Chapter 6

Dimensioning the Circular Problem

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

- **Trimming Centerlines with the Split Tool**
- **Using the Diameter Dimension Tool**
- **Using the Distance Between Dimension Tool**
- **Using the Radius Dimension Tool**
- **Using the By 2 Points Option of the Distance Between Dimension Tool**
- **Using the Angle Between Dimension Tool**
- **More Linear Dimensions**
- **Another by 2 Points Dimension**
- **Another Radius Dimension**
- **Using the Edit Definition-Prefix Tool to Add Text Below a Dimension Using a Subfix**

Trimming Centerlines with the Split Tool

Before dimensioning the circular problem, you will cut away some of the excess centerlines that may cause confusion for the print reader. This is accomplished by drawing two regular lines from the center of the circular problem past the outer periphery of the part. Select the **Line** tool from the **Draw** panel and, using the Endpoint IntelliSketch, select the line to start at the center of the part. Also, you want to make sure that the line type is set to visible before you finish drawing the line. Draw the line to the left of the topmost slot and make sure it stretches past the outer rim of the circle as in Figure 6.1. Repeat the routine, drawing the second line two slots over as shown in Figure 6.2.
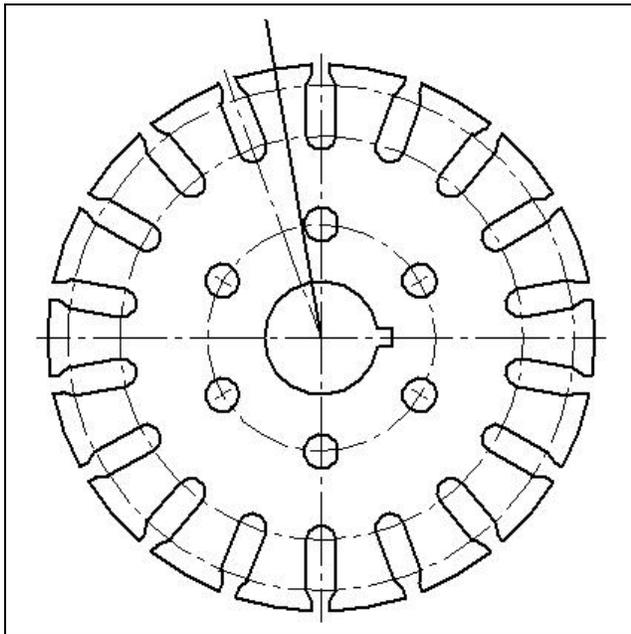

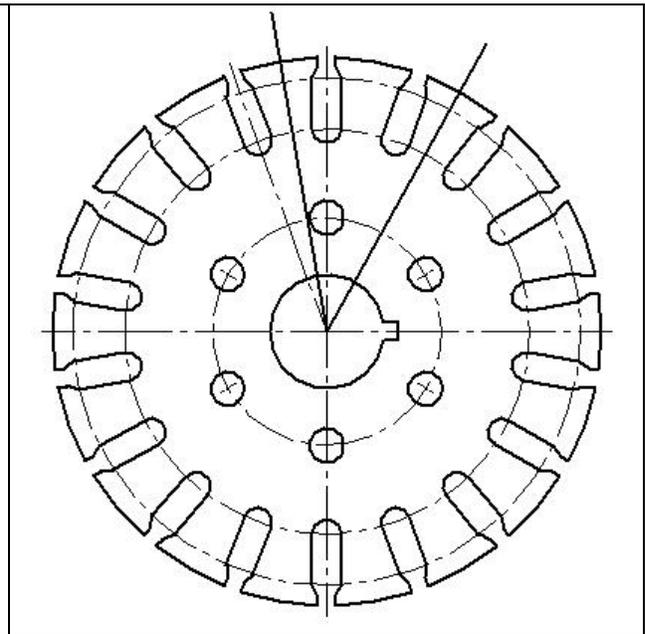**Figure 6.1 – Adding First Cutting Line**   **Figure 6.2 – Adding Second Cutting Line**

Next you need to use the **Split** tool from the **Draw** panel to separate the portions of two circles that are inside the lines you just drew. Select the **Split** tool and pick the outermost dashed circle. Next, pick the intersection point of the circle with the left line and a large arrow will extend from that point (Figure 6.3). If the Intersection IntelliSketch icon is not activated, go to the **IntelliSketch** panel and activate it. The icon is an X with a dot in the middle. After selecting the intersection of the circle with the left line, select the intersection with the right line. Do the same for the next dashed circle, splitting it at the intersection points of the line that you just drew (Figure 6.4). Push the **esc** key to exit the command. Now when you put your cursor over the circles, the portion inside and outside the cutting lines should be separate.
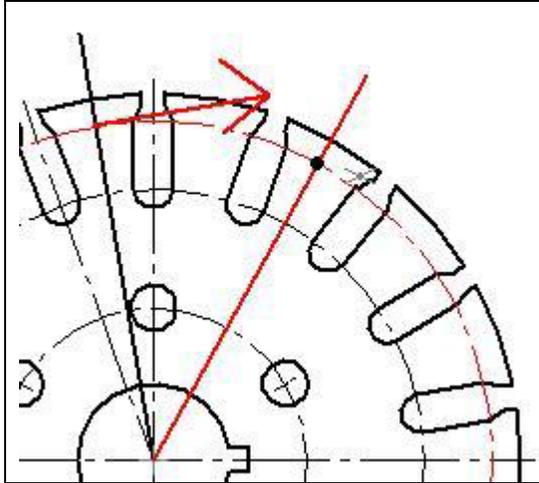
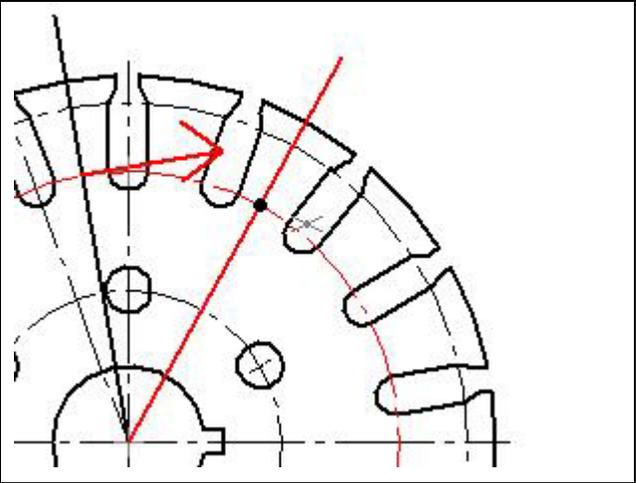**Figure 6.3 – Splitting First Circle**   **Figure 6.4 – Splitting Second Circle**

Now, pick the **Select** tool and select the two cutting lines and the portions of the two circles that are outside the cutting lines (Figure 6.5). Just as a reminder, you can select multiple objects by selecting one of the entities and then holding down the **Shift** key on your keyboard while you select the rest. With these four entities selected, hit the delete key on your keyboard to erase the lines and arcs (Figure 6.6). You will notice that you have rarely erased in any of the exercises in this textbook. Erasing lines is a task that 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 as the drawer wants based on their efficiency. If you are making lots of errors, as the World Class CAD saying goes, "You have to go slower to go faster." Accuracy counts.
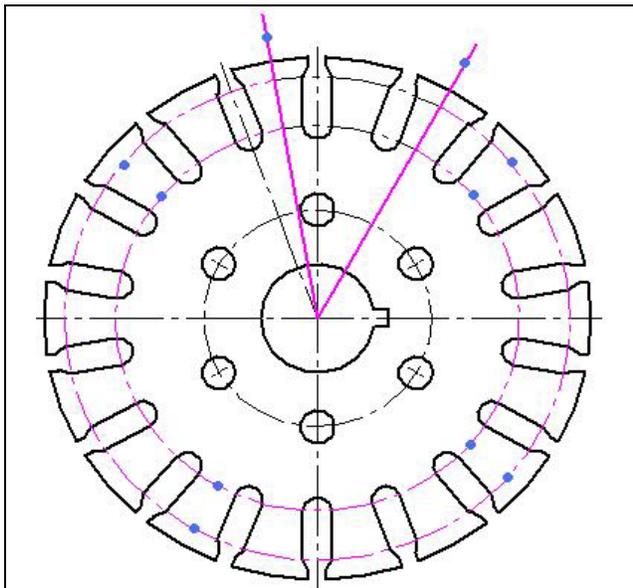

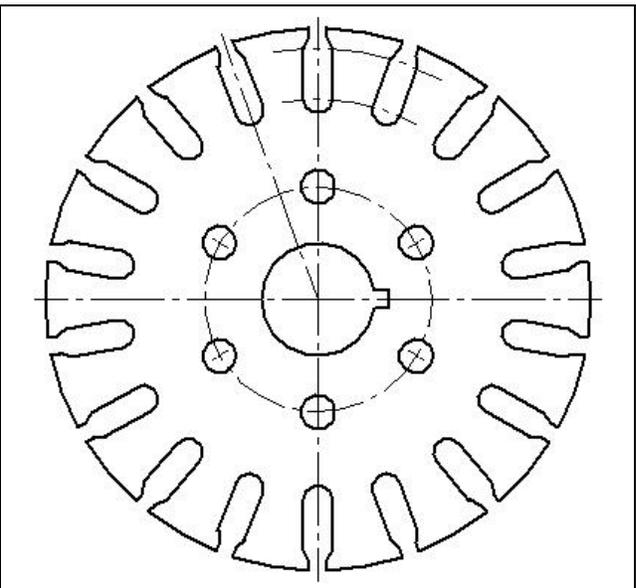**Figure 6.5 – Selecting Lines for Deletion**   **Figure 6.6 – Completed Trimming of Lines**

Using the Smart Dimension Tool to Add Diameter Dimensions

Dimensioning is the easiest task to be performed in CAD and the diameter is the easiest dimension to place. Begin by selecting the **Smart Dimension** tool on the **Dimension** panel under the **Home** tab. The Smart Dimension Screen will appear on the left of the screen.   In the Format section of the window, change the Dimension Style to **ANSI (inch)** and in the Text Scale box type "**1.00**" (Figure 6.7). Leave the Round-off as is. Now, select the 12 diameter circle and then select the **Diameter** tool on the Properties section of the mart Dimension Window. Place your cursor on the inside of the part and a line will be drawn across. The 12.000 dimension will appear in the center of the part. Left click with your mouse to place the dimension. Now, you do not want to leave the 12.000 dimension in the middle of the drawing so pick the **Select** tool and click on the 12.000 dimension, dragging it into place as in Figure 6.9.
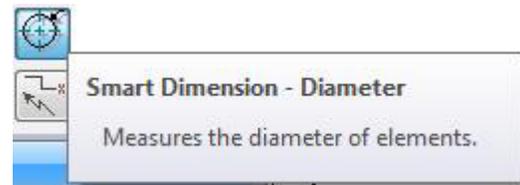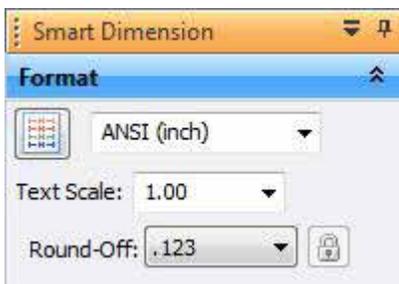
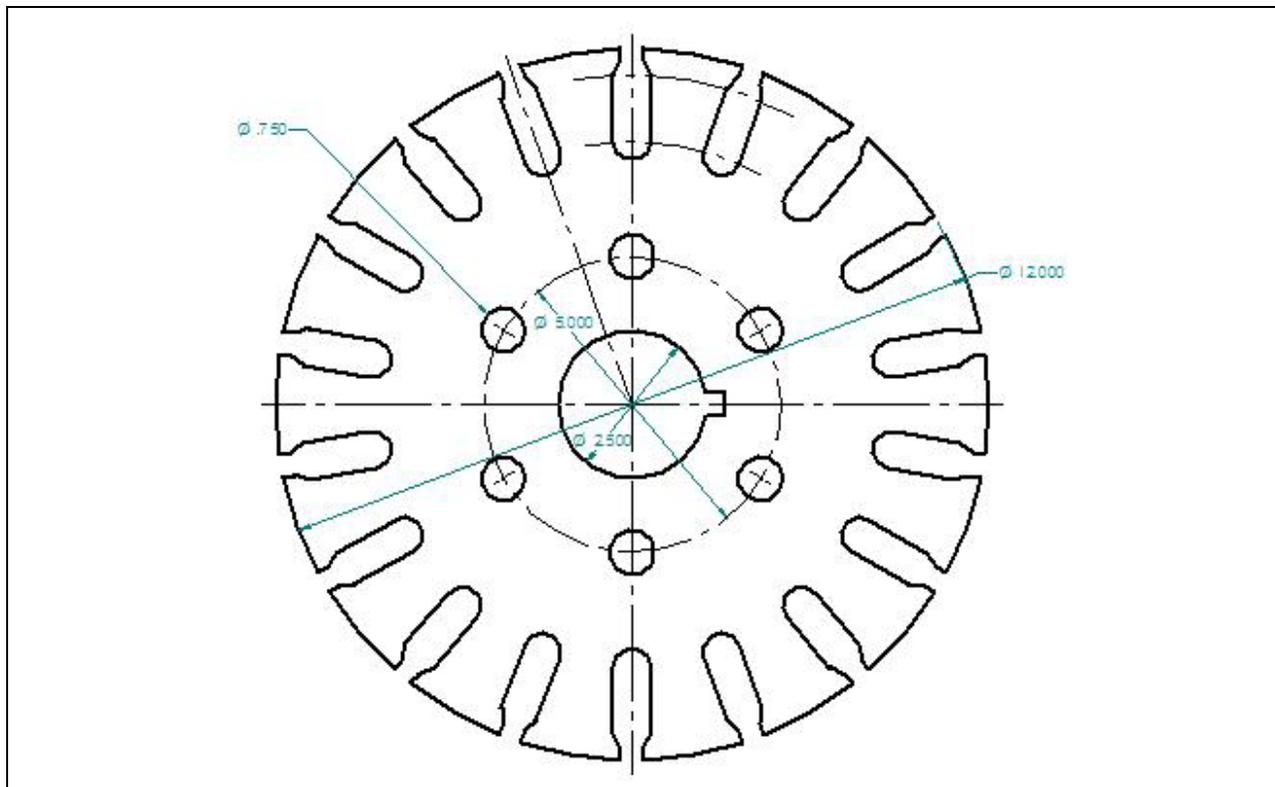**Figure 6.7 – Smart Dimension Window**   **Figure 6.8 – Diameter Tool**

Figure 6.9 – Dimensioning the Diameters Using the Smart Dimension Tool

Now, using the same procedure with the **Smart Dimension** and **Diameter** tools, pick the 5.000 inch bolt circle and place the dimension so that the line is crossing the circle. Once again, you will need to select the dimension and move it to its appropriate place (Figure 6.9). Repeat this process for the 2.500 circle as well (Figure 6.9). The final diameter is for the 0.75 inch hole. With the **Smart Dimension** tool selected, pick the upper left 0.75 hole and drag the .750 dimension text to the outside 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 at once, you are less likely to forget about one. 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 Distance Between Tool for Linear Dimension

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 **Distance Between** command from the **Dimension** panel and pick the west quadrant of the 2.5 diameter circle and the bottom endpoint of the keyway.
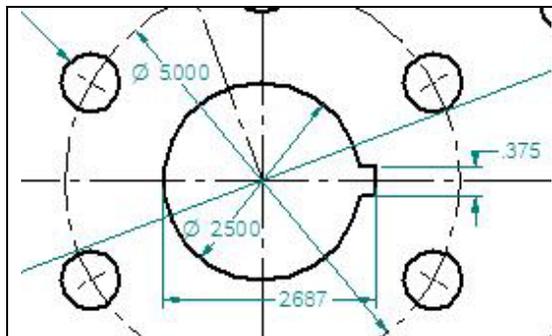
The 2.687 dimension will appear on the vertical centerline so select the 2.687 dimension and grab the text to move the numbers to the right of the line. Never allow a dimension arrow or text to be obstructed by a center, hidden, dimension or object line, so take your time to move the dimension off any entity that will cause confusion. Now, dimension the width of the keyway as shown in Figure 6.10 and pull the .375 text into place.

Figure 6.10 – Using Distance Between Tool to Dimension Keyway

Using the Smart Dimension Tool to Add a Radius Dimension
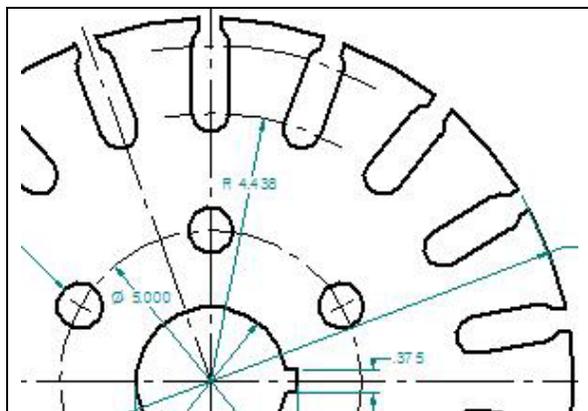
To describe the inner bolt circle of the slot, place a radius on the inner arc as shown in Figure 6.11. Select the **Smart Dimension** tool on the **Dimension** panel. Then, pick the inner arc and the **Smart Dimension** tool will automatically dimension the distance as a radius from the center. Move the text of the dimension out of the way of anything that may be blocking it. By placing the radius dimension as shown, the Distance Between Dimension can be added to measure the distance to the second arc.

Figure 6.10 – Using Smart Dimension Tool to Add 4.438 Radius Dimension

Using the Distance Between Command for Aligned Dimensions

The By 2 Points option of the **Distance Between** command is easily used to place a dimension in alignment with two picked reference points. You are going to use this option to place a dimension on the drawing to define the distance from the end of the 4.438 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 IntelliSketch settings.

Go to the IntelliSketch panel and click the small arrow at the bottom right of the panel. Just like in the previous chapter, you will need to go to the **Relationships** tab of the IntelliSketch Window and make sure that the Midpoint checkbox is selected and click OK. Feel free to change IntelliSketch 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 By 2 Points option in the drop-down box on the Properties section of the **Distance Between** command (Figure 6.11). Dimensions cannot be used to make other dimensions in the Solid Edge program so you will have to use the midpoints of the two arcs to draw the 1.125 dimension and then move in line with the 4.438 radius.
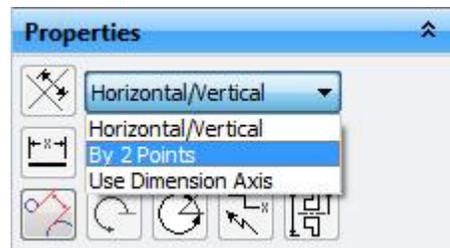
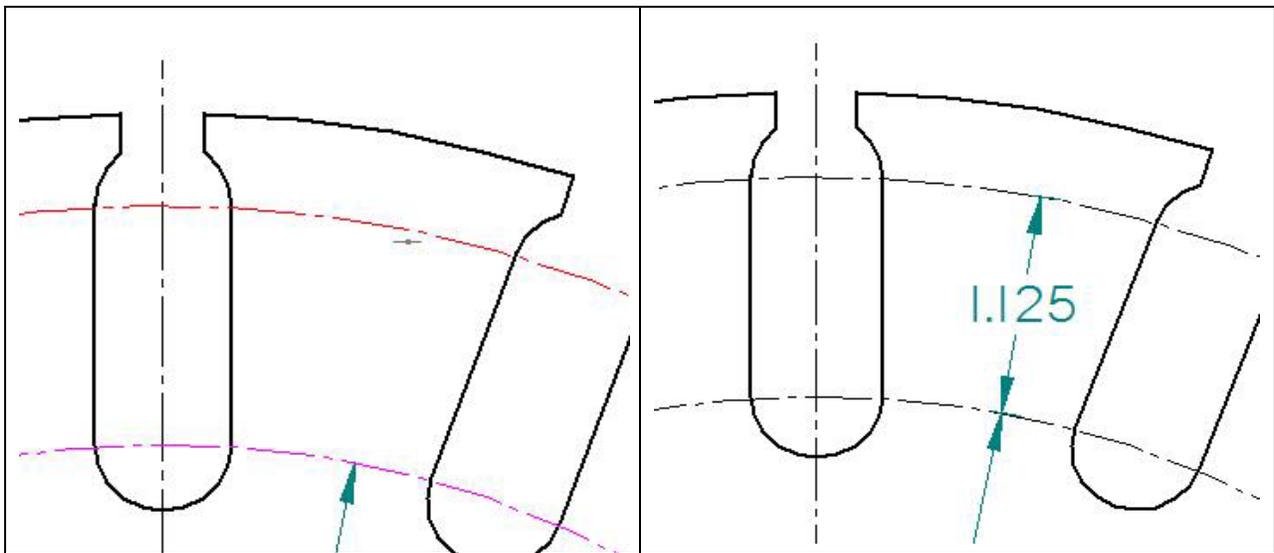**Figure 6.11 – Selecting By 2 Points Option**

Figure 6.12 – Picking the Midpoint **Figure 6.13 – Finished 1.125 Dimension**

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.

Using the Angle Between Dimension Tool

There are two angular dimensions to place in the circular drawing. Remember how easy thee command is to use? Select the **Angle Between** command on the **Dimension** panel. This command consists of three selections for both dimensions and will only dimension the angles if you make the selections properly.

For the first selection, pick the midpoint of the vertical centerline. Next, select the midpoint of the centerline inside of the upper right 0.75 diameter circle. For the final selection, pick the endpoint of the centerline that is at the center of the part. Now, just drag the dimension to the outside of the part and place it as in Figure 6.14. Also, the dimension should be 60.0 with one decimal place so you may need to change the round-off in the Format section of the command window.
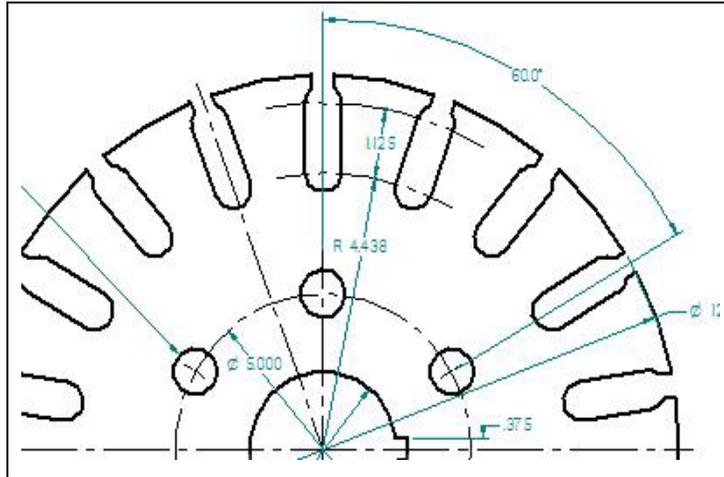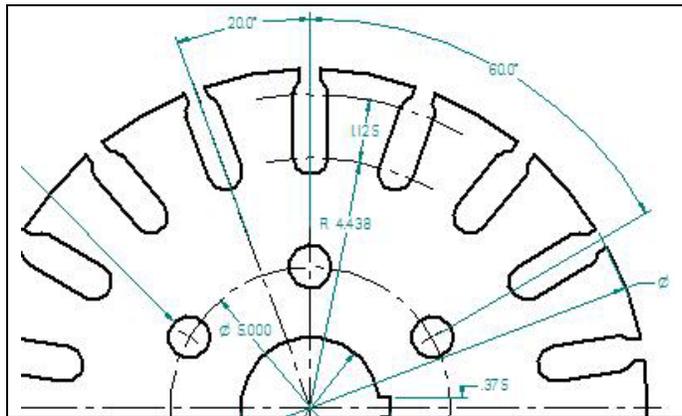

**Figure 6.14 – Placing 60.0° Angular Dimension**

For the 20.0 degree dimension, select the midpoint of the vertical centerline just like with the 60.0 degree dimension. Next, select the midpoint of the centerline to the left going through the next slot. For the third selection, pick the endpoint of the centerline at the middle of the part. Now, move the dimension so that it is exactly in line with the 60.0 degree dimension. When you have finished, your drawing should appear as in Figure 6.15.

Figure 6.15 – Placing 20.0° Angular Measurement

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. If your angular dimensions are too close to the outer perimeter, it is an easy fix. Just pick the **Select** tool and grab and drag the lines of the dimension farther away from the outer rim of the part. However, make sure that you still keep the two dimensions in line with each other when you move them.

# More Linear Dimensions with the Distance Between Tool

The last series of dimensions are located at the top quadrant of the circular problem. Select the **Distance Between** command on the Dimension to place two of these remaining measurements. For the .312 dimension, pick the vertical centerline and the left outside edge of the slot. Repeat the command for the .625 dimension, picking both outside lines on the slot.
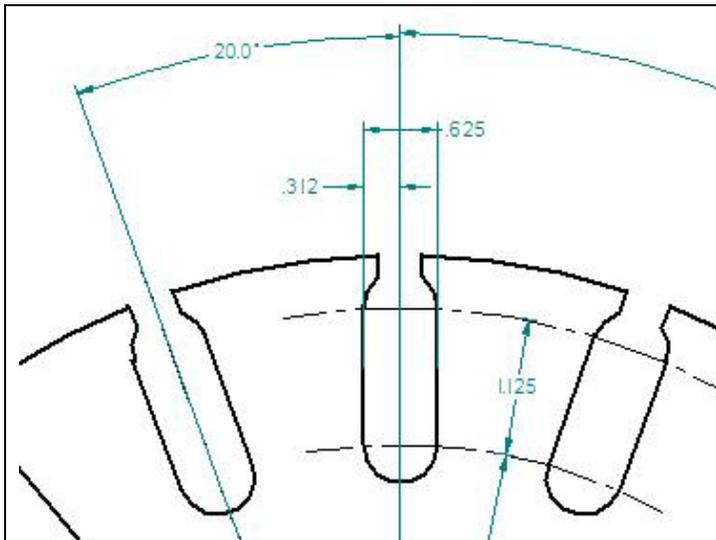
In many cases you may need to reference a detail like the slot from a centerline and then define the edge of the slot to the opposite side with another dimension. If you use the .312 dimension to define both sides, the tolerance will double when calculating .312 twice. Such is the case in Figure 6.16, so place a linear dimension for the .625 as shown.

Figure 6.16 – Using Distance Between Tool to Dimension Slot

# Another By 2 Points Dimension with the Distance Between Tool
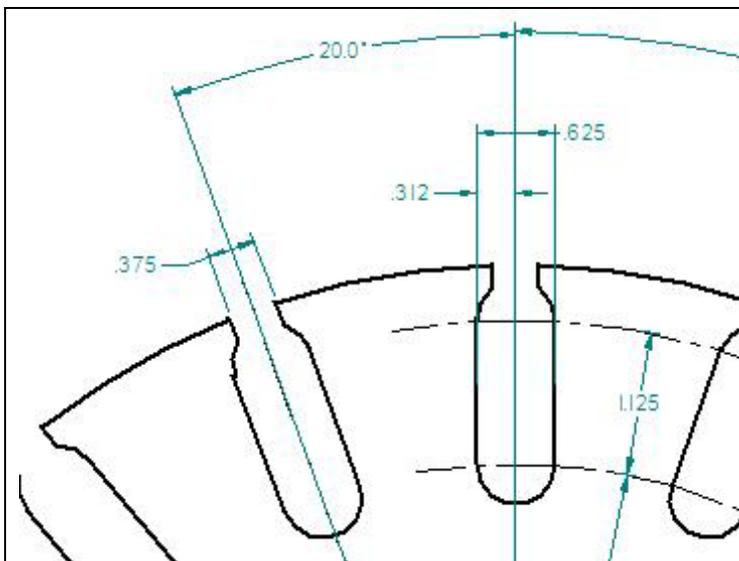
Choose the By 2 Points option of the **Distance Between** command and select the endpoint of the opening of the slot as shown in Figure 6.17. Make sure you place the dimension text to the left of the slot to avoid the crowding of dimensions between the two slots.

Figure 6.17 - Using the By 2 Points Option To Dimension Slot

Another Radius Dimension with the Smart Dimension Tool
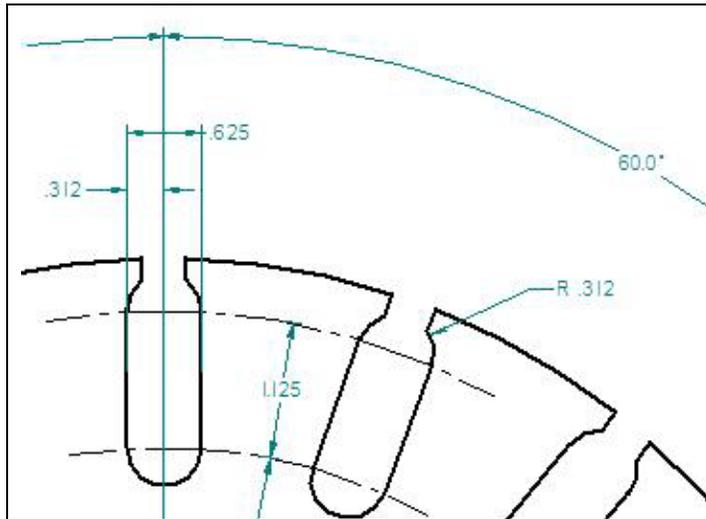
The last dimension on the circular problem is the .312 radius on the slot. Select the **Smart Dimension** tool on the **Dimension** panel. Pick the arc of the slot and the command will automatically make the .312 radius. Place the dimension as in Figure 6.18.

Figure 6.18 – Using Smart Dimension Tool to Add a Radius

* World Class CAD Challenge 22-14 * (See End of Chapter)

Adding Text Below the 0.750 Diameter Dimension

To add text to the 0.750 diameter dimension, stating that this description applies to "6 holes", select the 0.750 dimension with the **Select** tool. Next, pick the 0.750 dimension and the Edit Definition Window will appear to the left of the screen. On the Tolerance pane of the window, select the **Edit Definition-Prefix** tool (Figure 6.19). A window will appear on the screen as in Figure 6.20. In the box labeled **Subfix**, type "6 holes" (Figure 6.21). After typing in the text, click the OK button and the text will be added below the dimension (Figure 6.22).
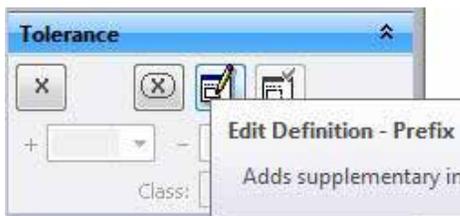

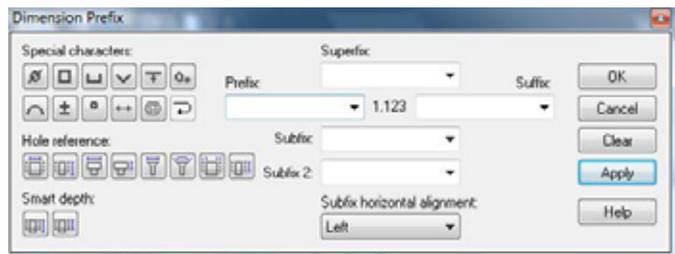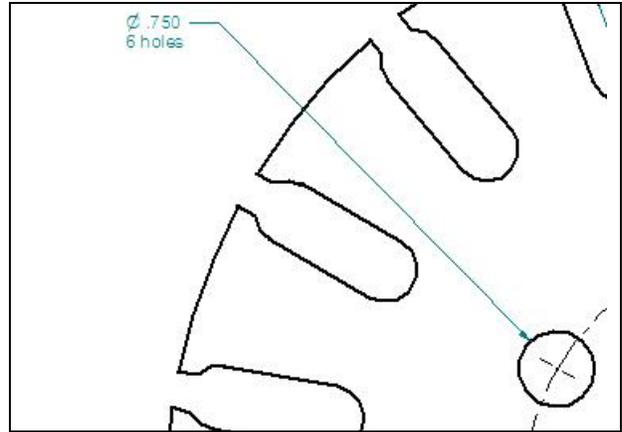**Figure 6.19 – Edit Definition Tool**   **Figure 6.20 – Dimension Prefix Window**
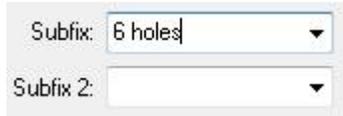
Figure 6.21 – "6 holes" Subfix   **Figure 6.22 – Finished Text**

The circular problem is finished and you should be satisfied that all the critical information is available for viewing. In Figure 6.23, 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 complete your set of notes.
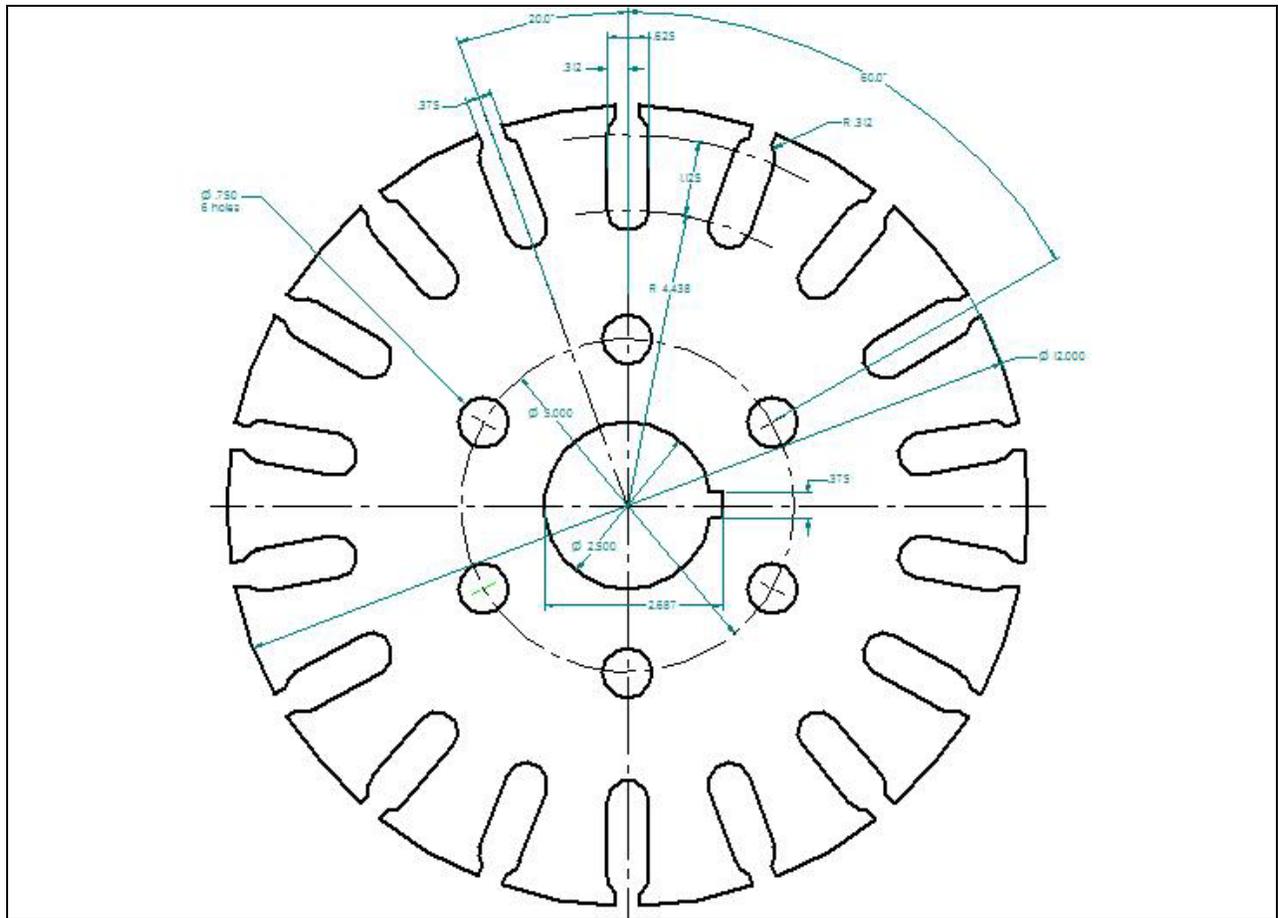

Figure 6.23 – Finished Circular Problem   * World Class CAD Challenge 22-15 *

World Class CAD Challenges for Chapter 6

* 22-14 * Create a New file, create the three new layers, set the IntelliSketch icons, draw the seven circles, move and copy the two circles, add lines for the keyway and slotted holes, do four arrays and add the 14 dimensions in 9 minutes. Continue this drill four times, each time completing the drawing in less than 9 minutes to maintain your World Class ranking.

* 22-15 * Create a New file, create the three new layers, set the IntelliSketch icons, draw the seven circles, move and copy the two circles, add lines for the keyway and slotted holes, do four arrays, add the 14 dimensions and add the text below the 0.750 circle in 10 minutes. Continue this drill four times, each time completing the drawing in less than 10 minutes to maintain your World Class ranking.