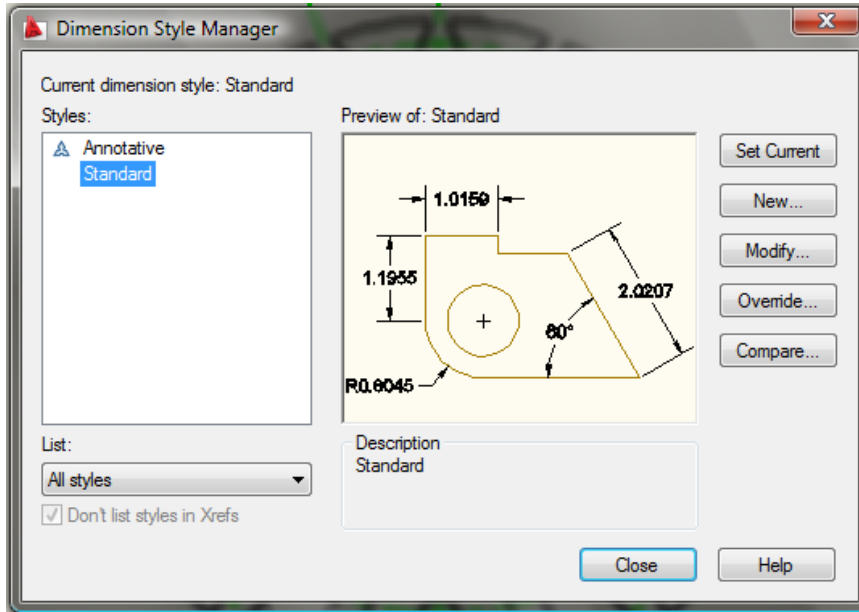
Relating an orthographic view to a machinist or construction worker with the AutoCAD system is easy and accurate. You can always control precision of the dimension with the Standard style already loaded in the software. In the Circular and the Rectangular problems, you are being exposed to a measurement control system that will enable you to build the templates for many of the drawings to come.

Figure 6.1 – Dimension Style Manager
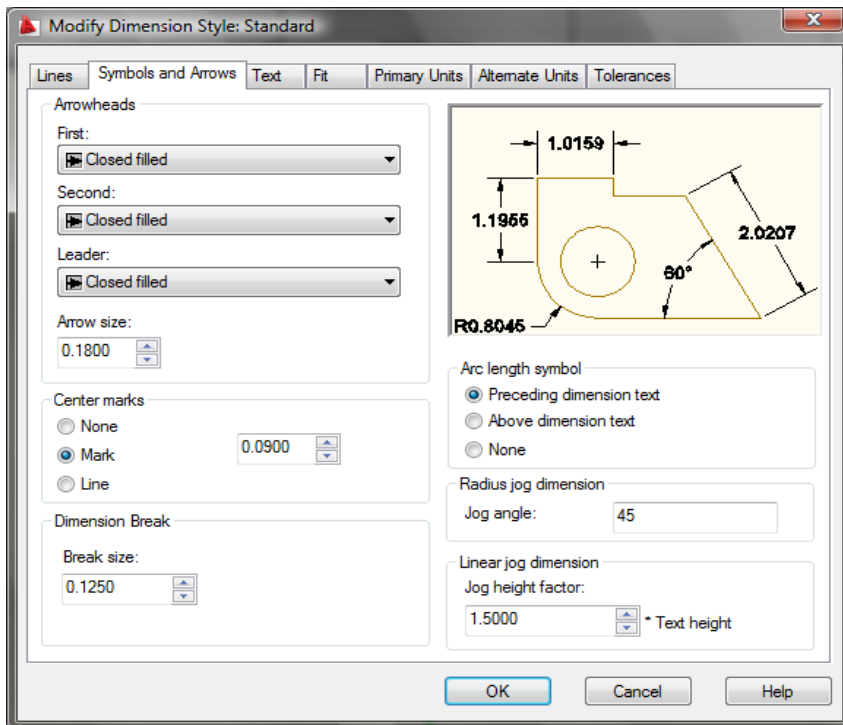
To modify the Standard dimension style which you will be using to dimension the Circular problem, select the Dimension Style tool on the Dimensions panel under the Annotate tab. The Dimension Style Manager will open as shown in Figure 6.1. Select the Modify button to make changes to the Standard style. The Symbols and Arrows tab shown in Figure 6.2 allows you to alter every part of Arrowheads Center Marks, Arc Length Symbols, and Radius Dimension Jogs.

Figure 6.2 – Modify Dimension Style, Lines and Arrows tab

For example, in an architectural drawing, you will change the Arrowheads from being Closed Filled to Architectural Tick, but most of the time this tab will be left alone.
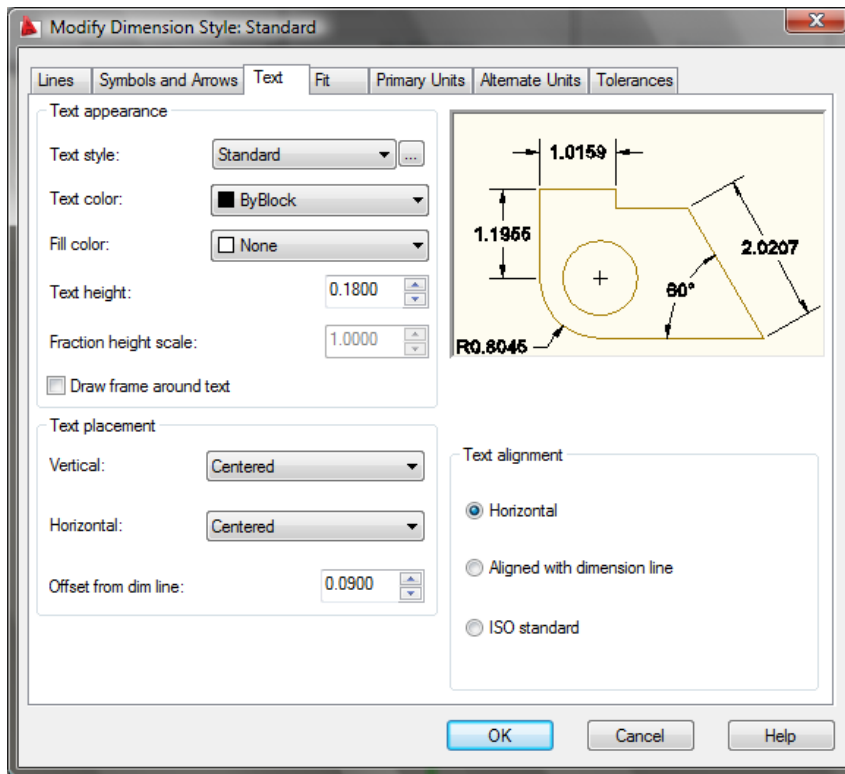
Proceeding to the Text tab, you can choose the Text Appearance, Text Placement and Text Alignment. Many of the establishments where you will work will change the Text Height to 0.937 or 0.125. The 1/8 (0.125) text height is plenty large enough to read on the print and allows the dimension to remain close to the feature that is being described. 3/32 is the smallest text can be printed that is still able to be interpreted. On the Circular drawing, leave all settings on this tab alone.
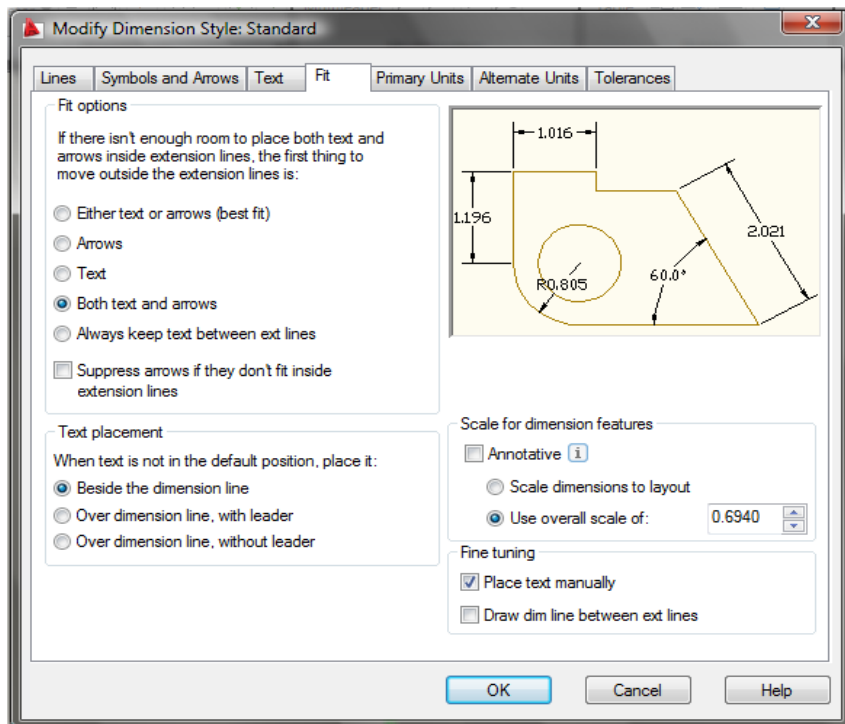
Figure 6.3 – Modify Dimension Style, Text tab

The Fit tab allows you to adjust the dimension variables to compensate for space issues. You will make a couple of changes on this tab. First select the radial button for "Both Text and Arrows". Choosing this option button will cause diameters to be flanked by regular dimension lines instead of using a leader. Also, you need to change the overall scale to 0.694 as shown in Figure 6.4 to reduce the size of the text and arrows to 0.125. Finally, select the "Place text manually" check box.

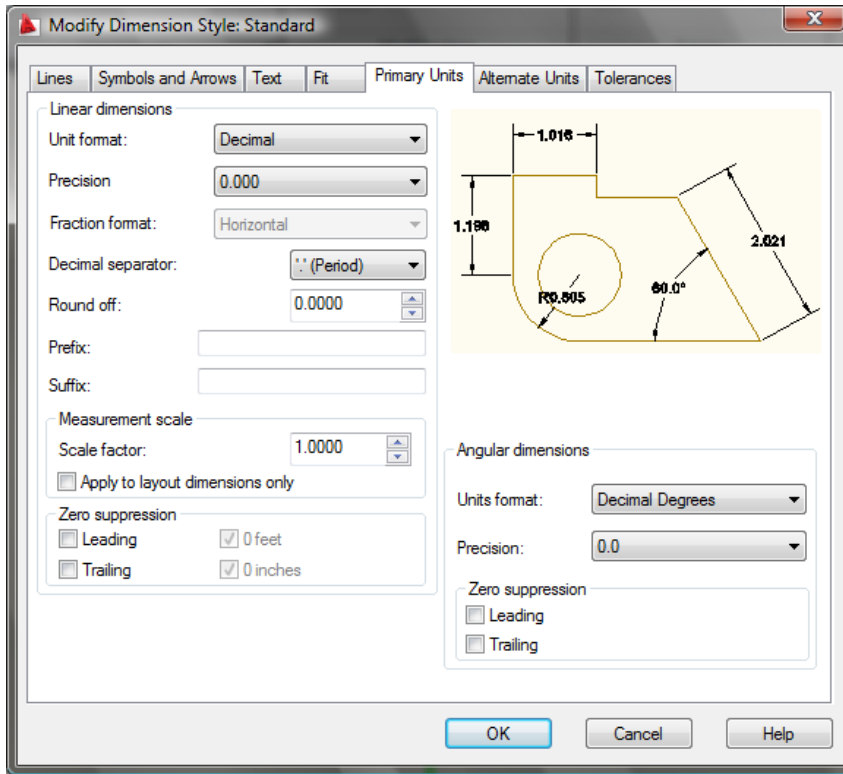**Figure 6.4 – Modify Dimension Style, Fit tab**

Figure 6.5 – Modify Dimension Style, Primary Units tab

In the Primary Units tab, the Unit format will remain Decimal, but change the Precision setting to 0.000 as shown in Figure 6.5. Also, set the Angular Precision to 0.0. The other Linear Dimension settings are used periodically, such as Round Off, but the 1.0000 Scale Factor in the Measurement Scale should stay always at one. Confusing this setting with other scale factors like line or text scale will change the value of all the dimensions in the AutoCAD drawing.
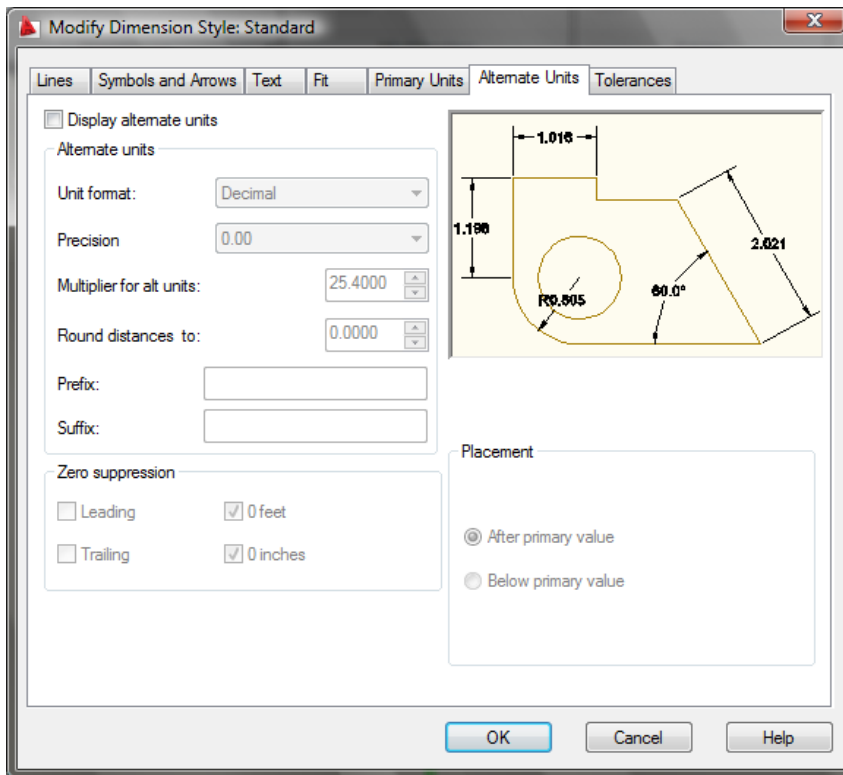

Figure 6.6 – Modify Dimension Style, Alternate Units tab

The Alternate Units tab would be used when you want to list the metric value of the English unit below or along side the dimension text. For example, if you checked "Display alternate units", a dimension showing one inch will have the alternate dimension (25.40) in parenthesis below the 1.000 text. You do not need to check the "Display alternate units" checkbox on the Circular problem.

Figure 6.7 – Modify Dimension Style, Tolerances tab
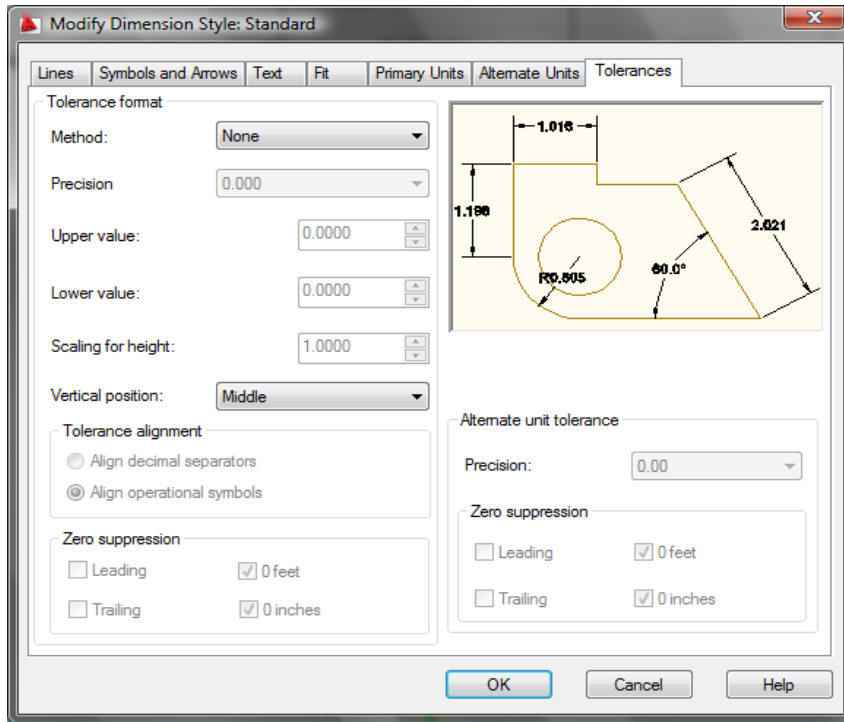
The Tolerance tab is an option mostly used by a Mechanical Designer or Engineer, where discussing the range of allowable error in the part when manufacturing or accepting the components at the production site. Right now this is being addressed in the notes that will be typed in every drawing you do. As parts become more complex, this feature will be utilized to correctly show detailed dimensions. (Figure 6.7)

To save the change that you made in the Modify Dimension Style window, select the OK button and then the Close button on the first window (see Figure 6.1).
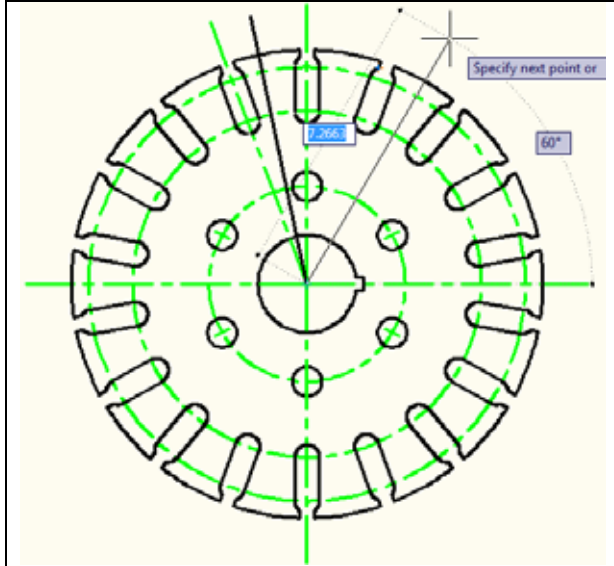
Trimming Centerlines with the Trim Tool

Figure 6.8 – Adding a cutting line
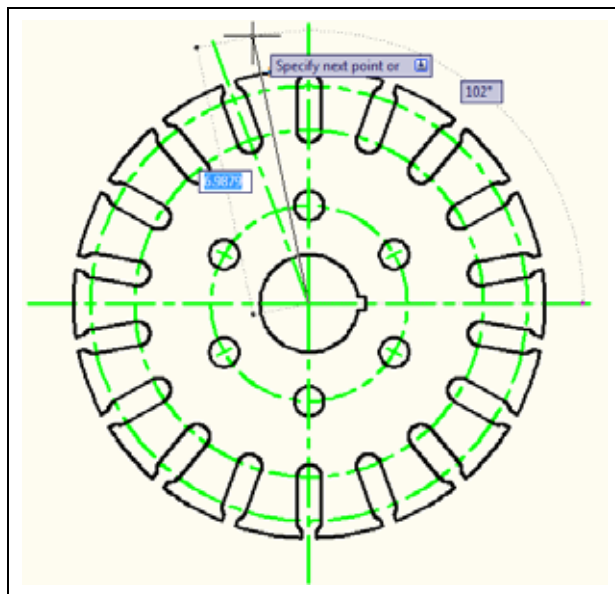
Before dimensioning the Circular problem, you will cut away some excess centerlines that may cause confusion for the reader. This is accomplished by drawing two regular lines from the center of Circular problem past the outer edges of the part. You will have to hit F8 to turn Ortho Mode off before drawing these lines. Select the Line tool on the Draw panel, and using the Center Osnap, select the center of the part. End the cutting line that you are drawing just to the upper left position of the top slot as shown in Figure 6.8. Repeat the routine, drawing the second line two slots over as shown in Figure 6.9.

Figure 6.9 – Adding a second cutting line

Now select the Trim tool on the Modify panel. At the cutting edge prompt, pick the two lines you have just drawn and press Enter to proceed with the command. Now select the two most outer circular centerlines on the outside of the cutting lines so that only the interior arc centerlines remain, as shown in Figure 6.10. Notice the rest of the outside of the orthographic view is rather clean in appearance. This trimming technique aids drawing clarity when placing dimensions.
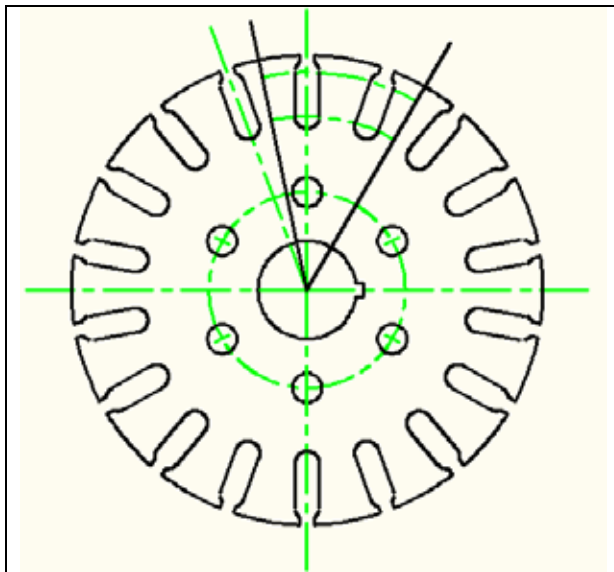
**Figure 6.10 – Trimming the centerlines**

Next select the Erase tool from the Modify panel and pick the two cutting lines drawn at the beginning of the trimming exercise. You will notice that you have rarely used the Erase tool in any of the exercises in this textbook. Erasing lines is a task that still comes from a user making an error and having to correct for this oversight. So, erasing may be done as often or as seldom based upon the efficiency of the drawer. If you are making lots of errors, as World Class CAD the saying goes, "You have to go slower to go faster." Accuracy counts.
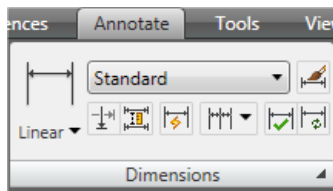
Figure 6.11 – The Dimensions Panel

Using the Diameter Dimension Tool

_____

Dimensioning is the easiest task to be performed in CAD and Diameter is the easiest dimension to place. Before you begin, make sure to change the active layer to Dimension by selecting it from the Layer Control list box in the Later panel under the Home tab. Next, select the Diameter dimension tool from the drop-down menu on the Dimensions tab. Select the 12.0 inch diameter circle and the dimension lines, arrows and text will instantly appear. Grab the dimension text and place the 12.000 toward the right side of the drawing as shown in the Figure 6.12. You do not have to position the text exactly like in the example drawing, but for right now try to be close, so you do not run into any dimension layout conflicts.
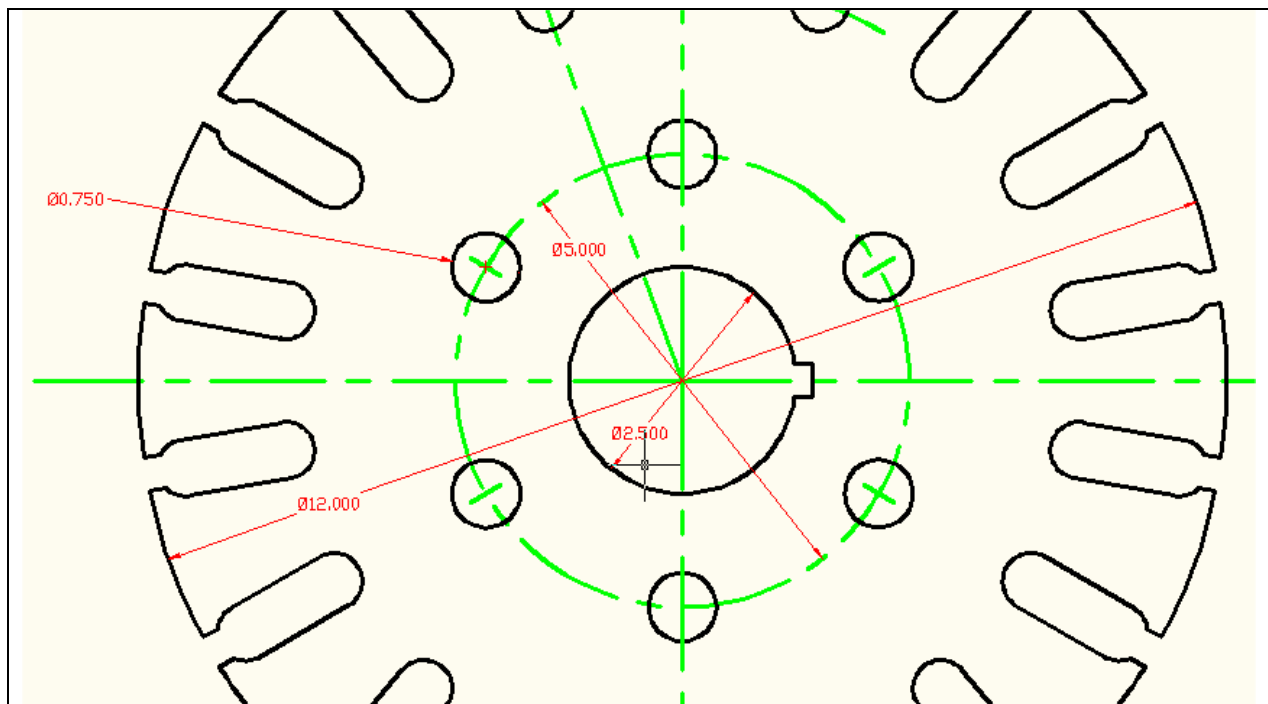

Figure 6.12 – Dimensioning using the Diameter Dimension Tool

Hit ENTER to repeat the Diameter dimension tool and pick the 5.000 inch bolt circle centerline and place the text to the upper left area as shown. Repeat the process again by choosing the innermost circle and place the 2.500 text as displayed above. The final diameter to be shown is for the 0.75 inch hole. Pick the upper left 0.75 hole and drag the 0.75 text to the outside the perimeter of the part. Again you will see that your efficiency will increase in the use of any CAD program if you repeat a procedure. By placing all the diameters on the drawing, you are less likely to forget to place a dimension on the drawing. A mistake like this will cause a machinist or construction worker to have to wait for your missing data to proceed.

Using the Linear Dimension Tool

_____

Most circular parts are designed with a keyway so that a key can be installed between the shaft and the hub to prevent slippage. A keyway is referenced from the opposite quadrant of the circle, so to place the horizontal dimension, select the Linear Dimension tool on the Dimension panel and pick the west quadrant of the circle and the bottom endpoint of the keyway.
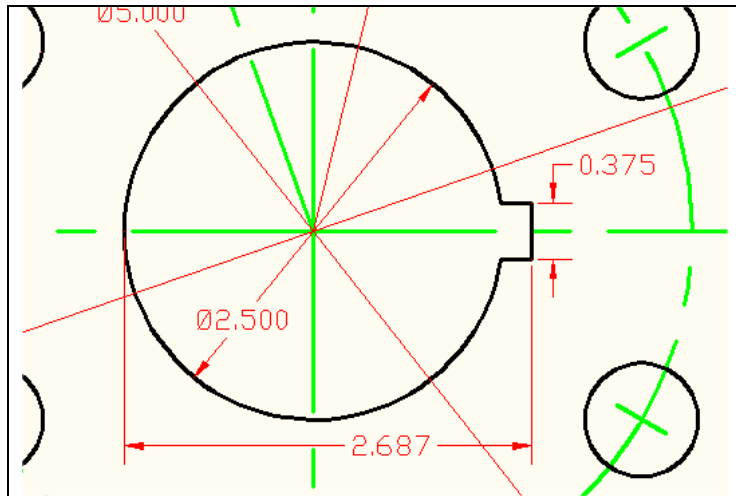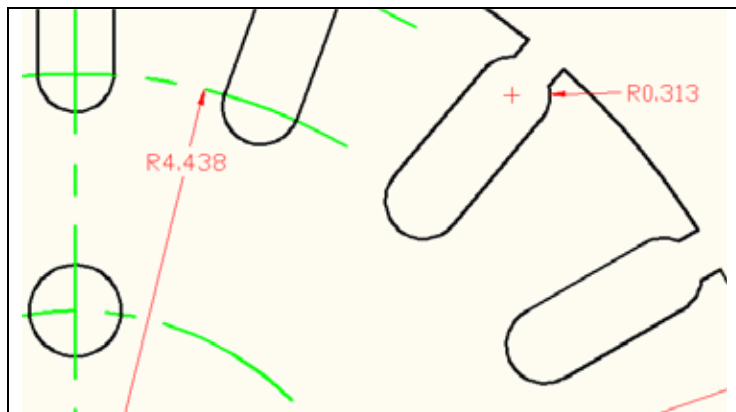

Never allow a dimension arrow or text to be obstructed by a center, hidden, dimension or object line, so take your time to move dimensions off any entity that will cause confusion. Now dimension the width of the keyway as shown in Figure 6.13 and pull the 0.375 text into place.

Figure 6.13 – Using the Linear Dimension Tool

Using the Radius Dimension Tool

_____

To describe the inner bolt circle of the slot, place a radius on the inner arc as shown in Figure 6.14. Select the Radius Dimension tool on the Dimension panel. The mouse pointer will change from an arrow to a pick box to allow you to choose an arc or circle for which to place a radius.

Figure 6.14 – Using the Radius Dimension Tool

Select anywhere on the green inner arc. You can move the radius dimension line back and forth moving the dimension arrow to the desired location between two slots. By placing the radius dimension as shown above, the Aligned Dimension can be added to measure the distance to the second arc.

Using the Aligned Dimension Tool

_____

The Aligned Dimension tool is used to easily place a dimension in alignment with two picked reference points. You are going to use the Aligned Dimension command to place a dimension on the drawing to define the distance from the end of the 4.437 radius to the other centerline arc. Make sure that you have the Perpendicular Osnap toggled on if it is not already.
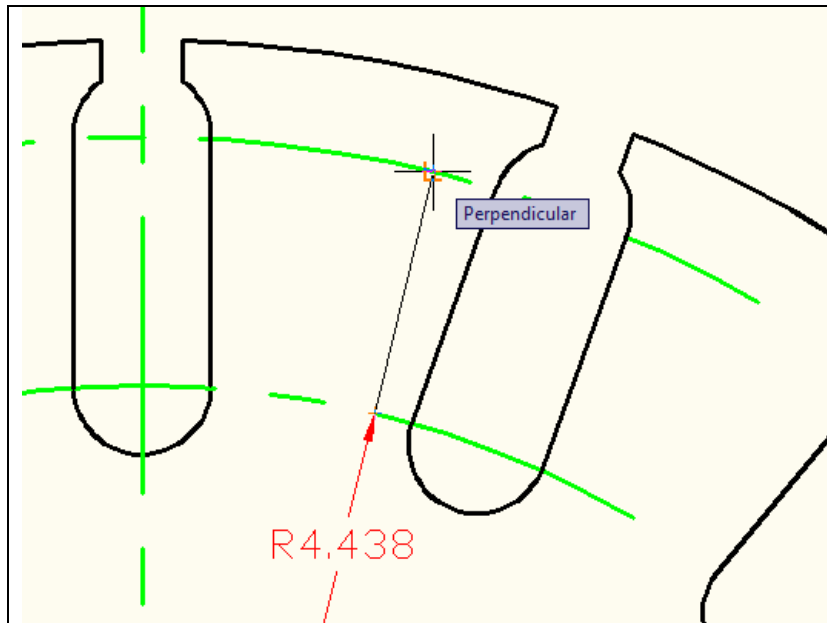
Pick the Aligned Dimension tool on the Dimension panel, and select the endpoint of the R4.437. For your second point place the aperture over the second arc and wait to see the Perpendicular Osnap symbol, as shown in Figure 6.15. Pick the second arc and a 1.125 dimension will appear that is aligned with the 4.437 Radius dimension. (See Figure 6.16)

**Figure 6.15 – Using the Aligned Dimension Tool**
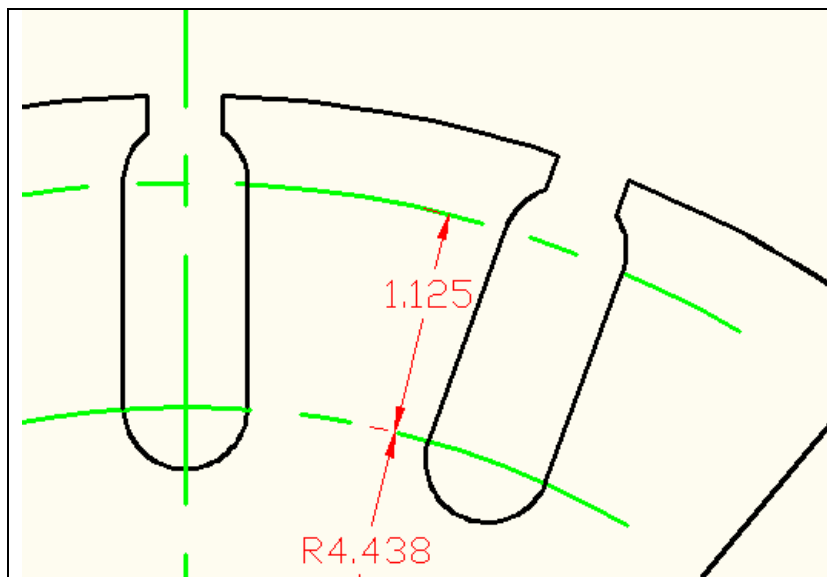
Many Engineers and Designers will want to place a second radius dimension on the second arc, but the reference and the tolerance for the feature will be from the wrong location. Slots like these will intermix with other parts. The dimension controlling these surfaces is based off the 1.125 dimension, so use an Aligned Dimension to explain the part.

Figure 6.16 – Using the Aligned Dimension Tool
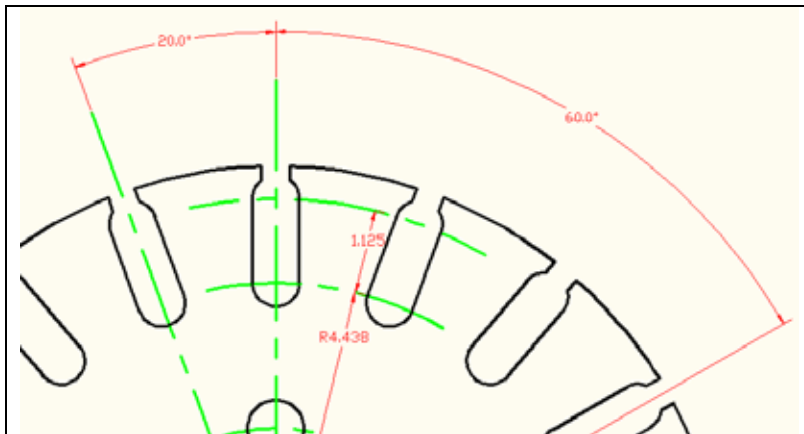
Using the Angular Dimension Tool

_____

There are two angular dimensions to place in the Circular drawing. Remember how easy the command is to use? Select the Angular Dimension tool on the Dimension panel and select two converging lines (non-parallel) to place the measurement on the drawing.
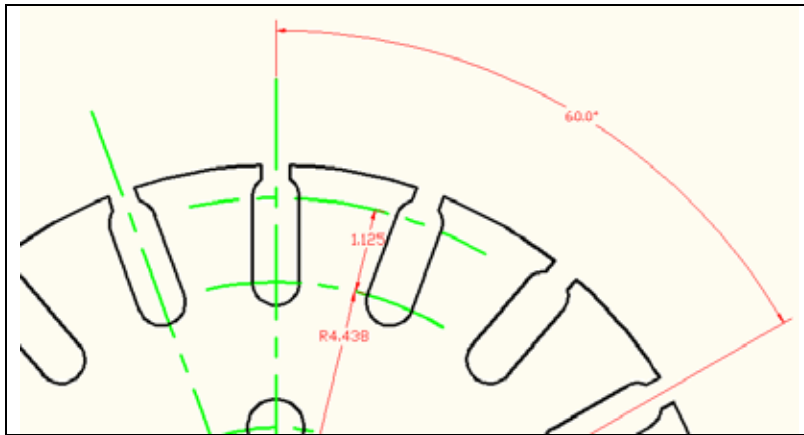
For the first dimension, select the green center mark on the upper right 0.75 hole. For the second pick, select the vertical centerline at the top of the drawing. You will need to make a third selection to place the 60.0-degree dimension text on the drawing (Figure 6.17).

Figure 6.17 – Using the Angular Dimension Tool

The last Angular dimension on the drawing is the 20.0 degree measurement between each of the 18 slots. After selecting the Angular Dimension tool, pick the top of the vertical centerline again and then the peak of the centerline over the slot to the left as shown in Figure 6.18.

Figure 6.18 – Using the Angular Dimension Tool

When placing a dimension on the outside perimeter of a part or a building detail, remember, you may want to place more dimensions below that measurement. In the case of the Circular problem, the Angular dimensions are approximately 1.5 inches off the outer surface of the 12.0 diameter. That brings you to detailing the slot opening and width.

More Linear Dimensions

_____

The last series of dimensions are located at the top quadrant of the Circular problem. Select the Linear Dimension tool on the Dimension panel to place two of these

remaining measurements. For the 0.313 dimension, pick the vertical centerline and the outside edge of the slot. Repeat the command for the 0.626 dimension, picking both outside lines on the slot as revealed.

Figure 6.19 – Using the Linear Dimension Tool
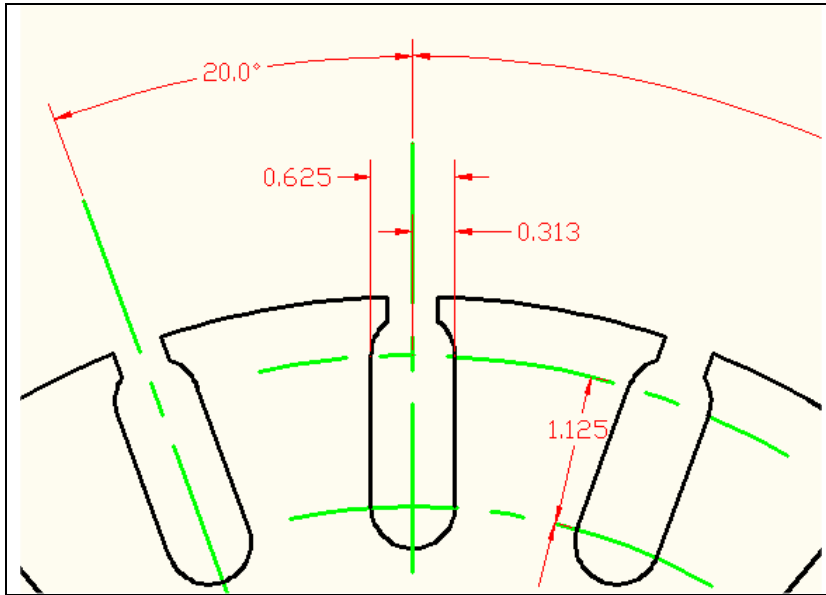
In many cases you may need to reference a detail like the slot from the centerline and then define the edge of the slot to the opposite side with another dimension. If you use the 0.313 dimension to define both sides, the tolerance will double when calculating 0.313 twice. So in the case in Figure 6.19, place a linear dimension for 0.625 as shown.

Another Aligned Dimension

_____

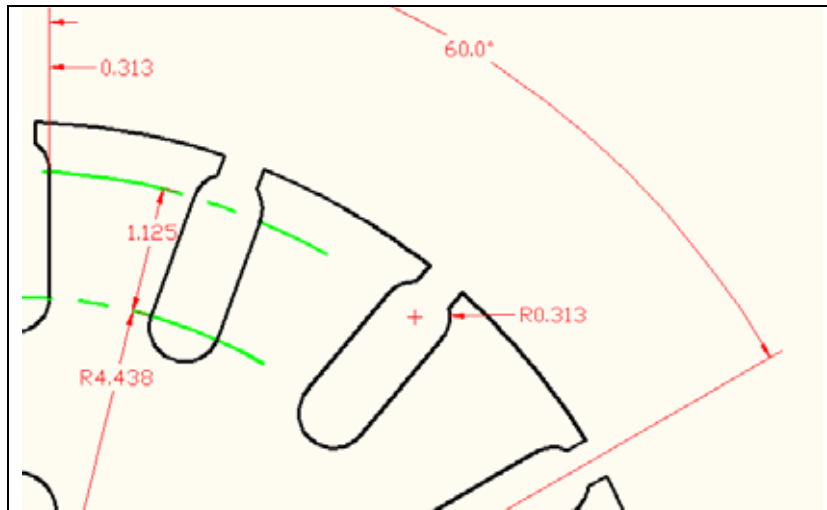**Figure 6.20 – Using the Aligned Dimension Tool**
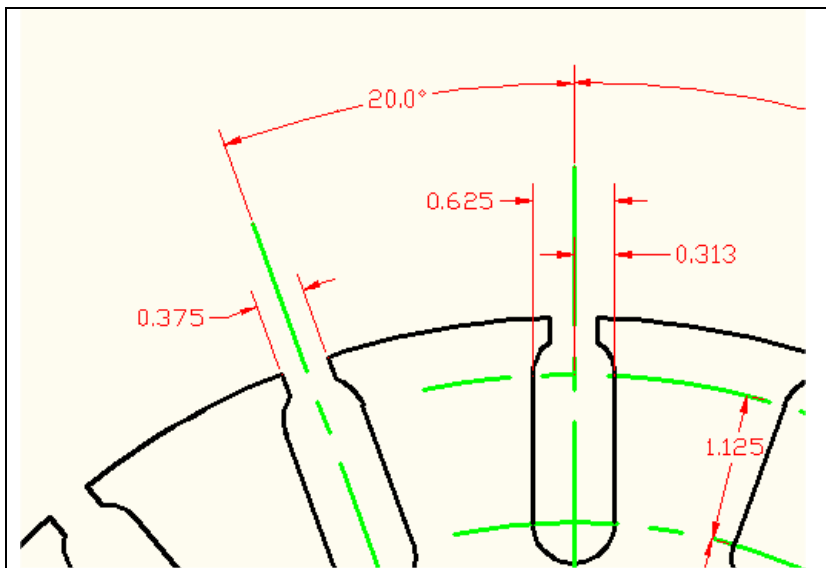
Choose the Aligned Dimension tool on the Dimension panel and select the endpoint of the opening of the slot as shown in Figure 6.20. By picking the line on the right followed by the line on the left, the dimension text will appear on the side picked second.

Another Radius Dimension

_____

The last dimension on the Circular problem is the 0.313 radius on the slot. Select the Radius Dimension tool located on the Dimension panel. The mouse pointer will change from an arrow to a pick box to allow you to choose an arc or circle in which to place a radius.

Figure 6.21 – Using the Radius Dimension Tool

Pick the arc on the slot with R0.313 dimension, as shown in Figure 6.21, and for the next selection pick the location of the dimension text.

* World Class CAD Challenge 02-14 * - 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, 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 Properties Tool to Modify a Dimension Variable

_____

When you need to make special changes from your Standard Dimension Style, you can modify an object through its Properties window. Many times you will leave the center mark option on the Lines and Arrows tab (Figure 6.2) To bring up the Properties window, select any object and right-click. Select Properties from the window and the Properties window will appear.

With the Properties window visible, pick the R0.313 dimension as shown in Figure 6.22. Expand the Lines and Arrow category and find Center Mark. In the list box, change the setting from Mark to None to remove the center mark in the associative dimension.

For the second modification, select the 1.125 aligned-dimension as shown in Figure 6.23 and the Properties window will change for that measurement. To remove the two small dimension extension lines that are at the points of the dimension arrows, double click on the On for Ext 1 and the text will change to OFF. Repeat the same process for the Ext 2 to change the text to OFF. Click on the Close button (X). For the second modification, select the 1.125 aligned-dimension as shown in Figure 6.23 and the Properties window will change for that

measurement. To remove the two small dimension extension lines that are at the points of the dimension arrows, double click on the On for Ext 1 and the text will change to OFF. Repeat the same process for the Ext 2 to change the text to OFF. Click on the Close button (X).
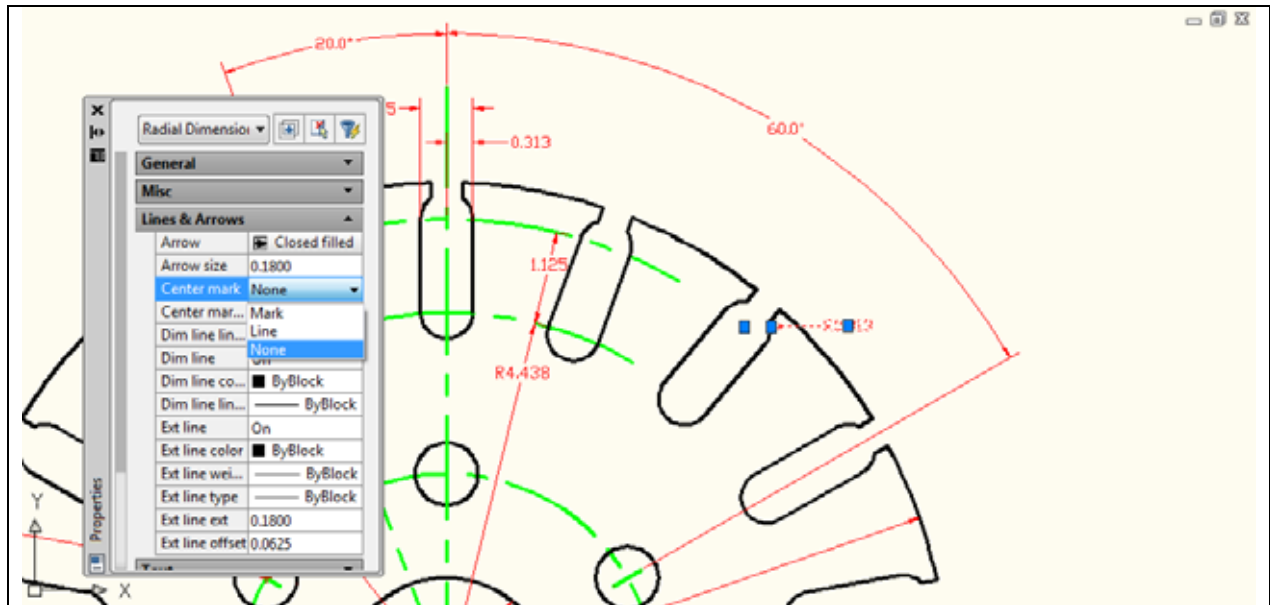

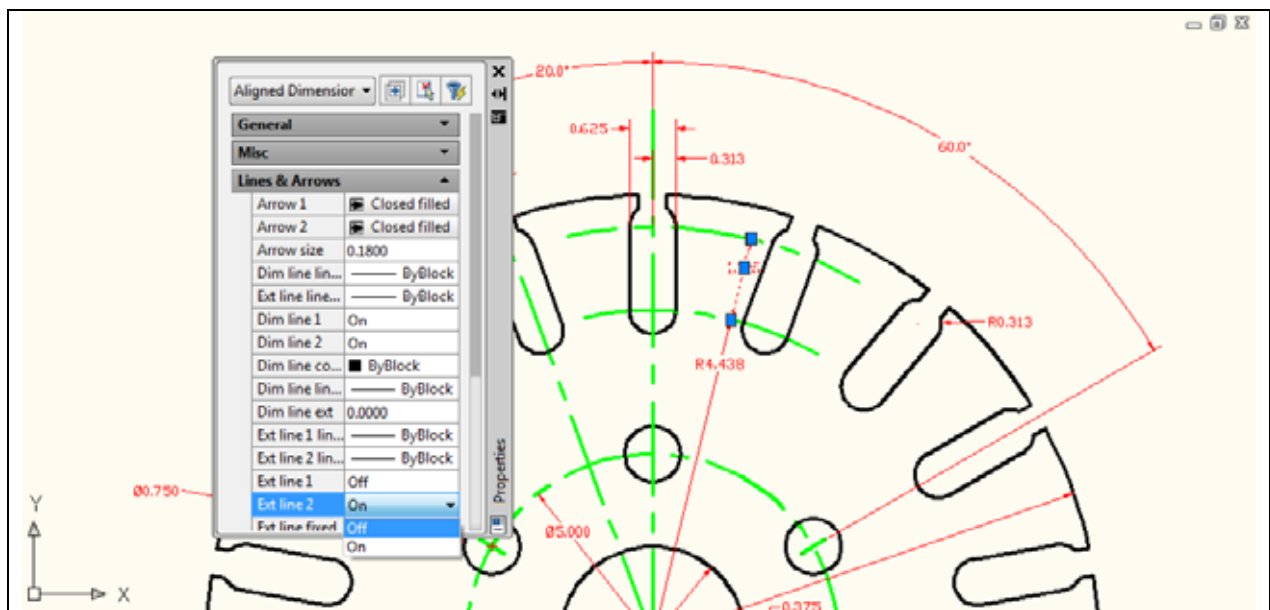**Figure 6.22 – Modifying Dimensions Using the Properties Tool**

**Figure 6.23 – Modifying Dimensions Using the Properties Tool**

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

_____

Select the Edit Text tool from the Menu Browser. Select the .750 radius dimension, and the Multiline Text tab will appear, along with a text box around the .750. With your mouse, place the text insertion bar after the dimension and hit and Enter to go to the second line. Type the text 6 HOLES and click away on the screen. The small note will be seen below the 0.75 text. (See Figure 6.24)
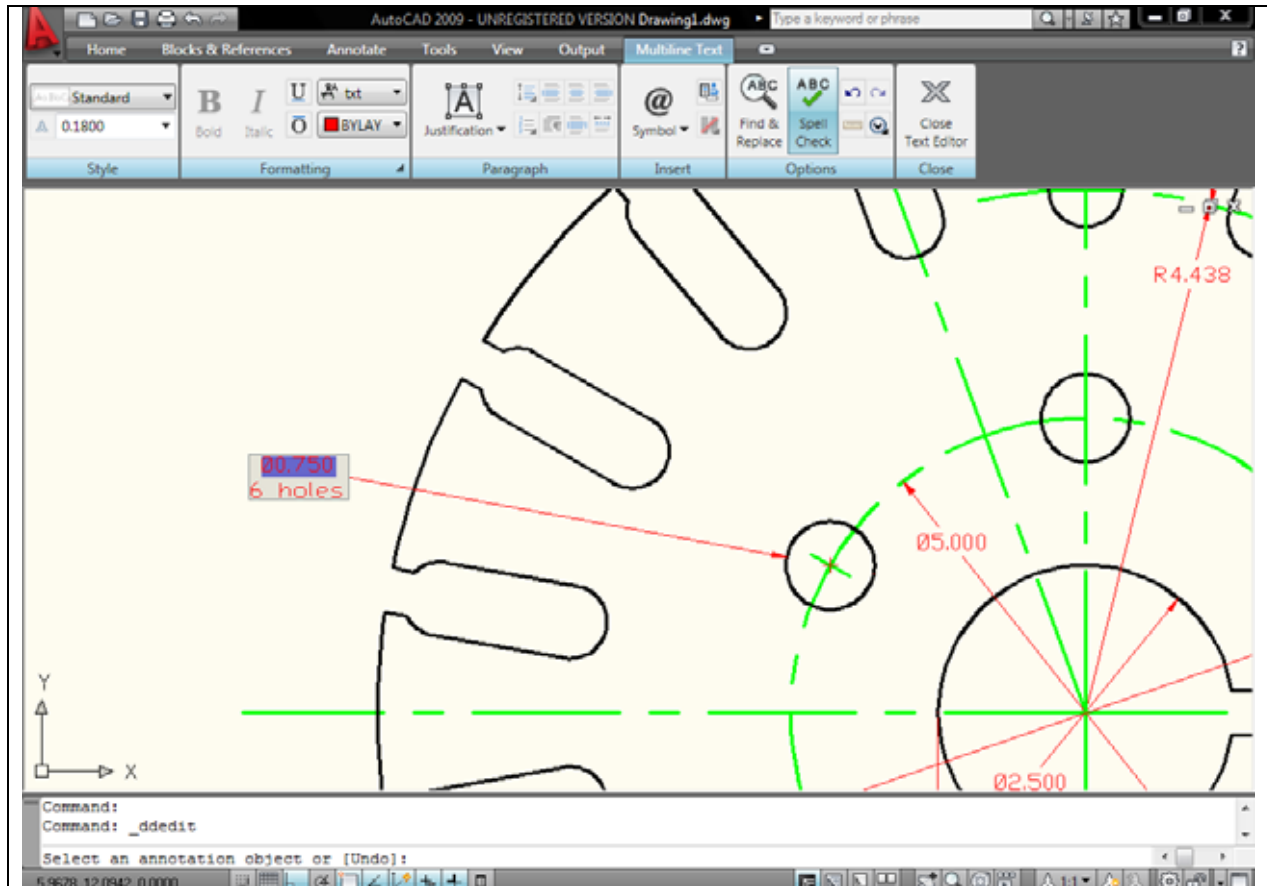
Figure 6.24 – Modifying Text using the Multiline Text Editor

The Circular problem is finished and you should be satisfied that all the critical information is available for viewing. In Figure 6.25, you can see the entire drawing without manufacturing notes. As in the Rectangular problem, the notes become significant in providing the material, machining information, coating requirements and tolerances. In Chapter 7 you will be shown the steps to completing your set of notes.
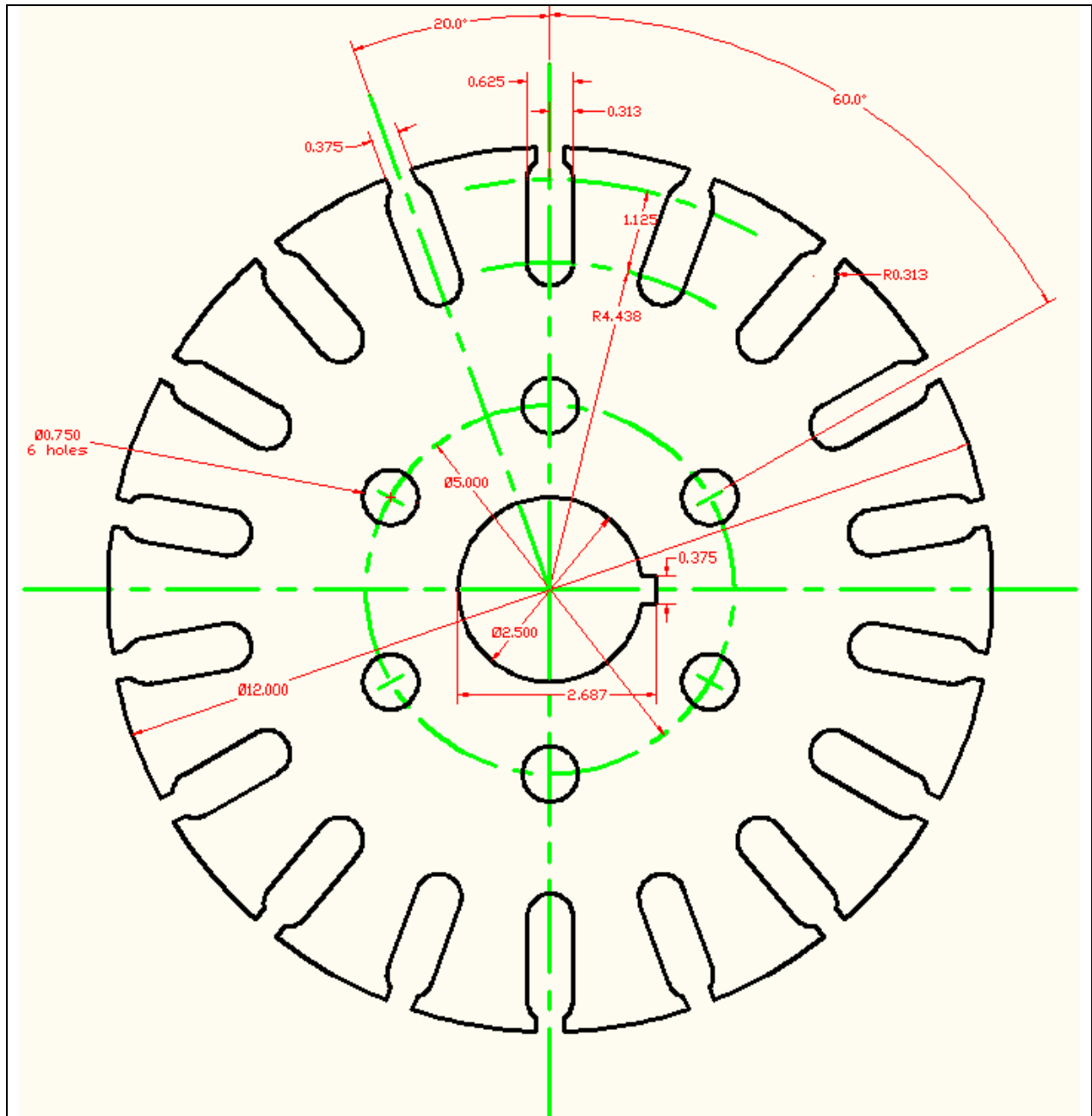
Figure 6.25 – Finished Dimensioned Circular Problem

* World Class CAD Challenge 02-15 * - 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, add the 14 dimensions and modify the 3 properties of the dimensions in 10 minutes. Continue this drill four times, each time completing the drawing under 10 minutes to maintain your World Class ranking.