C h a p t e r

6

Dimensioning the Circular Problem

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

1. A Basic Discussion of the Dimensioning Style Window
2. Trimming Centerlines with the Trim Tool
3. Using the Diameter Dimension Tool
4. Using the Linear Dimension Tool
5. Using the Radius Dimension Tool
6. Using the Aligned Dimension Tool
7. Using the Angular Dimension Tool
8. More Linear Dimensions
9. Another Aligned Dimension
10. Another Radius Dimension
11. Using the Properties Tool to Modify a Dimension Variable
12. Using the Edit Text Tool to Add a Note to a Dimension
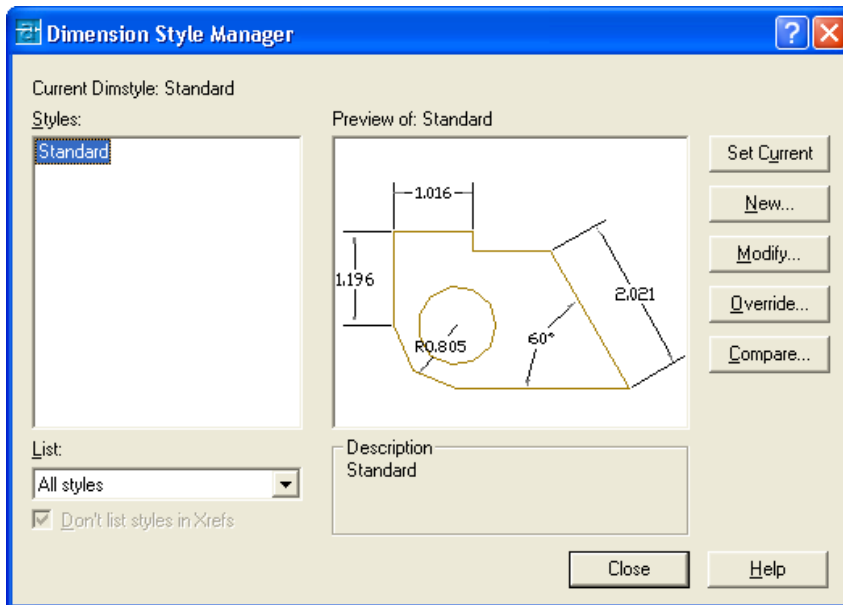
A Basic Discussion of the Dimensioning Style Window
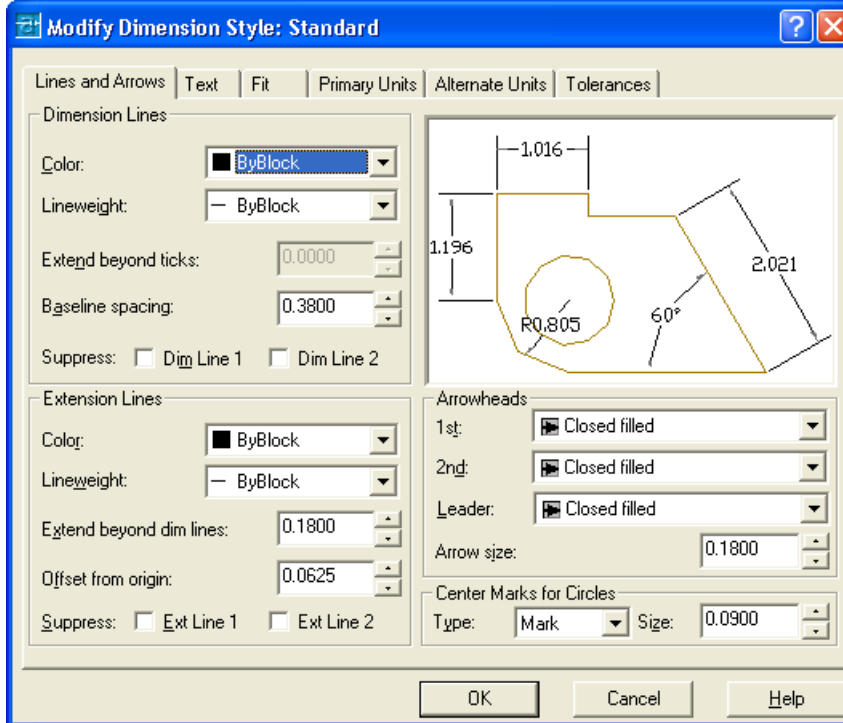
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Figure 6.1 – Dimension Style Manager

Relating an orthographic view to a machinist or construction worker with the AutoCAD system is exceptional and 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 system of measurement control that will enable you to build the templates for many of the drawings to come.

Figure 6.2 – Modify Dimension Style – Lines and Arrows

To control the Standard dimension style which you will be using to dimension the Circular problem, select the Dimension Style tool on the Dimension toolbar and the Dimension Style Manager will open as shown in Figure 6.1. Select the Modify button to make changes to the Standard style. The Lines and Arrows tab shown in Figure 6.2 allows you to alter every part of the Dimension Lines, Extension Lines, Arrowheads and Center Marks.

For example, in an architectural drawing, you will change the Arrowheads from being Closed

Filled to Architectural Tick, but for the majority of the times this tab will be left alone.
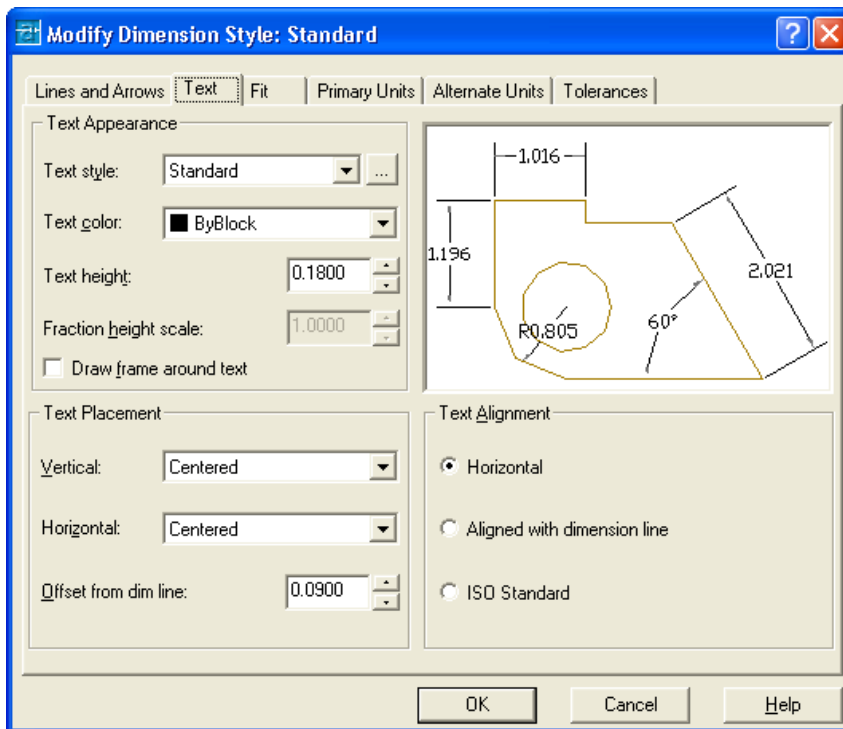

Figure 6.3 – Modify Dimension Style – Text

Proceeding to the next 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 in which is being described. 3/32 is as small of text to be printed and is still able to be interpreted. On the Circular drawing, leave all settings on this tab alone.
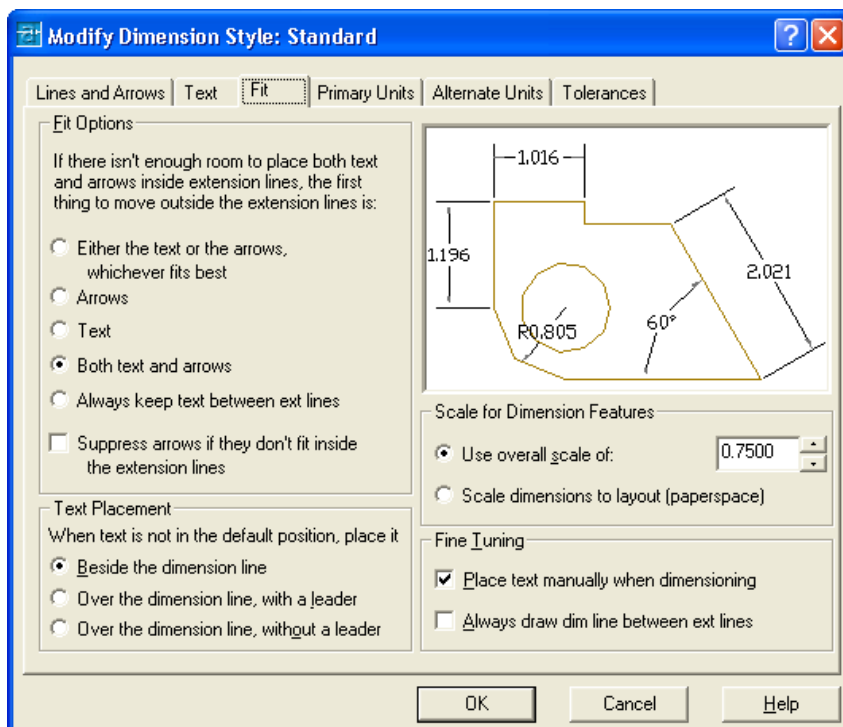

Figure 6.4 – Modify Dimension Style – Fit

The Fit tab allows you to adjust the dimension variables to compensate for room 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, the Diameters will 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.
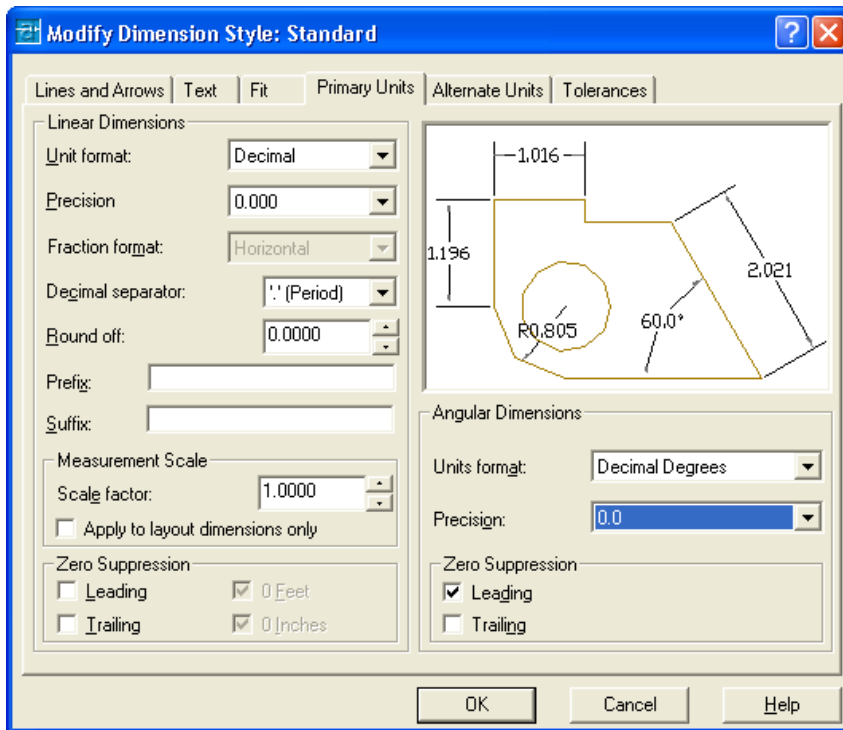
Figure 6.5 – Modify Dimension Style – Primary Units

In the Primary Units tab, the Unit format will remain Decimal, but the Precision setting will change to 0.000 as shown in Figure 6.5. 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 since confusing this setting with other scale factors like line or text scale will change the value of all the dimensions in the AutoCAD drawing.
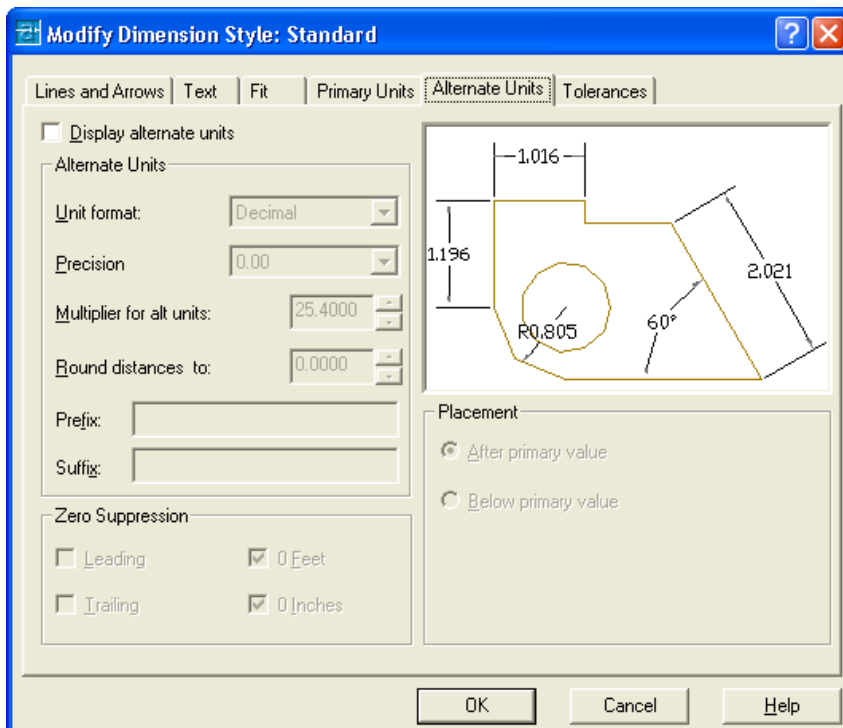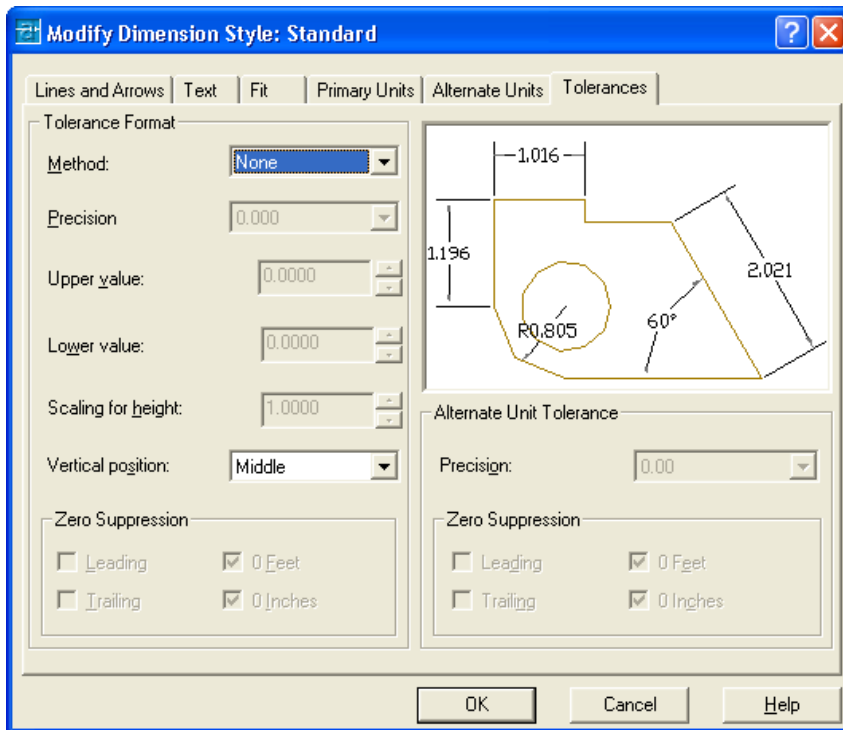

Figure 6.6 – Modify Dimension Style – Alternate Units

The Angular Dimensions are to be Decimal Degree and set to 0.0 precision to allow the reader of the print to see the angles in at least a tenth of a degree.

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 (25.40) alternate dimension in parenthesis below the 1.000 text.

You do not need to check the "Display alternate units" checkbox on the Circular problem.
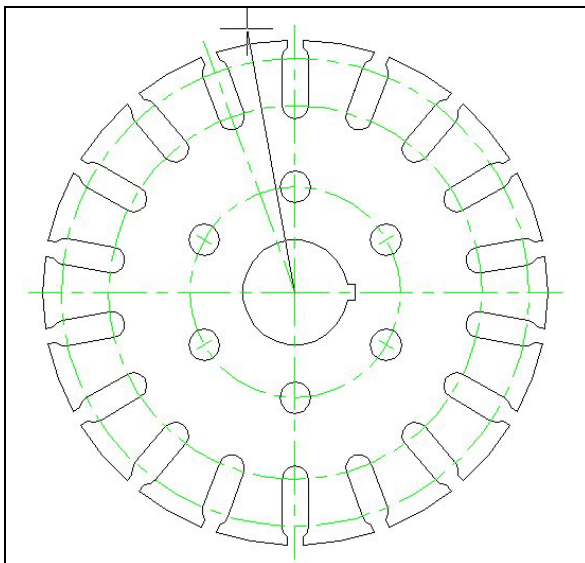
The Tolerance tab is an option mostly used by a Mechanical Designer or Engineer, where they need to discuss the range of allowable error in the part when manufacturing or accepting the components at the production site. Right now this is being addressed in the notes that will be typed in every drawing you do. As the parts become more complex, this feature needs to be utilized to correctly show the detail dimensions. (Figure 6.7)

Figure 6.7 – Modify Dimension Style – Tolerances

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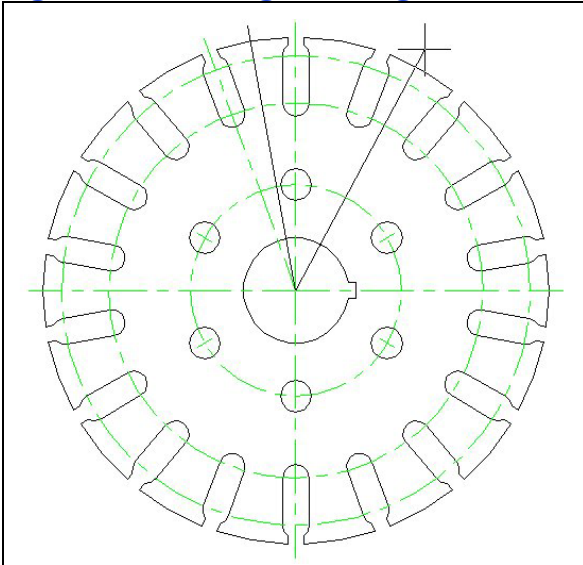
Trimming Centerlines with the Trim Tool

Before dimensioning the Circular problem, you will cut away some excess centerlines that may cause confusion for the print reader. This is accomplished by drawing two regular lines from the center of Circular problem past the outer periphery of the part. Select the Line tool on the Draw toolbar, and using the Center osnap select any circle and the line will start at 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.
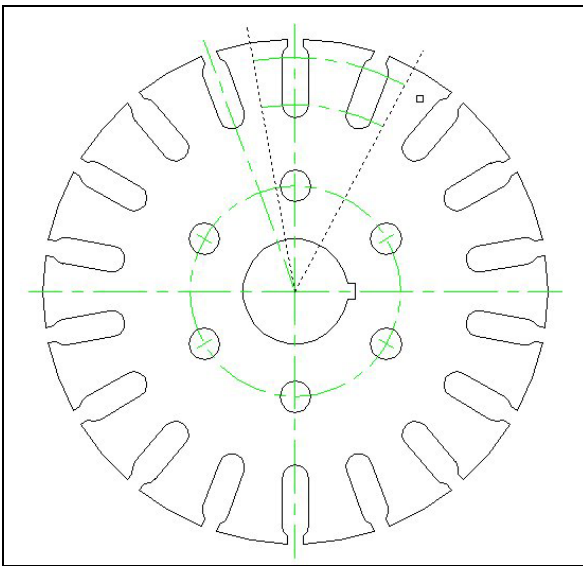
Figure 6.8 – Adding a Cutting Line

Figure 6.9 – Adding a Second Cutting Line

Now select the Trim tool on the Modify toolbar. At select 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 will pay you dividends for drawing clarity when placing in your dimensions.

Next select the Erase tool on Modify toolbar 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, at World Class CAD the saying goes, "You have to go slower, to go faster". Not so many mistakes please.

Figure 6.10 – Trimming the Centerlines

Figure 6.11 – The Dimension Toolbar

Using the Diameter Dimension Tool

Dimensioning is the easiest task to be performed in CAD and Diameter dimensioning

is the easiest. Select the 12.0 inch diameter circle and the dimension lines, arrows and text instantly appear. Grab the dimension text and place the 12.000 toward the lower left side of the drawing as shown in the Figure 6.12. You do not have to position the text exactly like in our drawing, but for right now try to be close, so you do not run in to any dimension layout issues.
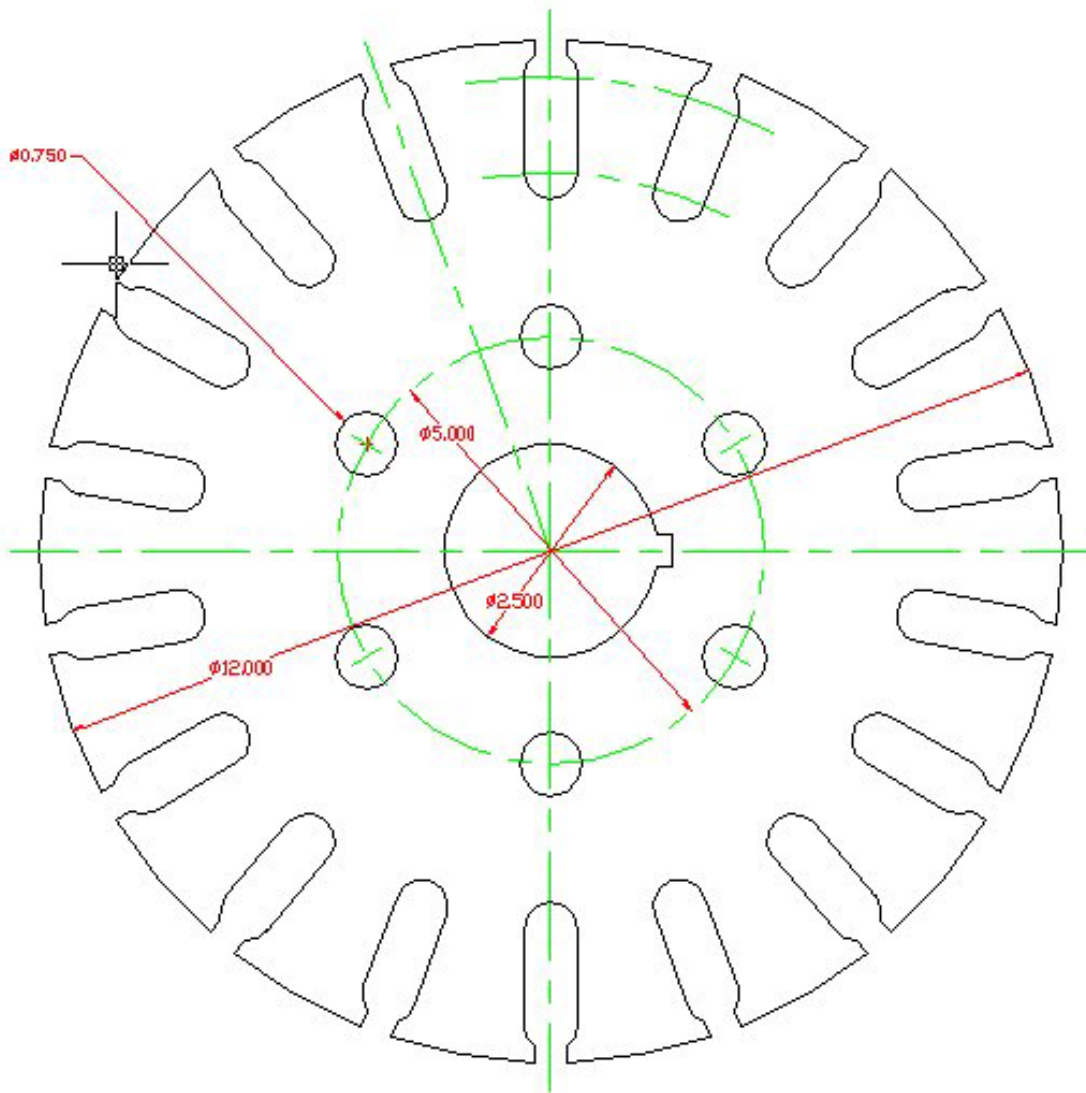

Figure 6.12 – Dimensioning Using the Diameter Dimension Tool

Now select the Diameter Dimension tool and pick the 5.000 inch bolt circle centerline and place the text to the upper left area as shown. Then repeat the process choosing the most inner circle and place the 2.500 text as displayed above. The final diameter to be shown is for the 0.75 inch hole. Select the Diameter Dimension tool, 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 have forgotten to place a dimension on the drawing which is an error of omission. A machinist or construction worker will have to wait for your missing data to proceed causing as many problems as if the dimension was

unclear.

Using the Linear Dimension Tool

On most circular parts you will discover a keyway so that a key will 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 toolbar and pick the west quadrant of the circle and the bottom endpoint of the keyway.
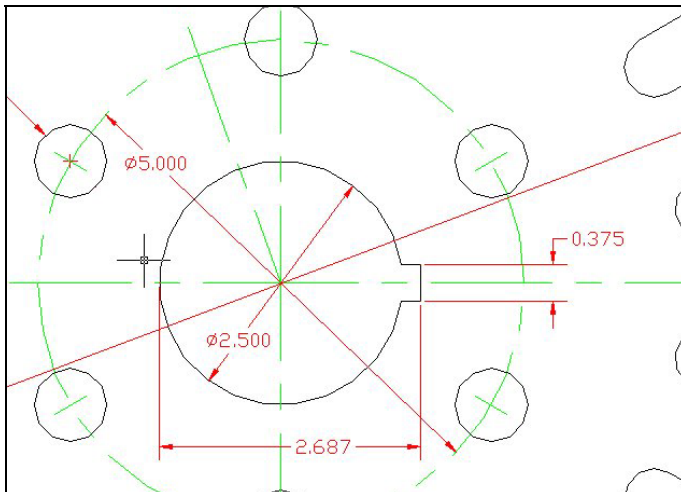

Figure 6.13 – Using the Linear Dimension Tool

The 2.687 dimension will appear on the vertical centerline, so select the 2.687 dimension and grab the text to move the numbers to the right of the line. Never allow a dimension arrow or text to be obstructed by a center, hidden, dimension or object line, so take your time to move the dimension 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.
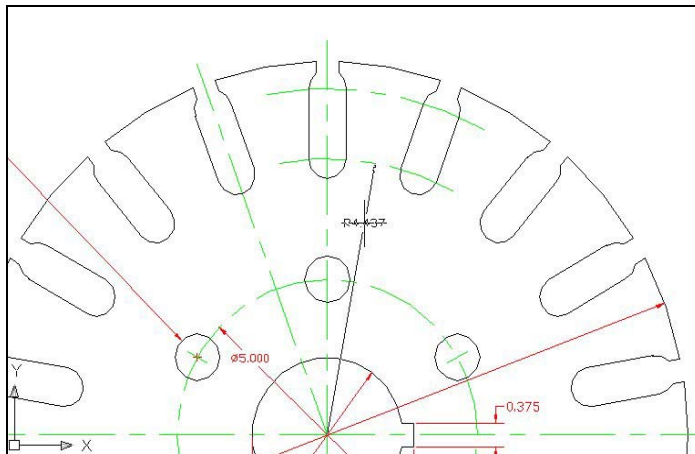
Using the Radius Dimension Tool

Figure 6.14 – Using the Radius Dimension Tool

Now to describe the inner bolt circle of the slot, you will place a radius on the inner arc as shown in Figure 6.14. Select the Radius Dimension tool on the Dimension toolbar, 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.

Select anywhere on the green inner arc and 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 and the perpendicular point opposite the end of the dimension arrow. In order to easily select the intersecting point of the centerlines, you need to change your Osnap settings.

Select the Object Snap Settings tool on the Object Snap toolbar to open the Drafting Settings window. Check the Perpendicular checkbox and press OK to close the window. Feel free to change Osnap settings at key times in the drawing process, such as between major command functions like drawing lines and circles or when moving from drawing to dimensioning.
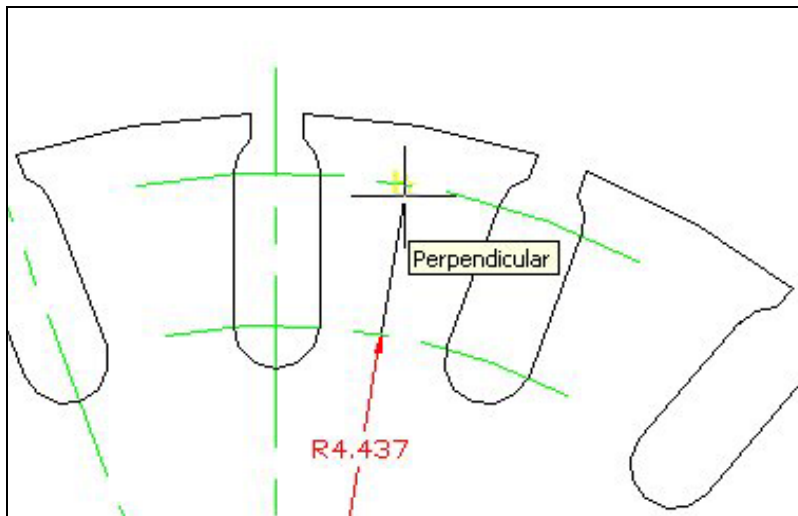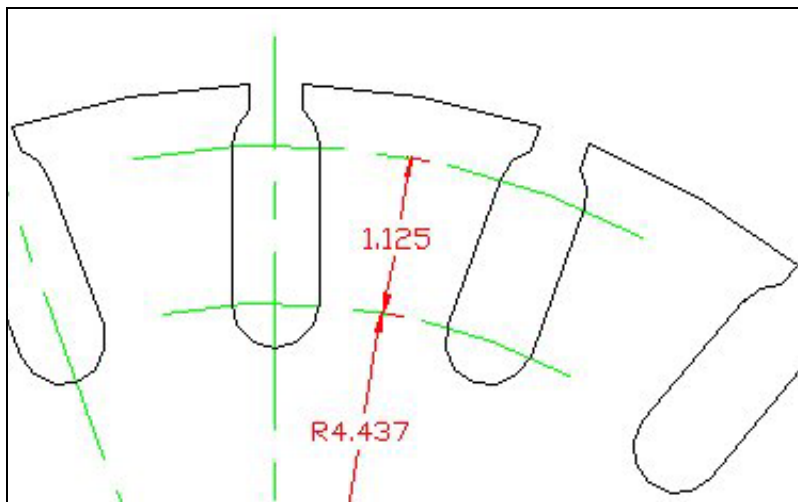
Pick the Aligned Dimension tool on the Dimension toolbar, and select the endpoint of the R4.437 and for your second point place the aperture over the second arc and wait to see the Perpendicular Osnap symbol as shown in Figure 6.15. Pick the second arc and a 1.125 dimension will appear that is aligned 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 mate 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

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There are two angular dimensions to explain in the Circular drawing and remember how easy the command is to use. Select the Angular Dimension tool on the Dimension toolbar and you need to select two converging lines (non-parallel) to place the one place decimal on the drawing.
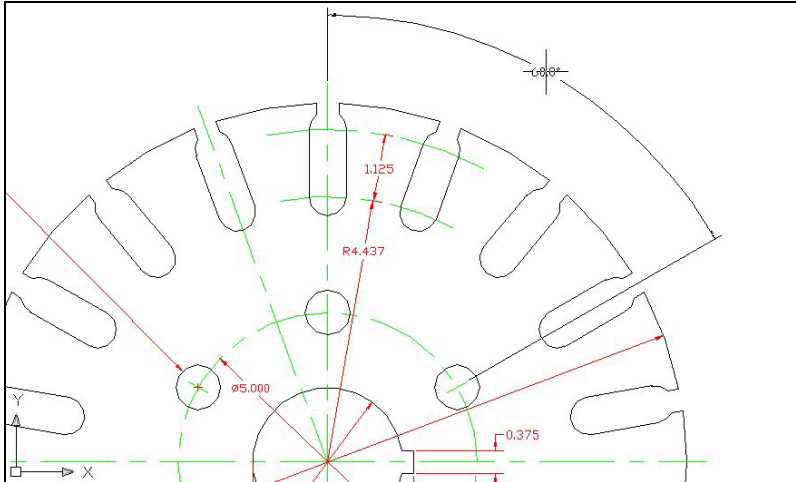

Figure 6.17 – Using the Angular Dimension Tool

For the first pick, select the green center mark on the upper right 0.75 hole and 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).
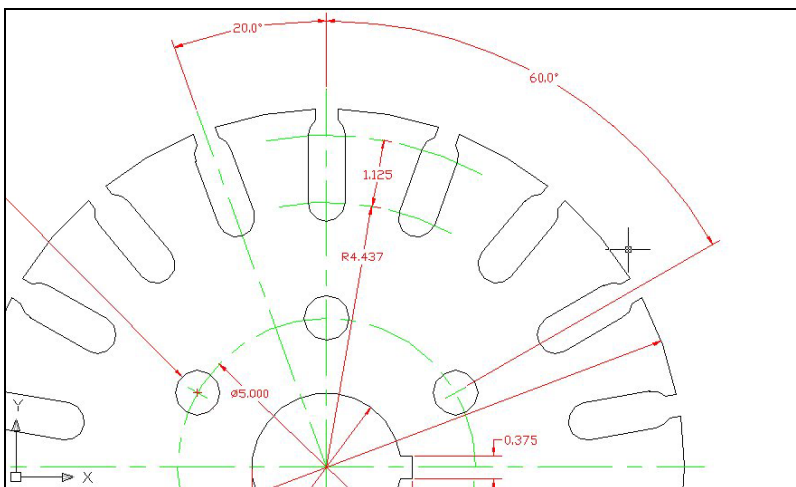

Figure 6.18 – Using the Angular Dimension Tool

The last Angular dimension on the drawing is the 20.0 degree measurement telling the angle between each of the 18 slots. After opting again for 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.
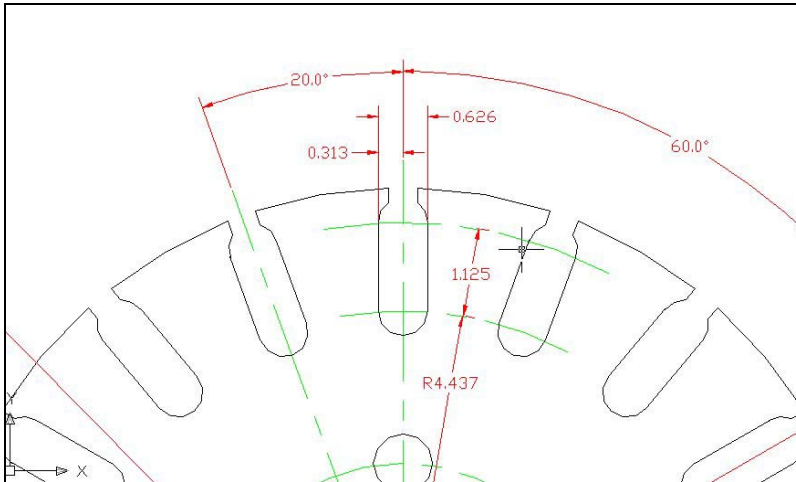
For the third selection, choose the space between the two centerlines and move the dimension arrows in alignment with the 60.0 degree dimension.

When placing dimension on the outside perimeter of a part or a building detail, remember, you may want to place more dimensions below that measurement. So 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.

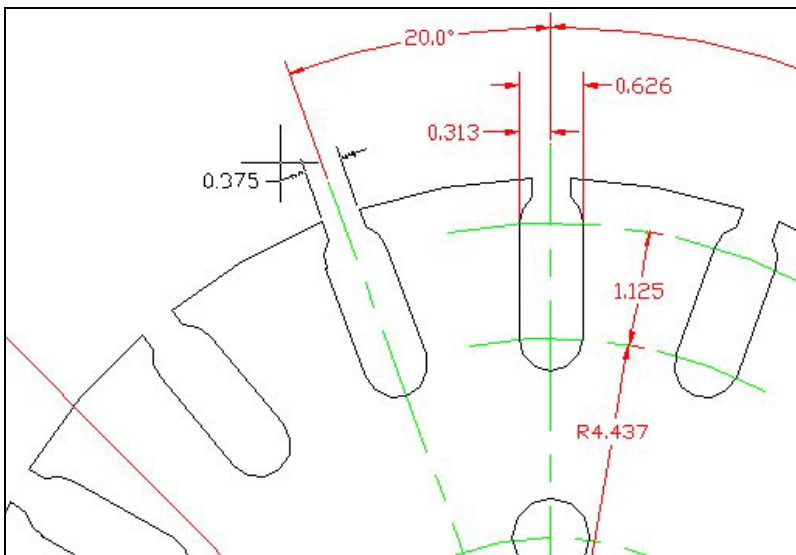# More Linear Dimensions

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The last series of dimensions are located at the top quadrant of the Circular problem. Go ahead and select the Linear Dimension tool on the Dimension toolbar to indicate 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.

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 our case in Figure 6.19, place a linear dimension for 0.626 as shown.

Figure 6.19 – Using the Linear Dimension Tool
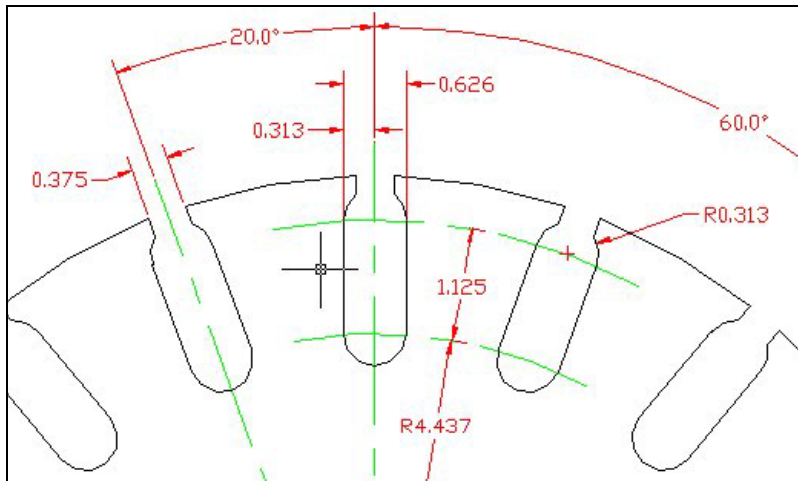
Another Aligned Dimension

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Choose the Aligned Dimension tool on the Dimension toolbar 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.

Figure 6.20 – Using the Aligned Dimension Tool

Another Radius Dimension

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Figure 6.21 – Using the Radius Dimension Tool

The last dimension on the Circular problem is the 0.313 radius on the slot. Once selecting the Radius Dimension tool located on the Dimension toolbar, 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.

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 * - 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

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When you need to make special changes form your Standard Dimension Style, you can select the Properties tool on the Standard toolbar. Many times you will leave the center mark option on the Lines and Arrows tab (Figure 6.2)

After selecting the Properties tool, pick the R0.313 dimension as shown in Figure 6.22 and the Properties window will appear. Make sure the Alphabetic tab is selected, and expand the Lines and Arrow category by picking the plus sign (+) and click in the Center Mark. In the list box, change form 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) which is in the upper right corner of the Properties window to close the window.
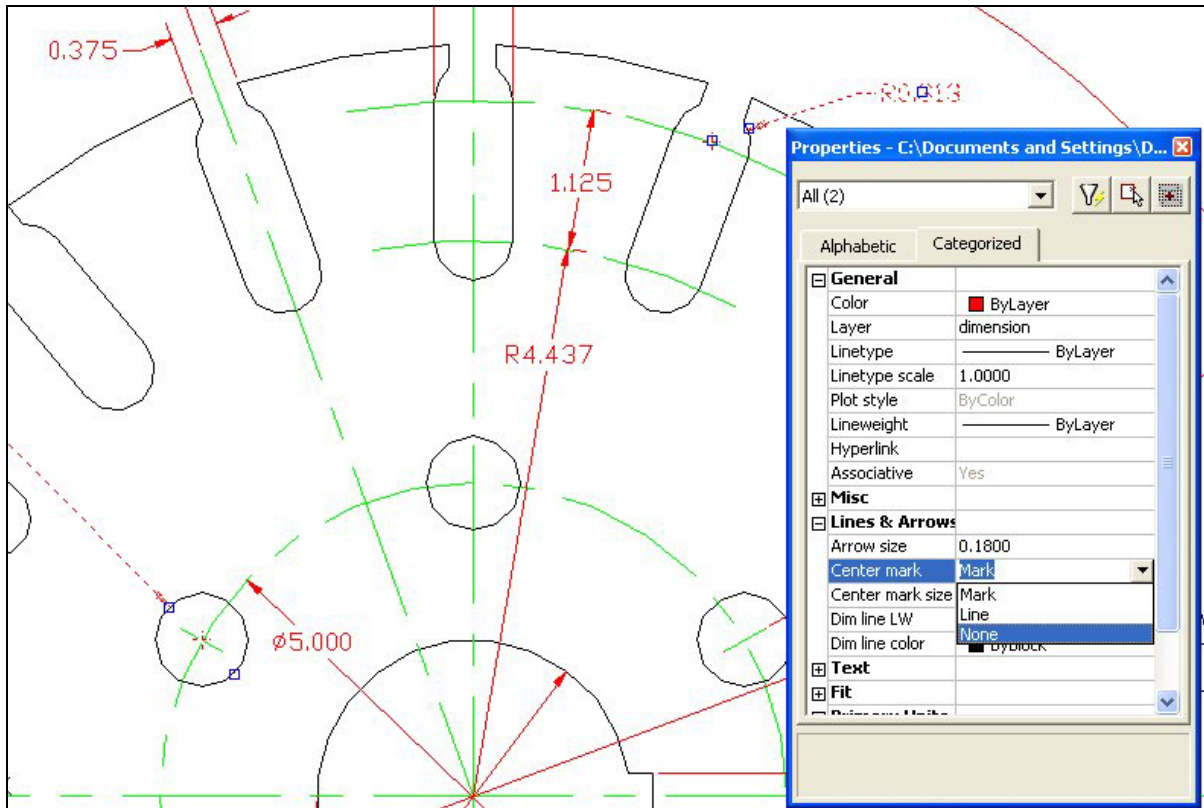
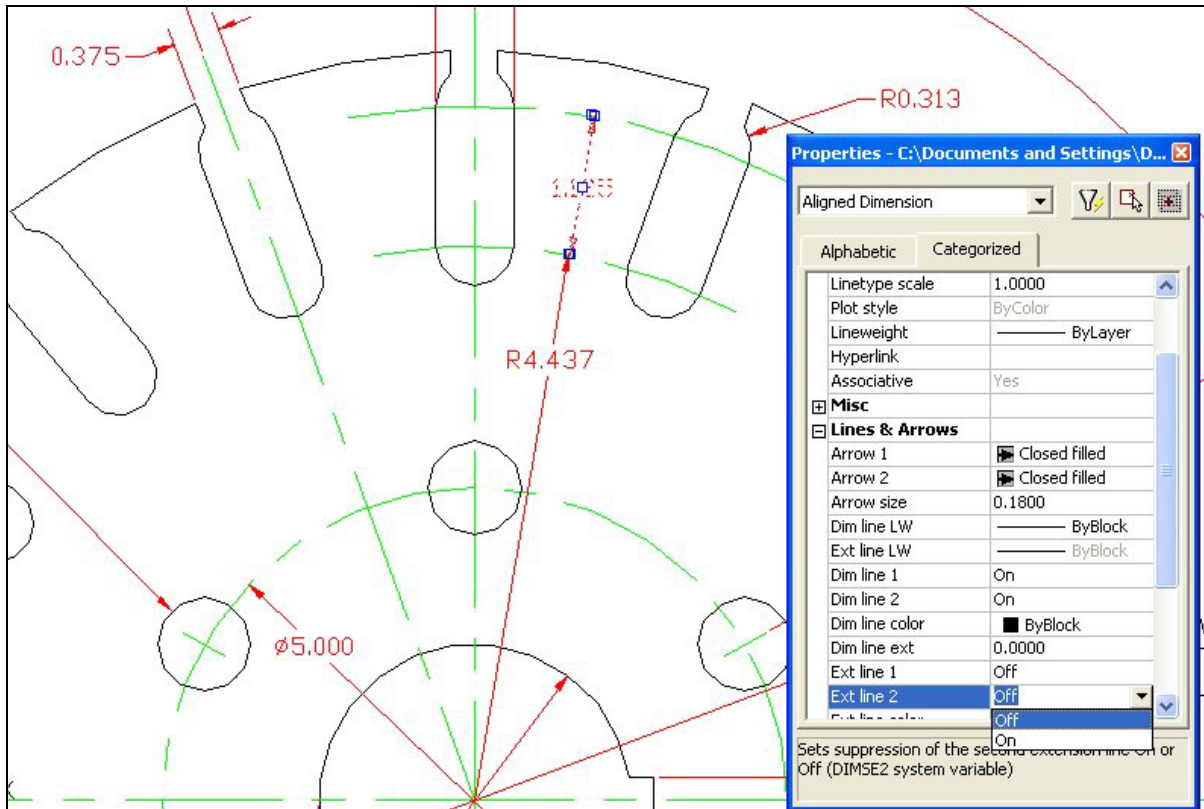**Figure 6.22 – Modifying Dimensions Using the Properties Tool**

Figure 6.23 – Modifying Dimensions Using the Properties Tool

6-13

Using the Edit Text Tool to Add a Note to a Dimension
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Choose the 0.75 dimension text and select the Edit Text tool on the Text Editing
toolbar the Multiline Text Editor window will appear. With your mouse, place the
text insertion bar behind the associative dimension symbol <>, and Enter to go to the
second line. Type the text 6 HOLES and press the OK button. The small note will be
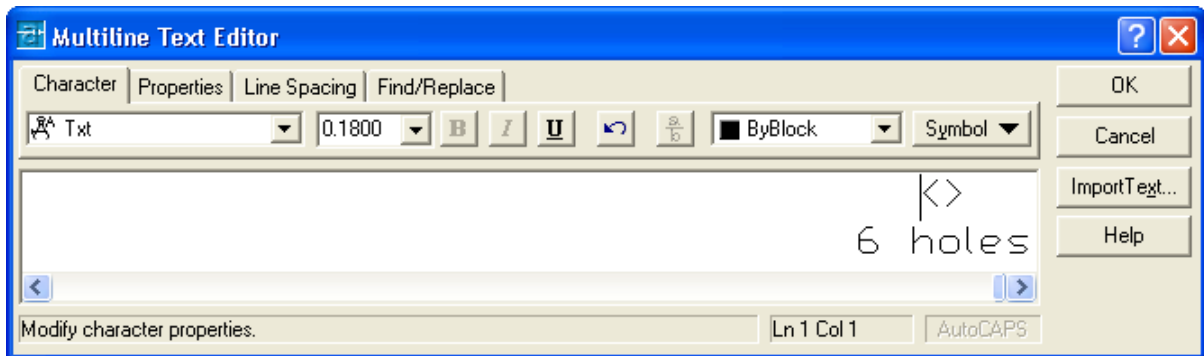seen below the 0.75 text. (See Figure 6.24)

Figure 6.24 – Modifying Text using the Multiline Text Editor

Dimensioning the second drawing, the Circular problem is finished and you can 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.
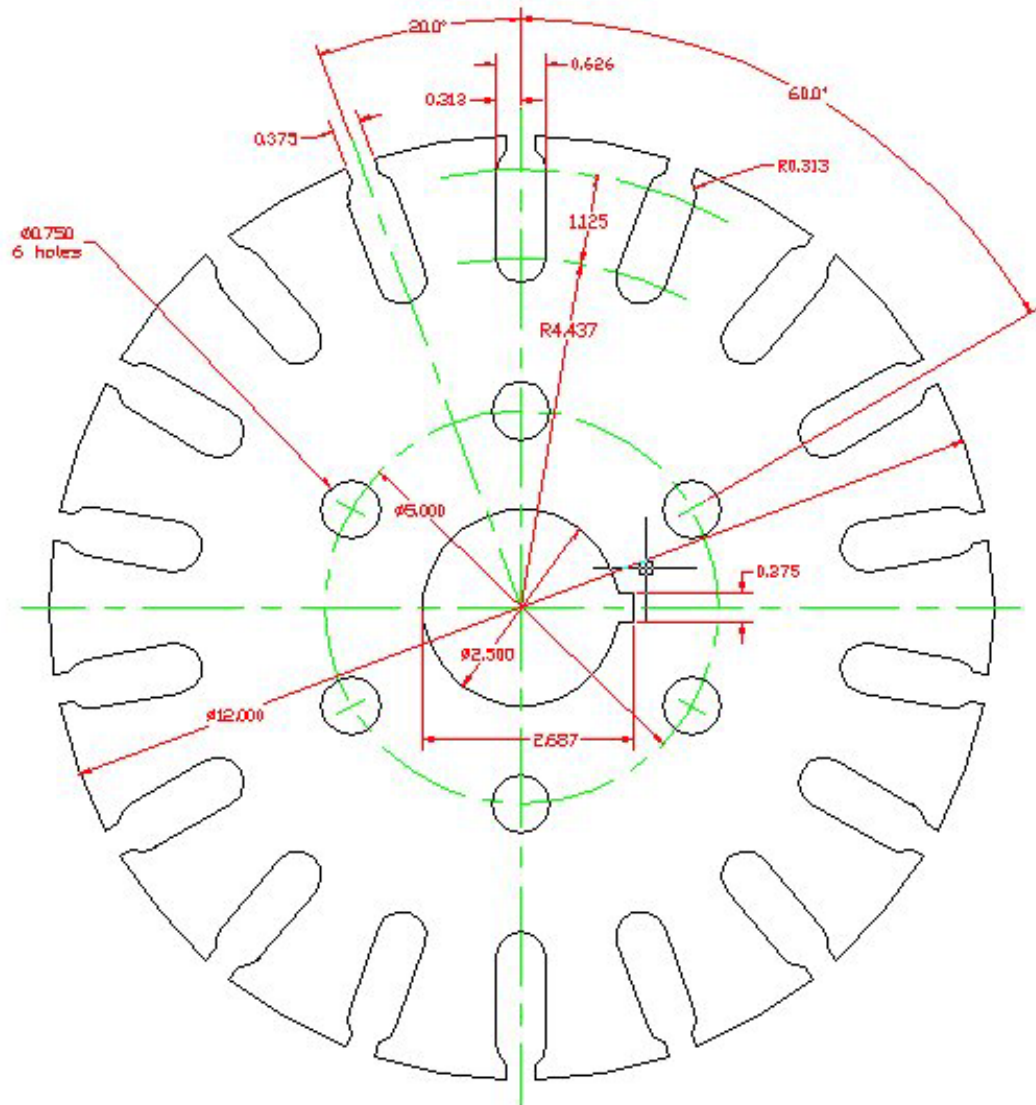
Figure 6.25 – Finished Dimensioned Circular Problem

* World Class CAD Challenge * - 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.