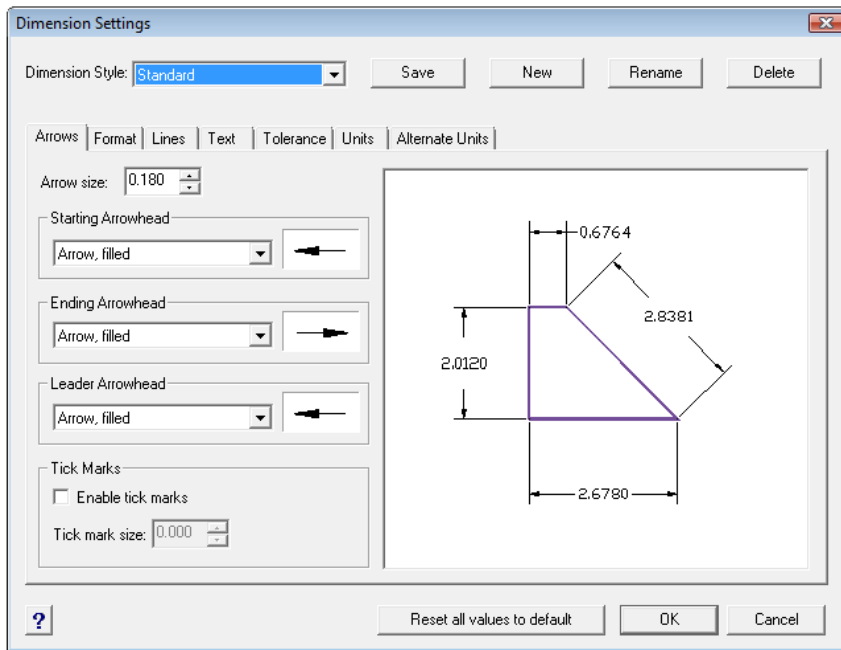


Dimensioning the Circular Problem

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

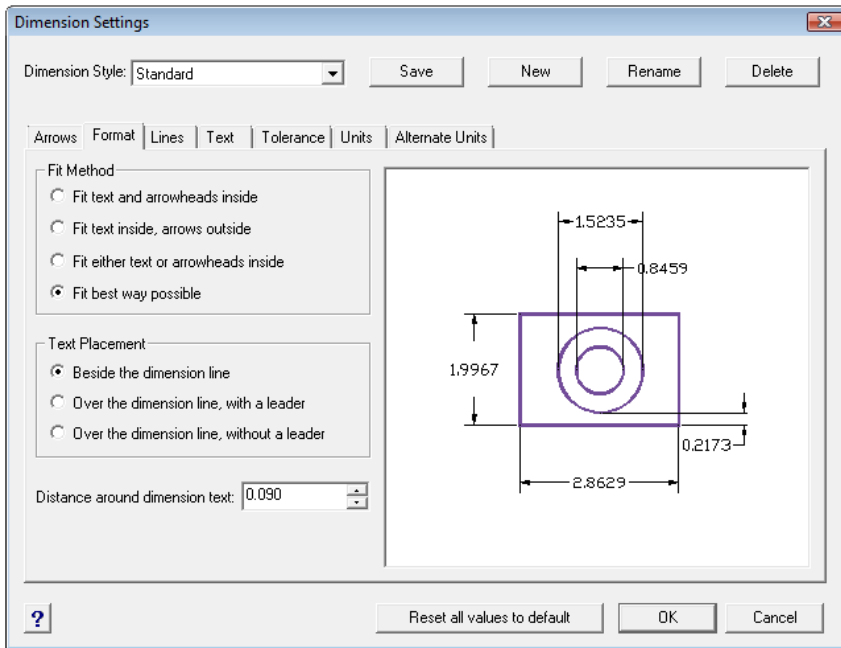
- 1. A Basic Discussion of the Dimensioning Style Window**
- 2. Trimming Centerlines with the Trim Tool**
- 3. Using the Diameter Dimension Tool**
- 4. Using the Linear Dimension Tool**
- 5. Using the Radius Dimension Tool**
- 6. Using the Aligned Dimension Tool**
- 7. Using the Angular Dimension Tool**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

A Basic Discussion of the Dimension Settings Window



Relating an orthographic view to a machinist or construction worker with the progeCAD system is easy and accurate. You can always control the 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 Settings Window – Arrows



To modify the Standard dimension style which you will be using to dimension the Circular problem, select the Dimension Style tool from Dimension on the menu bar. The Dimension Settings window will open as shown in Figure 6.1. Select Standard from the Dimension Style drop box as the starting block for your custom dimension style. The Arrows tab shown in Figure 6.1 allows you to alter the Arrow Size, shape, and Tick Marks.

Figure 6.2 – Dimension Settings Window – Format

For example, in an architectural drawing, you will change the Arrowheads from being Arrow, Filled to Architectural Tick, but most of the time this tab will be left alone. Under the Format tab, make sure that the Fit Method is set to “Fit best way possible” (Figure 6.2).

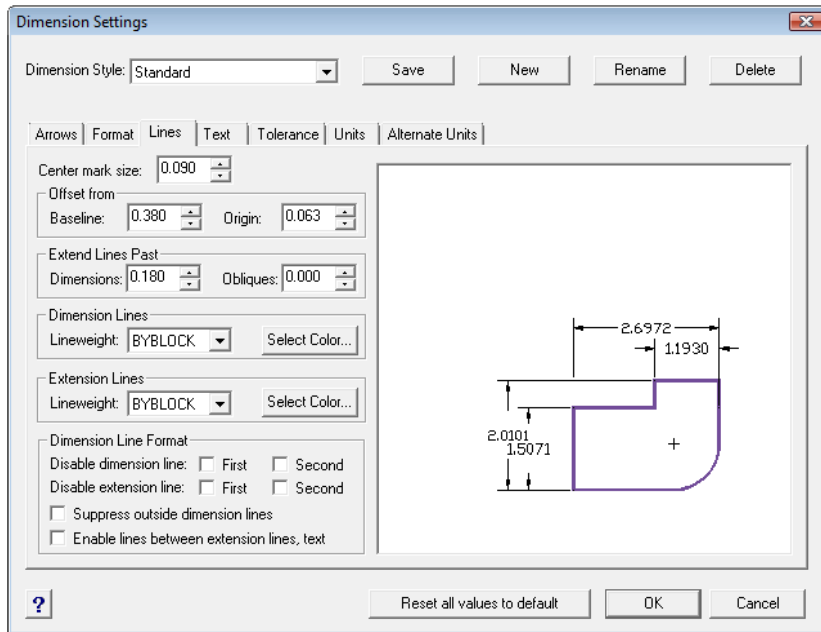


Figure 6.3 – Dimension Settings Window – Lines

Proceeding to the Lines tab, you can set up the Center Mark size, Offset, Line Extension, Dimension and Extension Lineweights, and Dimension Line Format (Figure 6.3). We don't need to make any changes to these options for the Circular problem.

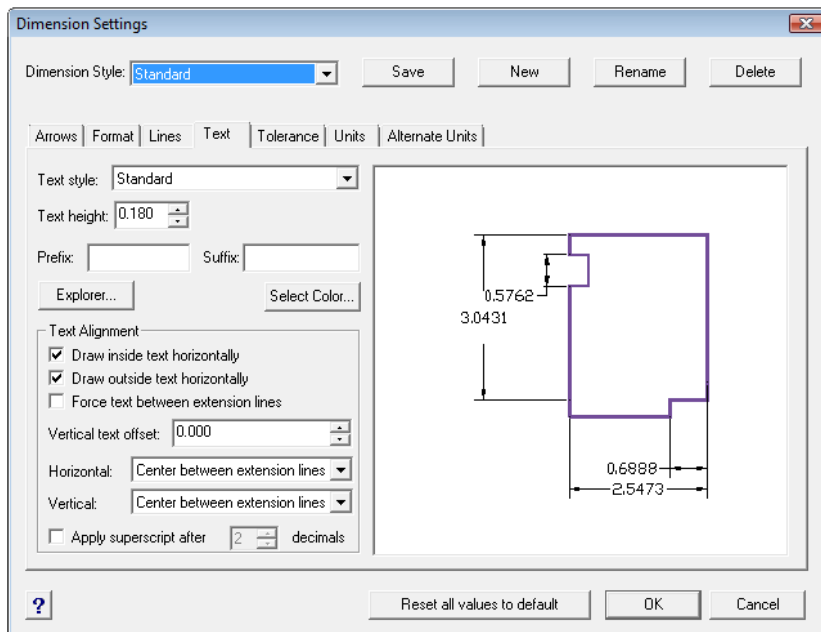
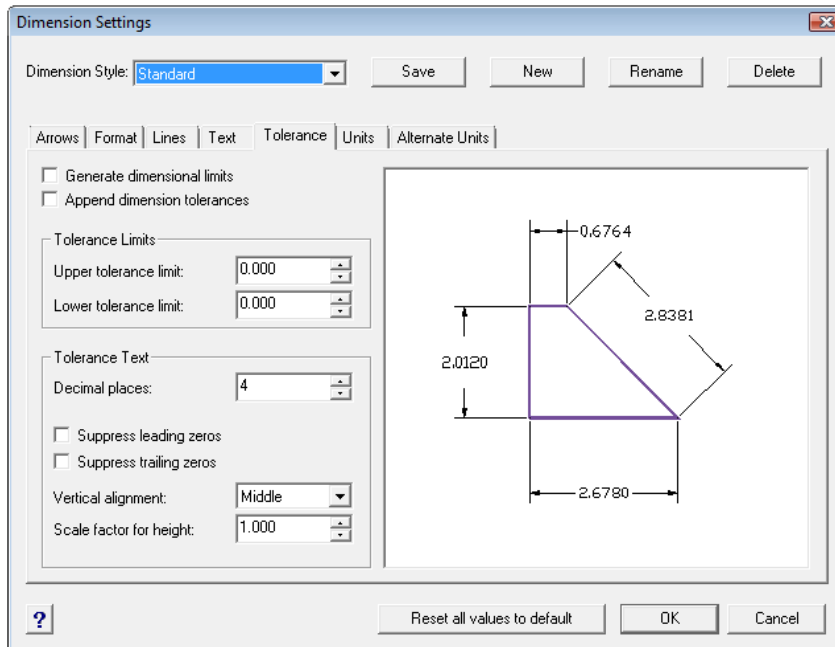


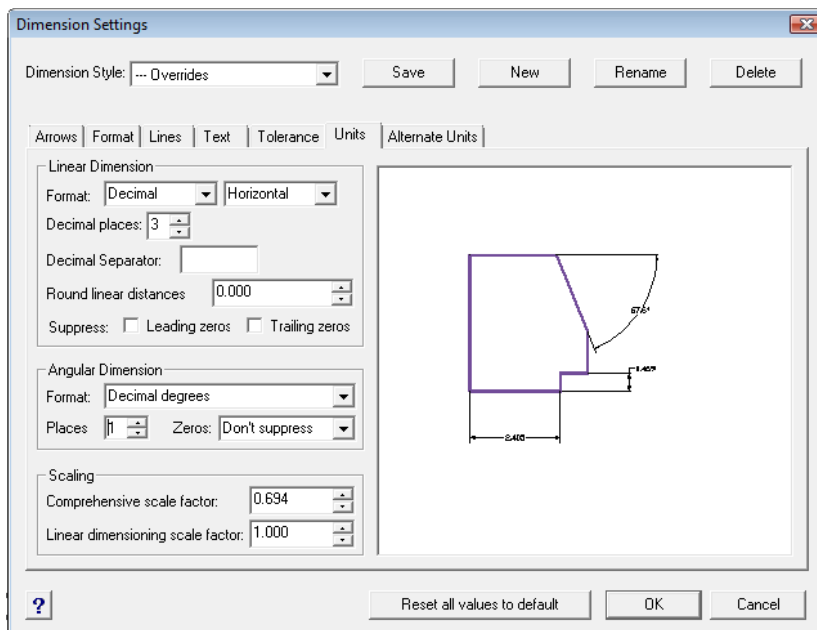
Figure 6.4 – Dimension Settings Window – Text

The Text tab allows you to customize the text that will appear as labels on your dimensions (Figure 6.4). 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.



The Tolerance tab (Figure 6.5) 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 detail dimensions.

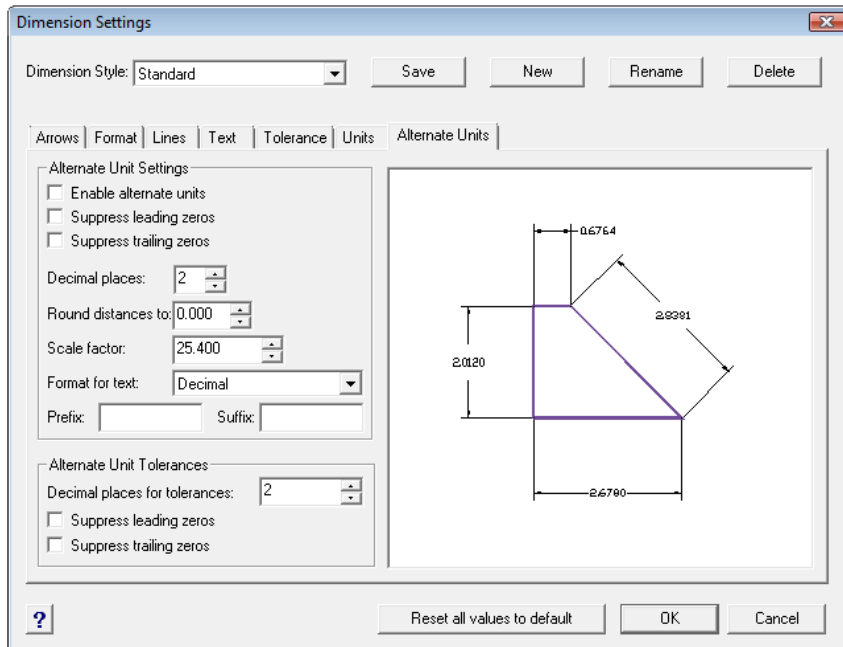
Figure 6.5 – Dimension Settings Window – Tolerance



In the Units tab, the Linear Dimension Format will remain as Decimal. Set Decimal Places to 3 as shown in Figure 6.6. The other Linear Dimension settings are used periodically, such as Round Linear Distances, but the 1.0000 Linear Dimension Scale Factor under Scaling should stay always at 1. Confusing this setting with other scale factors like Comprehensive Scale Factor will change the value of all the dimensions in the progeCAD drawing.

Figure 6.6 – Dimension Settings Window – Units

Also, you need to change the Comprehensive Scale Factor to 0.694 as shown in Figure 6.6 to reduce the size of the text and arrows to 0.125. Also change the decimal Places in the Angular Dimension section to 1.

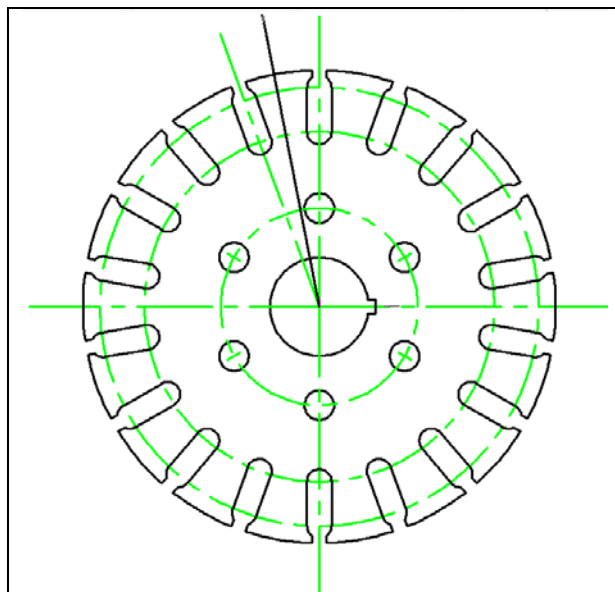


The Alternate Units tab (Figure 6.7) would be used when you want to list the metric value of the English unit below or alongside the dimension text. For example, if you checked “Enable alternate units,” a dimension showing one inch will have the Scale Factor (25.40) in parenthesis below the 1.000 text. You do not need to check the “Display alternate units” checkbox on the Circular problem.

Figure 6.7 – Dimension Settings Window – Alternate Units

To save the change that you made in the Dimension Settings window, select the OK button.

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 ESnap, select any circle. 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

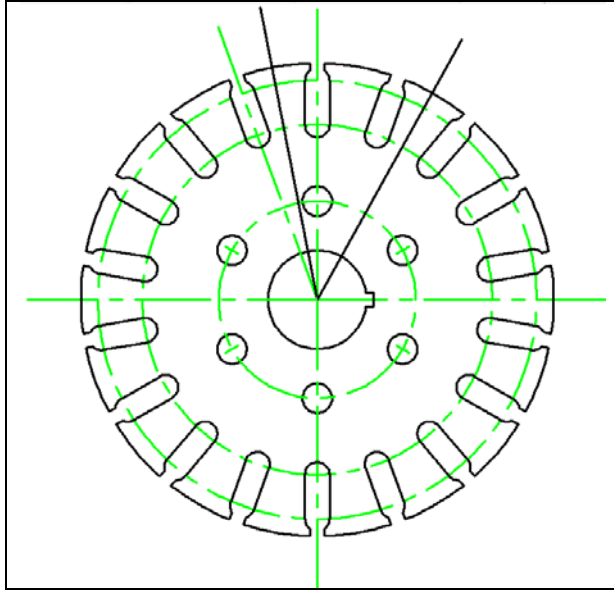


Figure 6.9 – Adding a Second Cutting Line

Now select the Trim tool on the Modify toolbar. At the cutting entity 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.

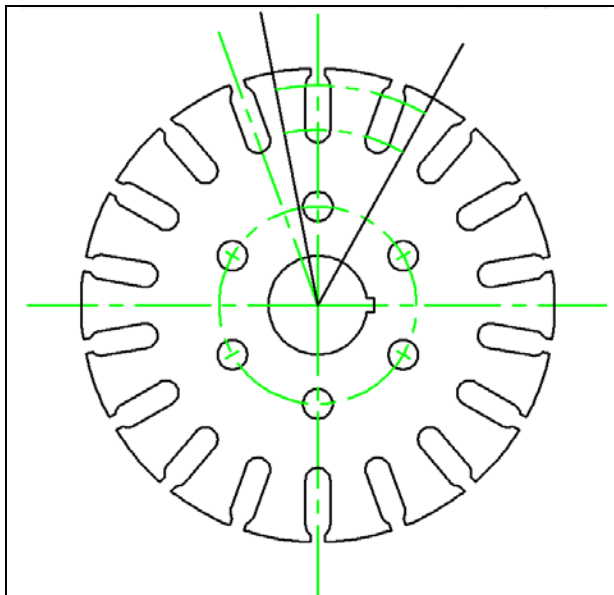


Figure 6.10 – Trimming the Centerlines

Next select the Erase tool from the Modify 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, as World Class CAD the saying goes, “You have to go slower to go faster.” Accuracy counts.

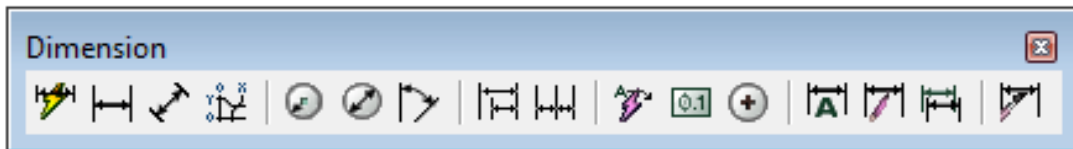


Figure 6.11 – The Dimension Toolbar

Using the Diameter Dimension Tool

Dimensioning is the easiest task to be performed in CAD and Diameter is one of the easiest dimensions to place. Select the 12.0 inch diameter circle and the dimension lines, arrows and text instantly appear for you to place on the screen. Place the 12.000 toward the upper right 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.

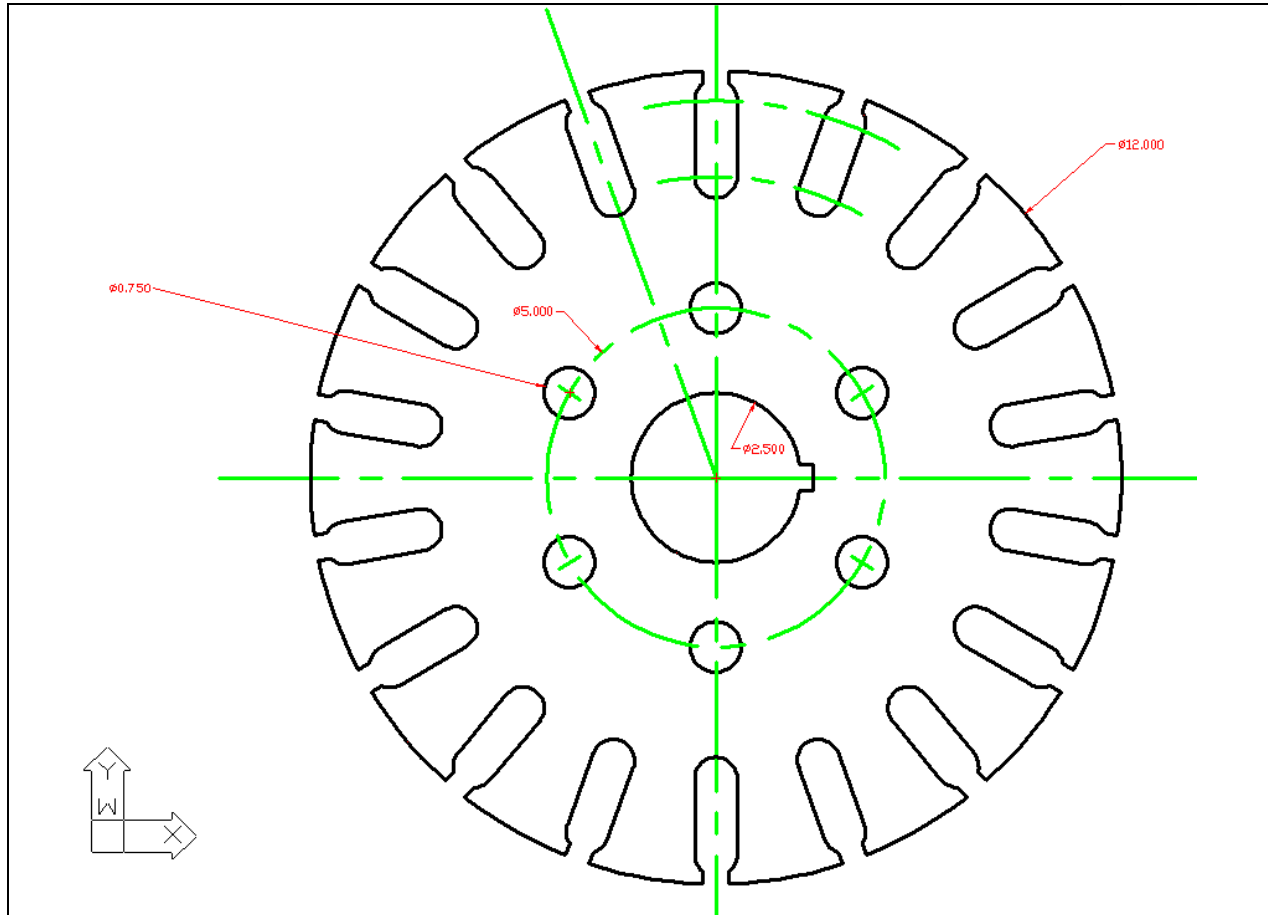
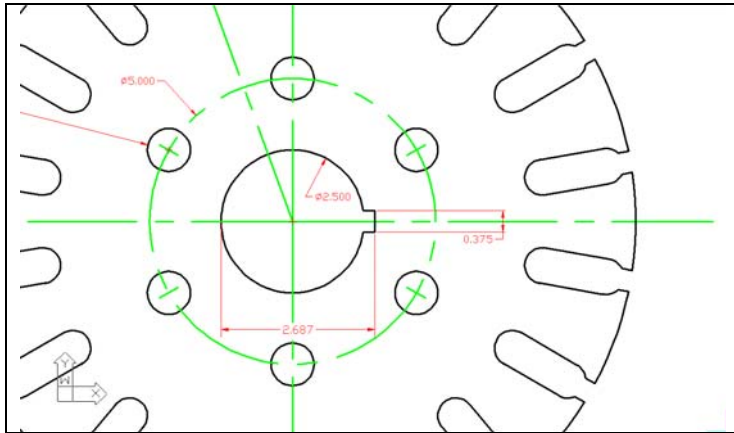


Figure 6.12 – Dimensioning Using the Diameter Dimension Tool

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 forget to place a dimension on the drawing. 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

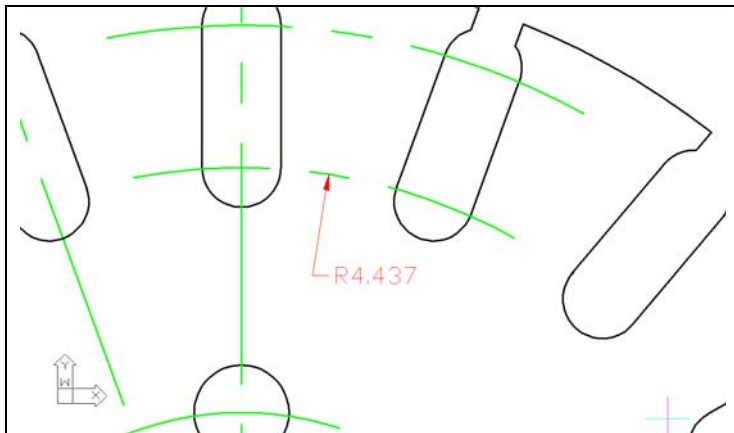
Most circular parts are designed with a keyway so that a key can be installed between the shaft and the hub to prevent slippage. A keyway is referenced from the opposite quadrant of the circle, so to place the horizontal dimension select the Linear Dimension tool on the Dimension toolbar. Pick the west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



The 2.687 dimension will appear for you to place, so 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. 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.

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 toolbar. 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

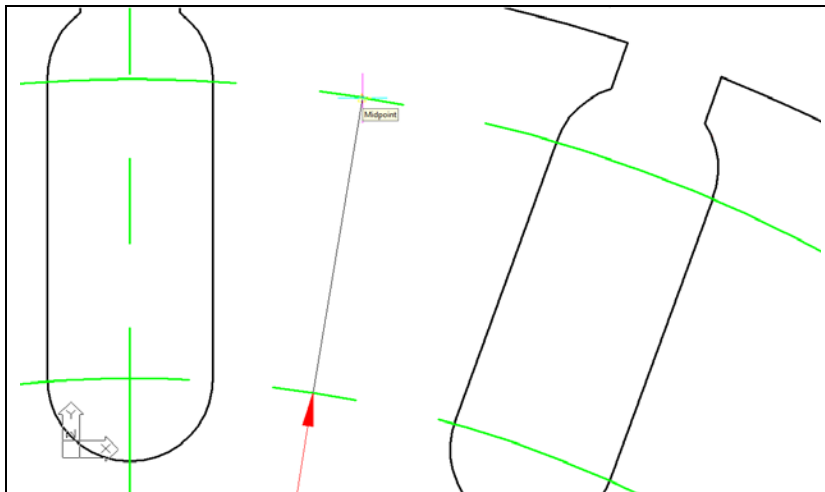
Select anywhere on the green inner arc. You can move the radius dimension line back and forth moving the dimension arrow to the desired location between two slots. By placing the radius dimension as shown above, the Aligned Dimension can be added to measure the distance to the second arc.

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 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 Esnap settings.

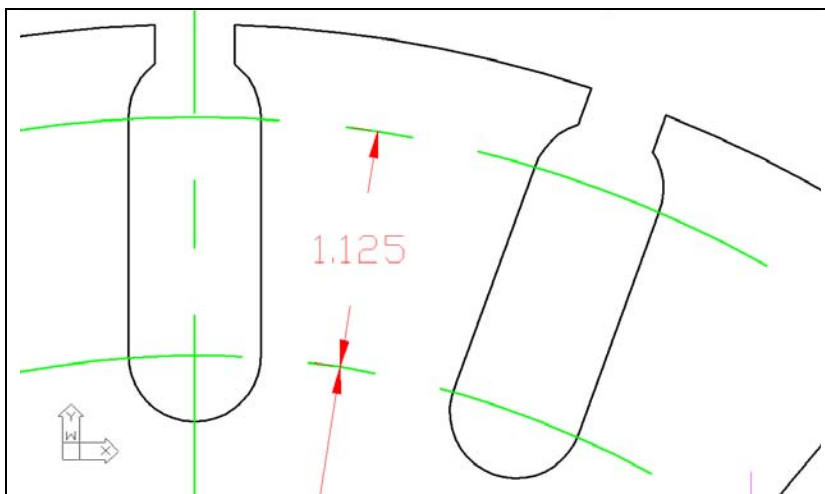


Select the ESnap Settings tool on the Object Snap toolbar to open the Drawing Settings window. Make sure that the Perpendicular checkbox and Endpoint checkbox are checked and press OK to close the window. Feel free to change Esnap settings at key times in the drawing process, such as between major command 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. For your second point place the aperture over the second arc and wait to see the Perpendicular Esnap 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 (Figure 6.16).

Figure 6.15 – Using the Aligned Dimension Tool

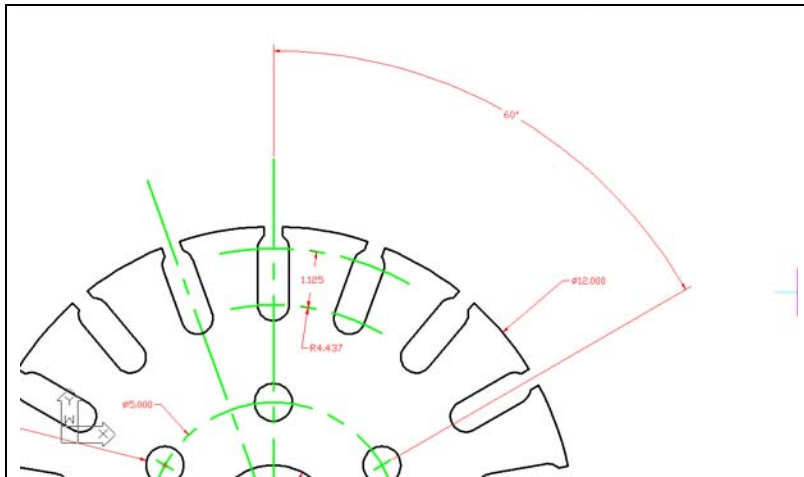


Many engineers and designers will want to place a second radius dimension on the second arc, but the reference and the tolerance for the feature will be from the wrong location. Slots like these will mate with other parts. The dimension controlling these surfaces is based off the 1.125 dimension, so use an Aligned Dimension to explain the part.

Figure 6.16 –The Aligned Dimension

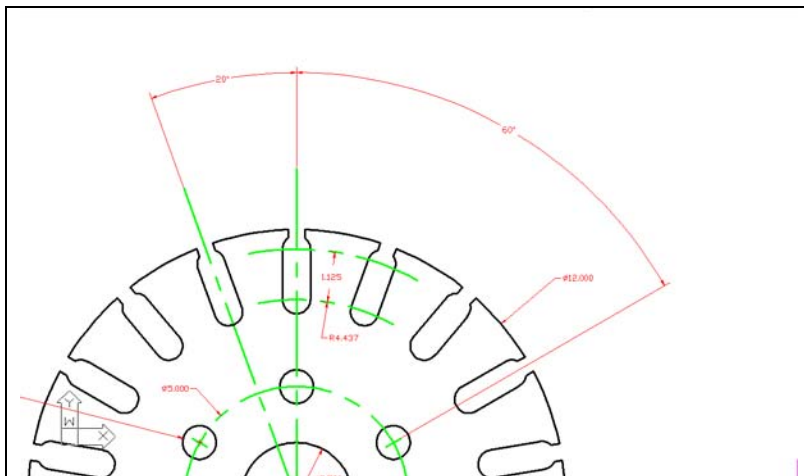
Using the Angular Dimension Tool

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 toolbar and select two converging lines (non-parallel) to place the one place decimal on the drawing.



For the first angle 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-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 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.

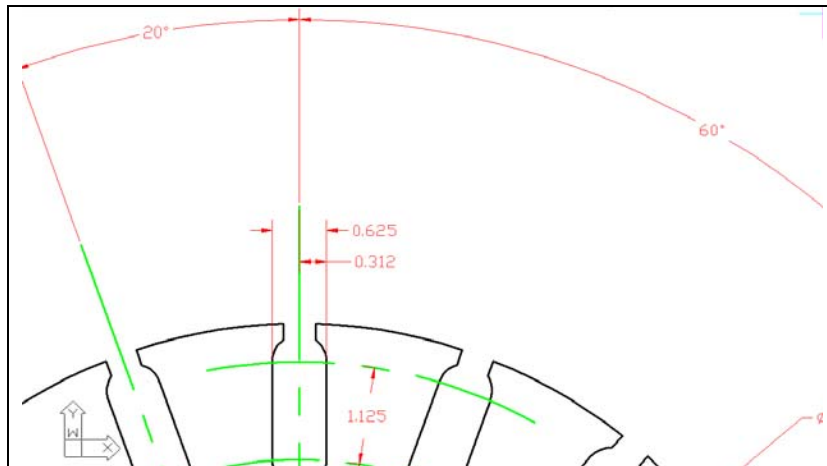
Figure 6.18 – Using the Angular Dimension Tool

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

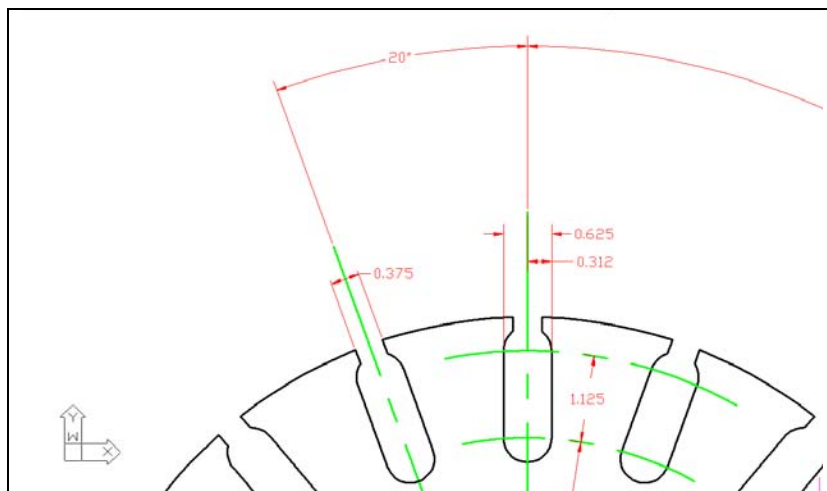
The last series of dimensions are located at the top quadrant of the Circular problem. Select the Linear Dimension tool on the Dimension toolbar to place two of these remaining measurements. For the 0.312 dimension, pick the vertical centerline and the outside edge of the slot. Repeat the command for 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 the 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



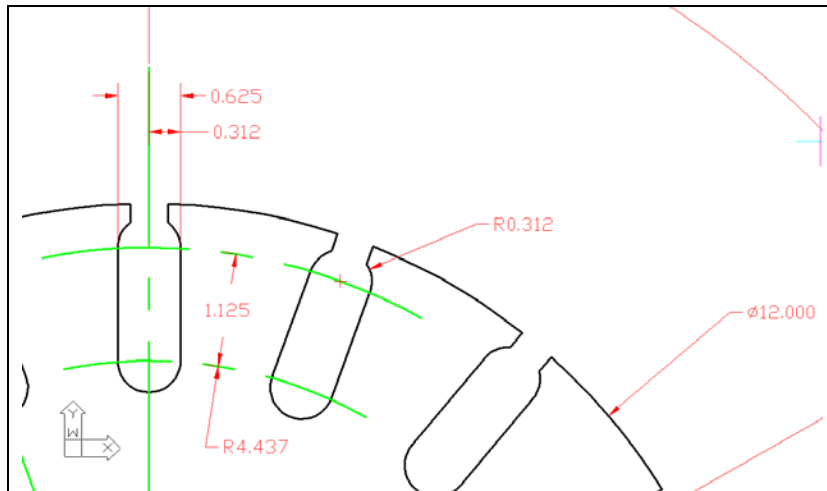
Choose the Aligned Dimension tool on the Dimension toolbar and select the endpoint of the opening of the slot as shown in Figure 6.20. By picking the line on the right followed by the line on the left, the dimension text will appear on the side picked second.



Figure 6.20 – Using the Aligned Dimension Tool

If you are ever unable to place a dimension with the label aligned properly, you can always paste the dimension and move the label later. Selecting the dimensions should bring up handles you can use to move the label to a better place on the drawing.

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 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.

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 100-14 * - Create a New file, create the three new layers, set the Entity 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 Explode Tool to Modify a Dimension

When you need to make special changes from your Standard Dimension Style, you can select the Explode tool on the Standard toolbar. This tool will disable your ability to manipulate the dimension label as a complete dimension, but you will be able to get rid of unnecessary details.



First select the entity you want to Explode: in this case, pick the R0.313 dimension. Now select the Explode tool from the Modify toolbar. Now you will be able to select the two lines that make up the center mark of the dimension and delete them, so your drawing looks like Figure 6.22. For the second modification, Explode the 1.125 aligned-dimension and remove the extension lines at each end of the dimension as shown in Figure 6.23

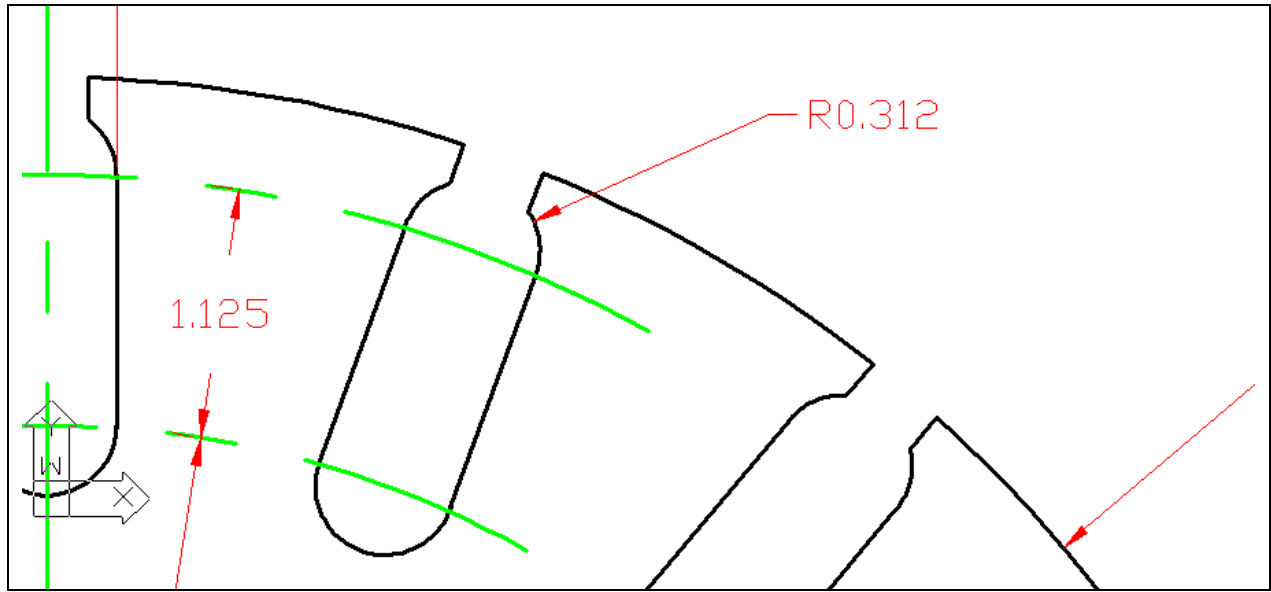


Figure 6.22 – Removing the center mark from the exploded dimension

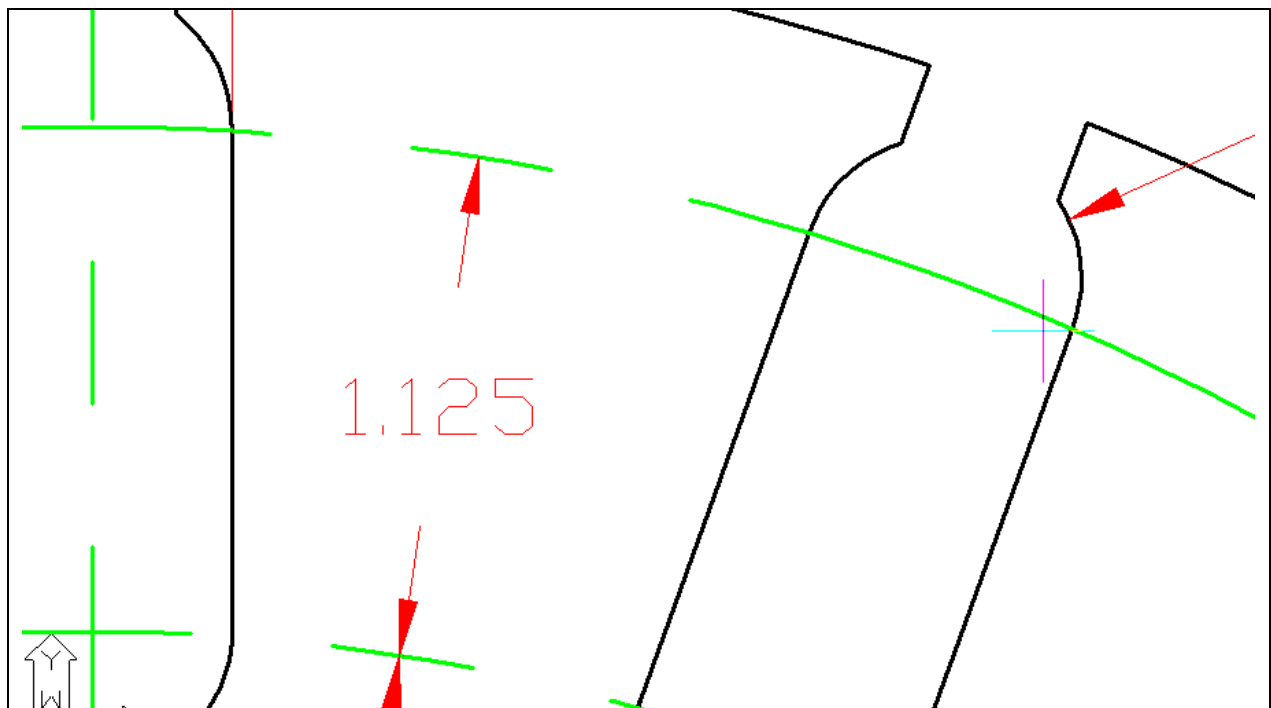


Figure 6.23 – Removing the extension lines from the exploded dimension

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

Choose the 0.75 dimension text and select the “Text...” tool on the Text toolbar. Select the .750 radius dimension and the Entity Properties window will appear (Figure 6.24).

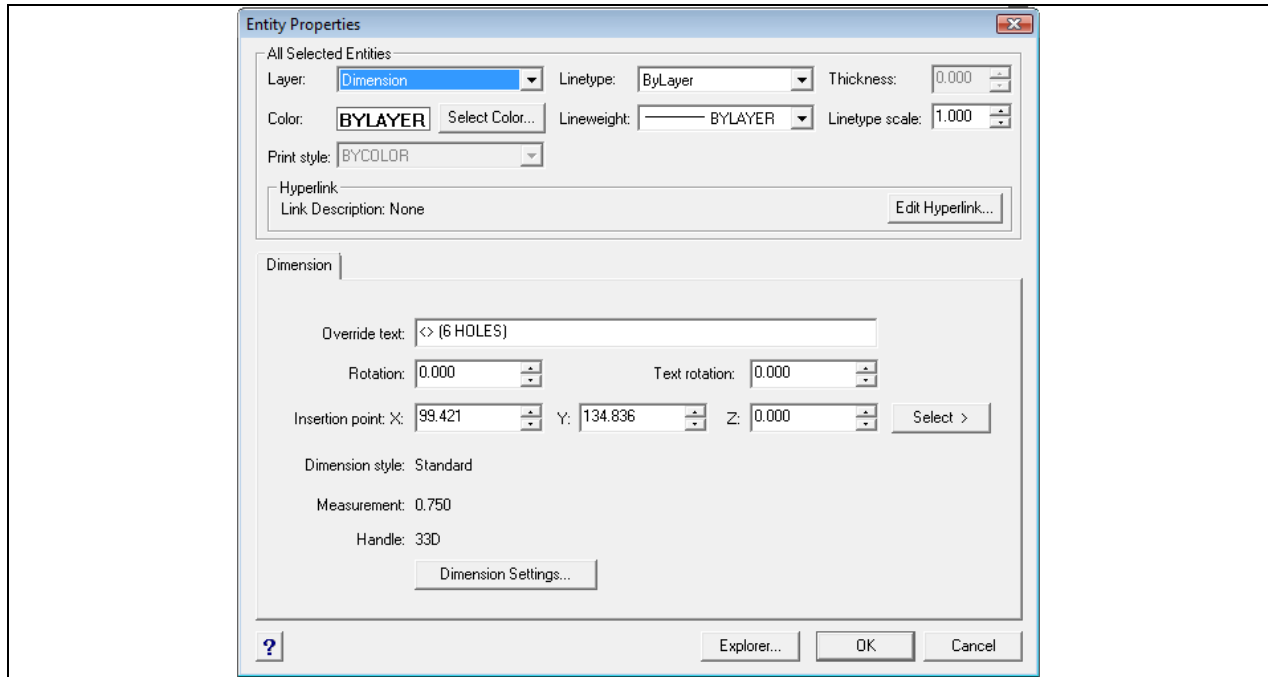


Figure 6.24 – Modifying Text using the Edit Text tool

Beside the “<>” in the Override text field (which represents the dimension label text) type (6 HOLES) and hit ok. Now after the 0.750 dimension the note tells the machinist that each of the six holes is .750 in diameter.

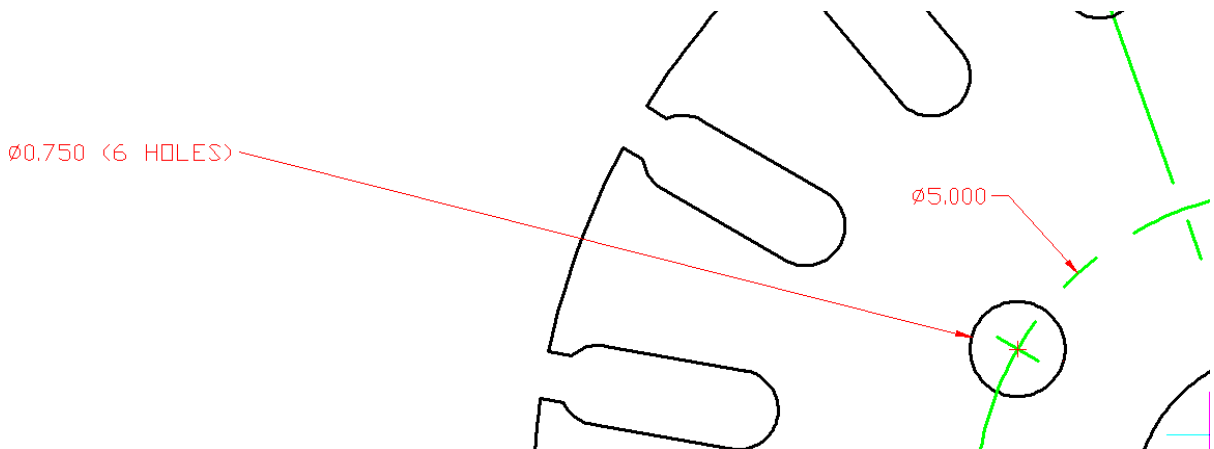


Figure 6.25 – The Edited .750 Diameter Dimension

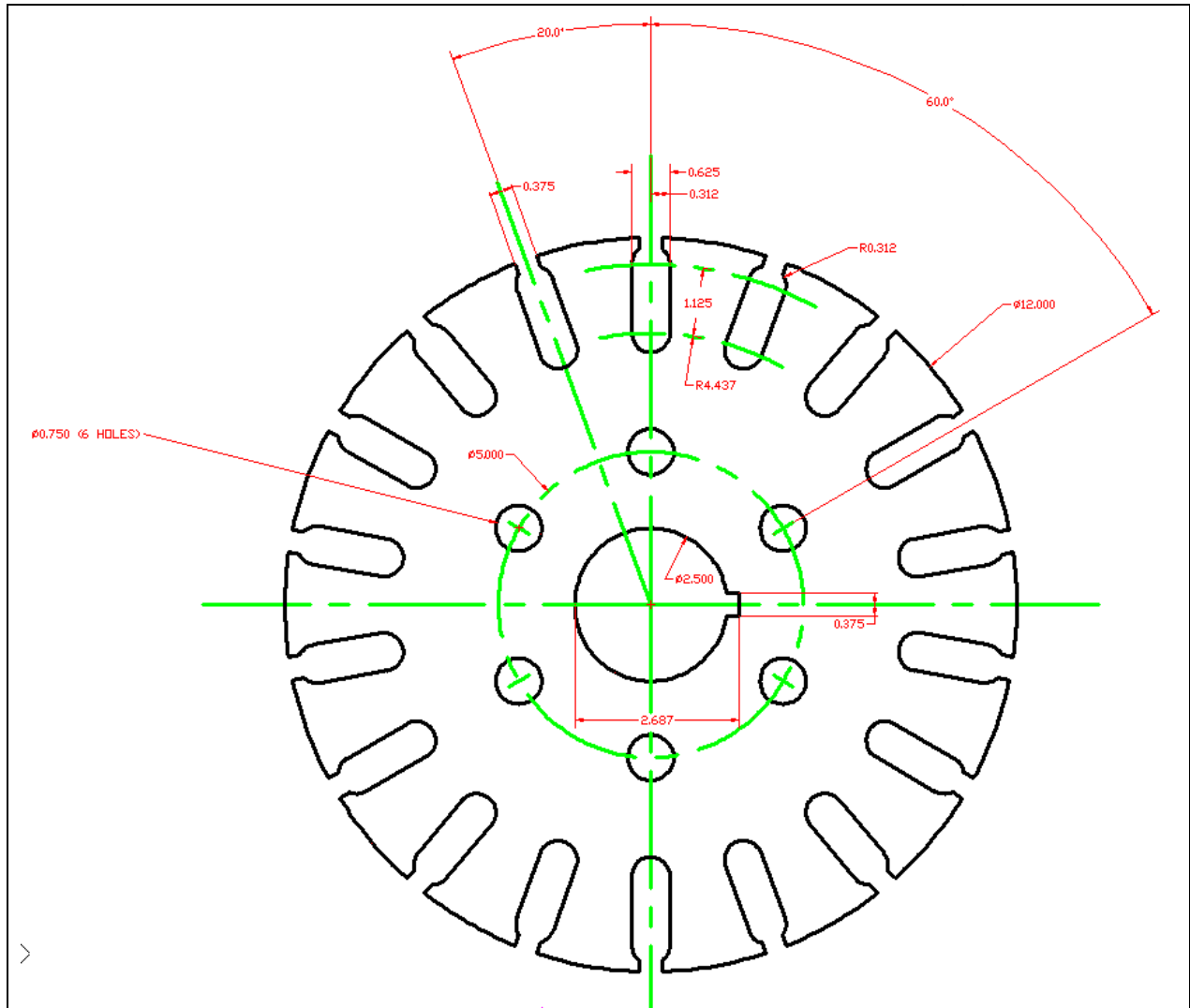


Figure 6.26 – Finished Dimensioned Circular Problem

*** World Class CAD Challenge 100-15 * - Create a New file, create the three new layers, set the Entity 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.**