C h a p t e r

11

Dimensioning the Bracket Problem

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

1. **Dimensioning a Multiple View Drawing**
2. **Placing Center Marks on a Multiple View Drawing**
3. **Placing Linear Dimensions on a Multiple View Drawing**
4. **Placing Radius Dimensions on a Multiple View Drawing**
5. **Placing Diameter Dimensions on a Multiple View Drawing**
6. **Using the Dimension Settings Window to Modify a Dimension**
7. **The Difference Between Model and Paper Space**

Dimensioning a Multiple View Drawing

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When learning how to dimension a multiple view drawing, the goal is to portray a clear and concise visual representation of the part or assembly. A great rule of thumb is to place the dimension in the view in which the part is facing when the machining or construction practice in progress. For example, if a worker is drilling four 14 mm holes in the top plate of the Bracket, then draw the dimensions in the top view, both the x and y-axis location and the drill diameter. Likewise, in a forged or cast part, add the radius or fillets in the proper view, so the pattern maker can easily read the drawing and make the correct choices. Check every circle, line and arc at the end of the dimensioning process guaranteeing that there are not any missing measurements.

For those new CAD operators with previous experience, you already know that this Bracket is missing the draft angle and the necessary fillets on the edges to prevent a poorly made casting. Manufacturing this part from stock will add an extreme amount of time to the machining process, and therefore a large cost to the finished product. There are additional World Class CAD chapters in other text that will discuss the intricate steps of designing a cast part, so you can believe for the present that making the Bracket is a process of machining from a solid block of material.
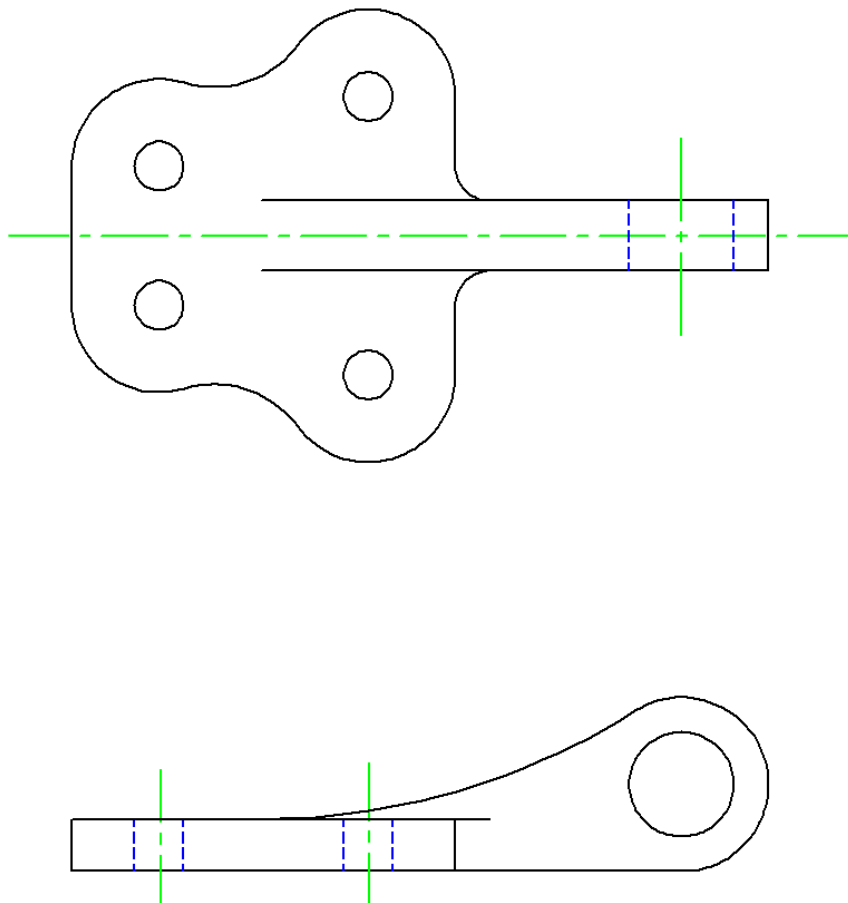
Figure 11.01 – Dimensioning the Bracket Problem

Placing Center Marks on a Multiple View Drawing

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In the Layer Control list box, select the Center Layer. When you add the center marks to the drawing, they will be green. Next, you need to place center marks inside the four circles and inside the loop detail in the front view, select the Center Mark tool on the Dimension toolbar. The mouse arrow will turn into a pick box. Select the circle on the lower left corner of the part (Figure 11.02). A green center mark will appear inside the circle. Press ENTER to repeat the command. Lastly, select the circle on the loop detail. You will have five center marks on the drawing when finished.
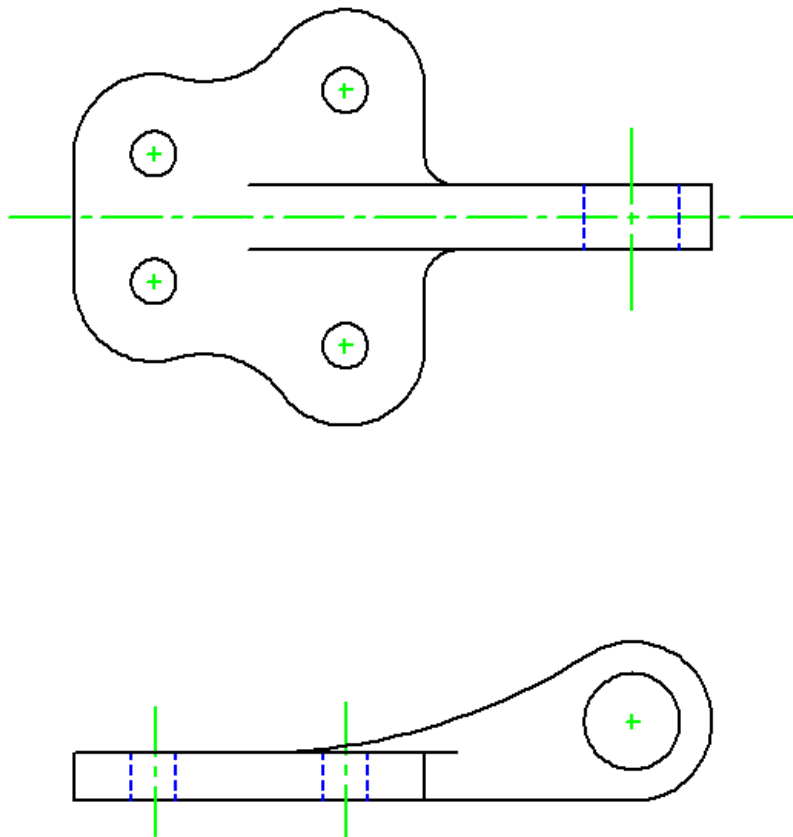
**Figure 11.02– Center Marks on the Bracket Problem**

Placing Linear Dimensions on a Multiple View Drawing

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Change the Esnap Settings to Endpoint only for inserting Linear Dimensions so you won't have any other Object Snap interfering with selecting the end of a centerline or center mark. In the Layer Control list box, select the Dimension Layer. Now, pick the Linear Dimension tool on the Dimension toolbar and insert dimensions as shown in Figure 11.03.

Draw 10, 20 and 40 mm dimensions that are coming off the centerline of the top view. Then there are 40, 60 and 80 mm dimensions between the holes. Add 15 and 20 mm dimensions showing the thickness of the material. Finally, add a 150 mm dimension on the front view, indicating the distance from the front hole to the center of the loop. This completes all the Linear Dimensions. If you have any trouble getting the placement of a dimension right, use the handles that appear when you select an already placed dimension to modify its location.
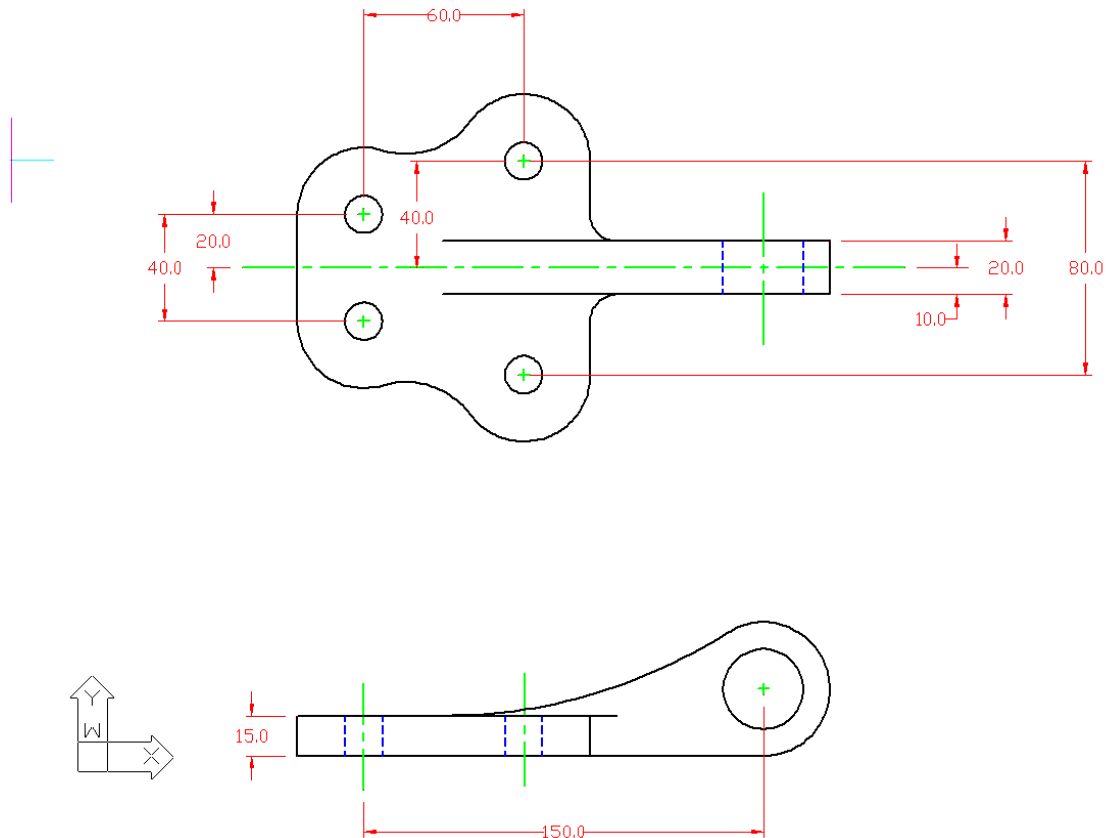
Figure 11.03 – Using the Linear Dimension Tool

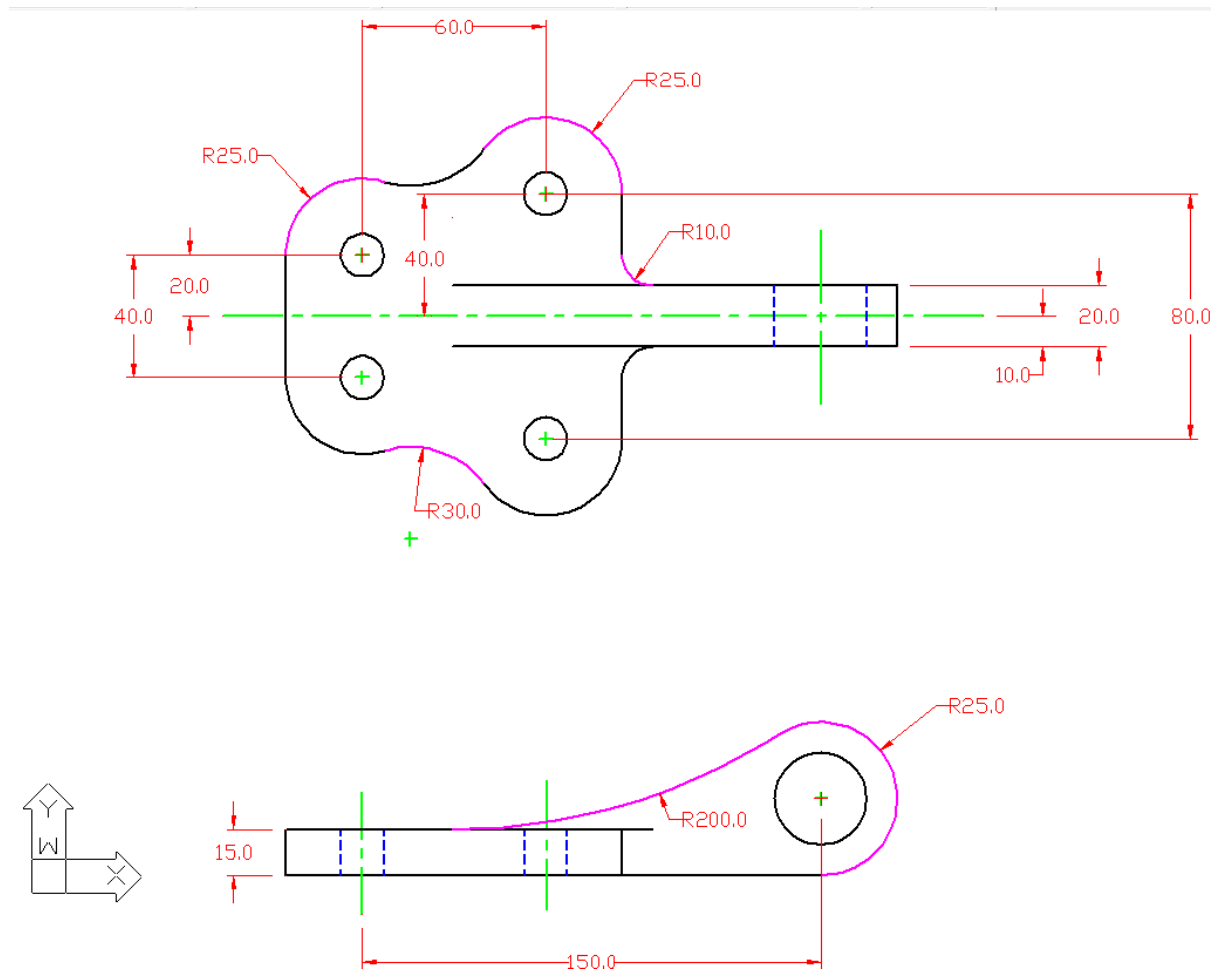## Placing Radius Dimensions on a Multiple View Drawing

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There are six radius dimensions for the Bracket problem, so you need to identify the four on the top view and two in the front view. Select the Radius Dimension tool on the Dimension toolbar and the mouse pointer will change from an arrow to a pick box to allow you to choose an arc or circle. In Figure 11.03, you can see these arcs shown in magenta. There are 10, 25, 25 and 30 mm arcs in the top view, and 25 and 150 mm arcs in the front view.

On the top view, you probably want to dimension both 25 mm arcs because there are so many curved surfaces that will not be labeled. Showing a centerline down the middle of the

Bracket problem's top view means that whatever you dimension on one side is the same on the other side of the line of symmetry.

Typically, you will not place dimensions inside the part because the placement of arrows, dimension lines, and extension lines could interfere with a person's ability to read the drawing, but you can place the 200mm radius dimension in the part. Placing the measurement on the inside of the arc will give you a 200 mm leader, so place the dimension as shown in Figure 11.04.


Figure 11.04 – Using the Radius Dimension Tool

Placing Diameter Dimensions on a Multiple View Drawing
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There are two diameter dimensions for the Bracket problem, so you need to identify the one on the top view and one in the front view. Select the Diameter Dimension tool on the Dimension toolbar. The mouse pointer will change from an arrow to a pick box to allow you to choose a circle in which to place a diameter. In Figure 11.05, you can see the two circles shown in magenta. There are 14 and 30 mm circles

to dimension. After placing both dimensions in the drawing, you will want to add the text "4 holes" to the 14 mm hole diameter.
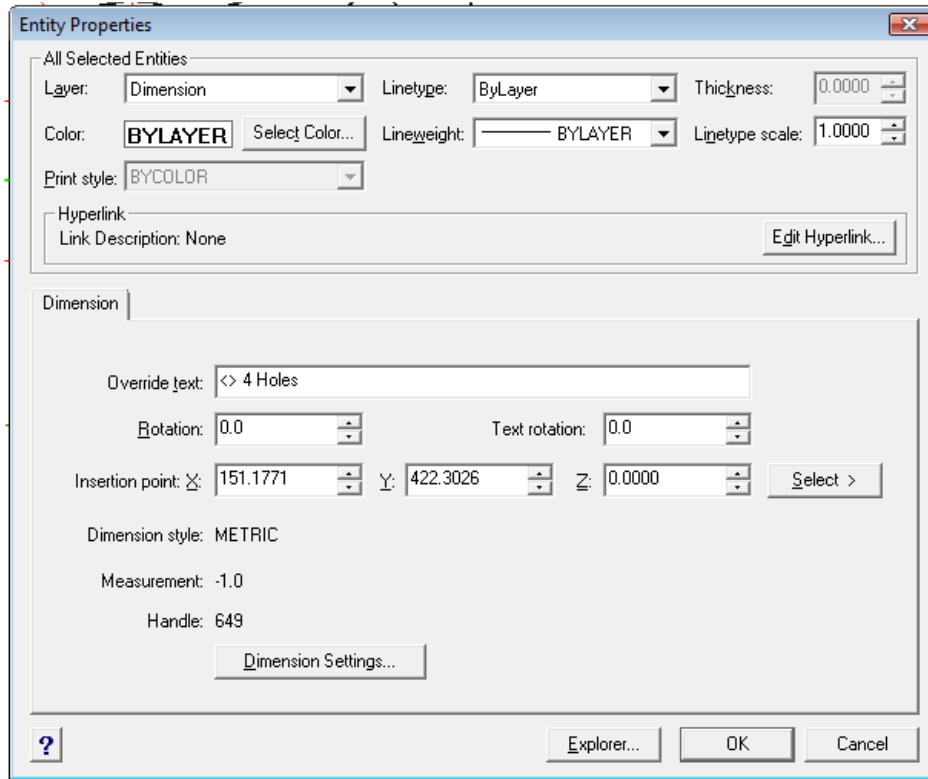
Figure 11.05 – Adding a Note to a Dimension

To add text the Ø 14 mm dimension stating that this description applies to "4 holes," select the "Text…" tool on the Text toolbar. The Entity Properties window will appear in the graphical screen. The "<>" symbol represents the numerical dimension, so place the cursor behind the "<>" symbol, type "4 holes" and select "OK" to close the window.
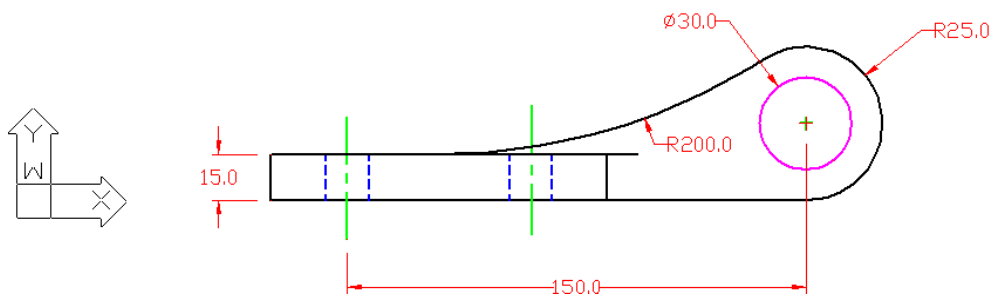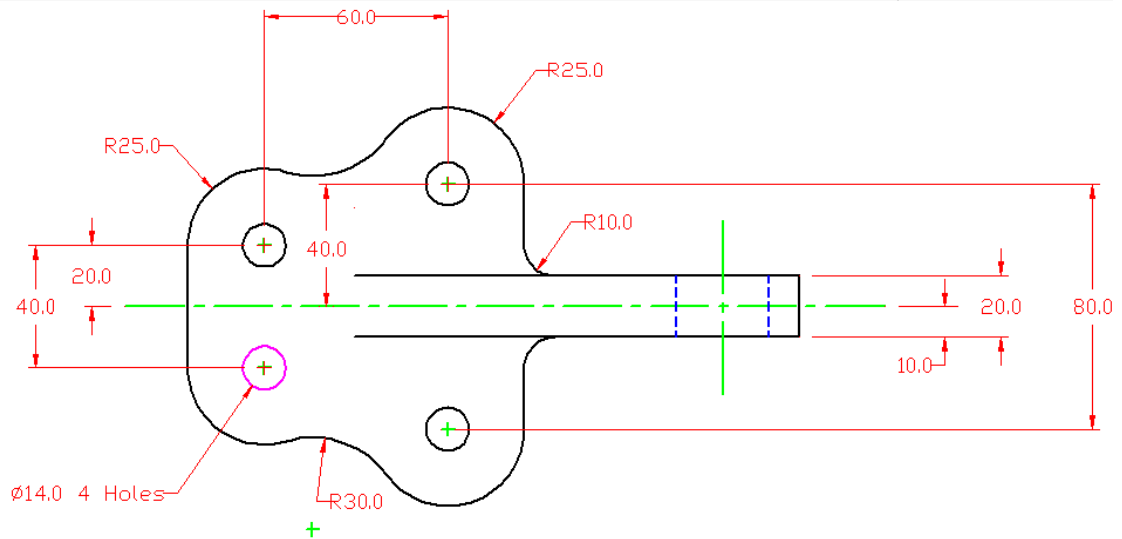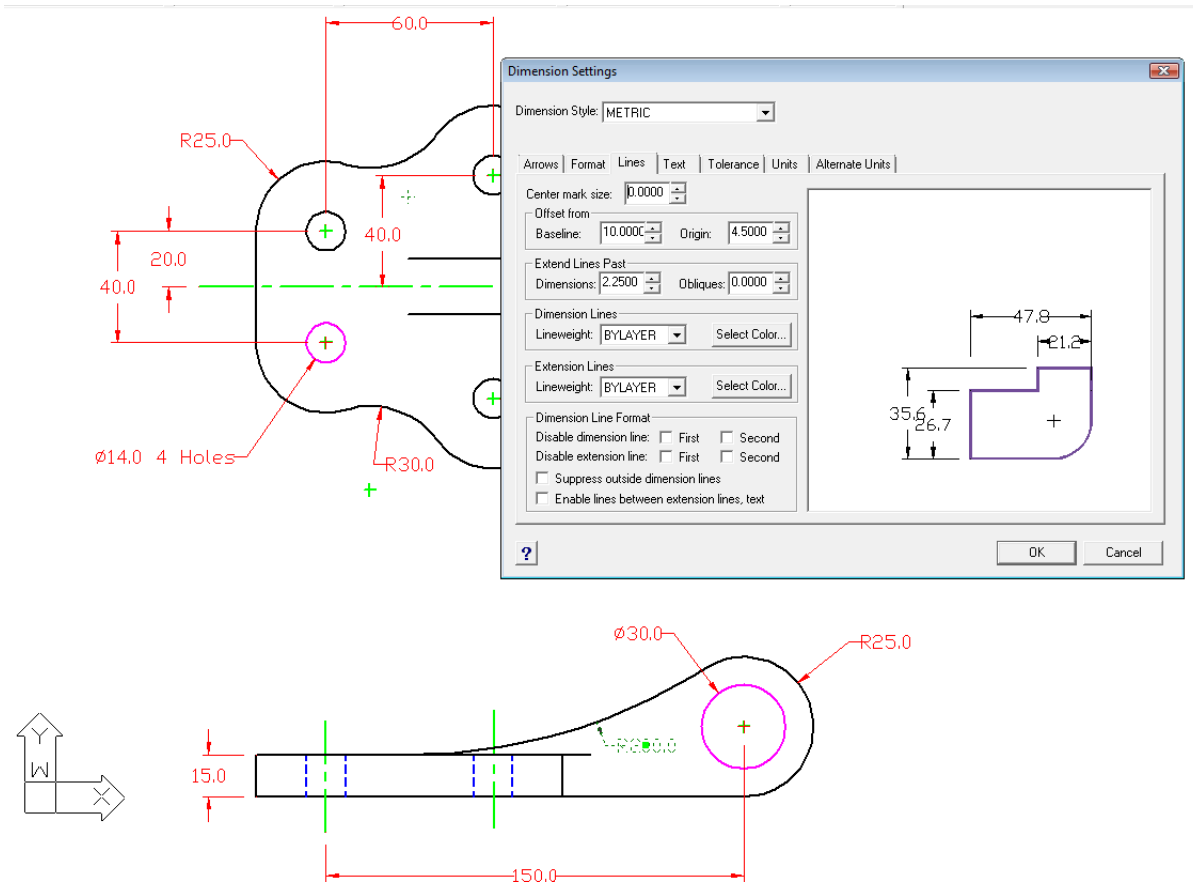
Figure 11.06 – Using the Diameter Dimension and Edit Text Tool

In the Layer Control list box, select the Center Layer. Then you can add an additional center mark to the drawing at the R30 dimension on the top view.

Using Dimension Settings to Modify Individual Dimensions
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Next, to turn off the center marks on the radius and diameter dimensions, zoom out to view the entire Bracket Problem and select one of the radius and diameter dimensions. Double click on the dimension to bring up the Dimension Settings window. If you look carefully you will see that the window is missing a few buttons, like the New and Save buttons that would normally appear on the Dimension Settings window. This is because you will only be able to modify the properties of the dimension that you double-clicked. Go to the Lines tab and change Center Mark Size to 0 (Figure 11.07). This will remove the red center mark from the selected dimension. Do the same for all radial and diameter dimensions (Figure 11.08).

Figure 11.07 – Modifying a Dimension with the Properties Tool
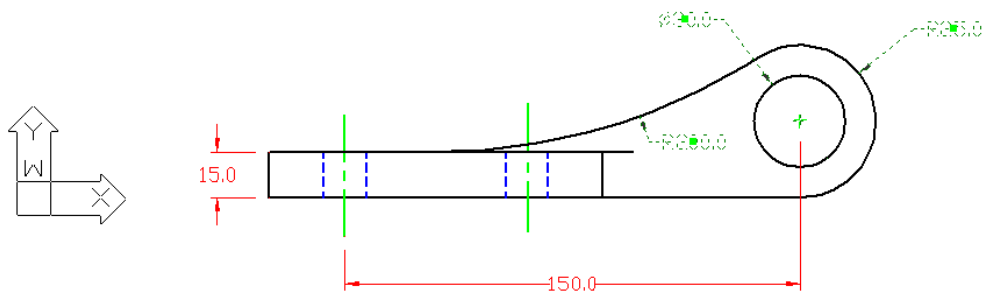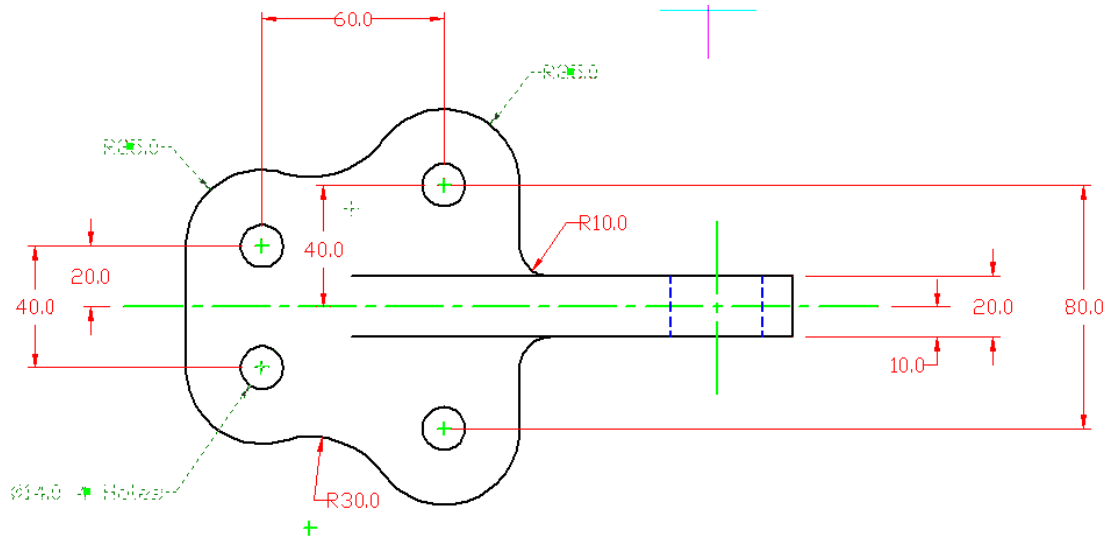
Discussing the Difference Between Model and Paper Space
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Now that you are finishing the dimensioning of the Bracket problem, the next step is to enter another new realm of the computer aided design world: paper space. So far to date, you have been operating in Model Space, as you can see on the Status Bar that you are in Model Space (Figure 11.08).


Figure 11.09 – The progeCAD Command Line and Status Bar

When you begin the next chapter, "Placing the Bracket Problem in a B Size Border," you will be working in Paper Space. In the next textbook, you will be working in both Model and Paper Space. Model Space is for the 3D part or Assembly. Paper Space is used to show the part or assembly with the dimensions and notes on orthographic views. You can draw in three

dimensions in Model Space. Paper Space is the two-dimensional surface surrounding Model Space like the thin film around a bubble of air. From Paper Space, you can look into Model Space from any direction, and have those orthographic views lie next to each other in the correct projections just as if you were drawing them manually. After completing the next chapter, you will be able to plot a drawing in the "B" size border and release the signed print to a production facility.

Below is the Bracket problem with center marks and dimensions. Try to draw the exercise to the World Class standard to maintain your status.
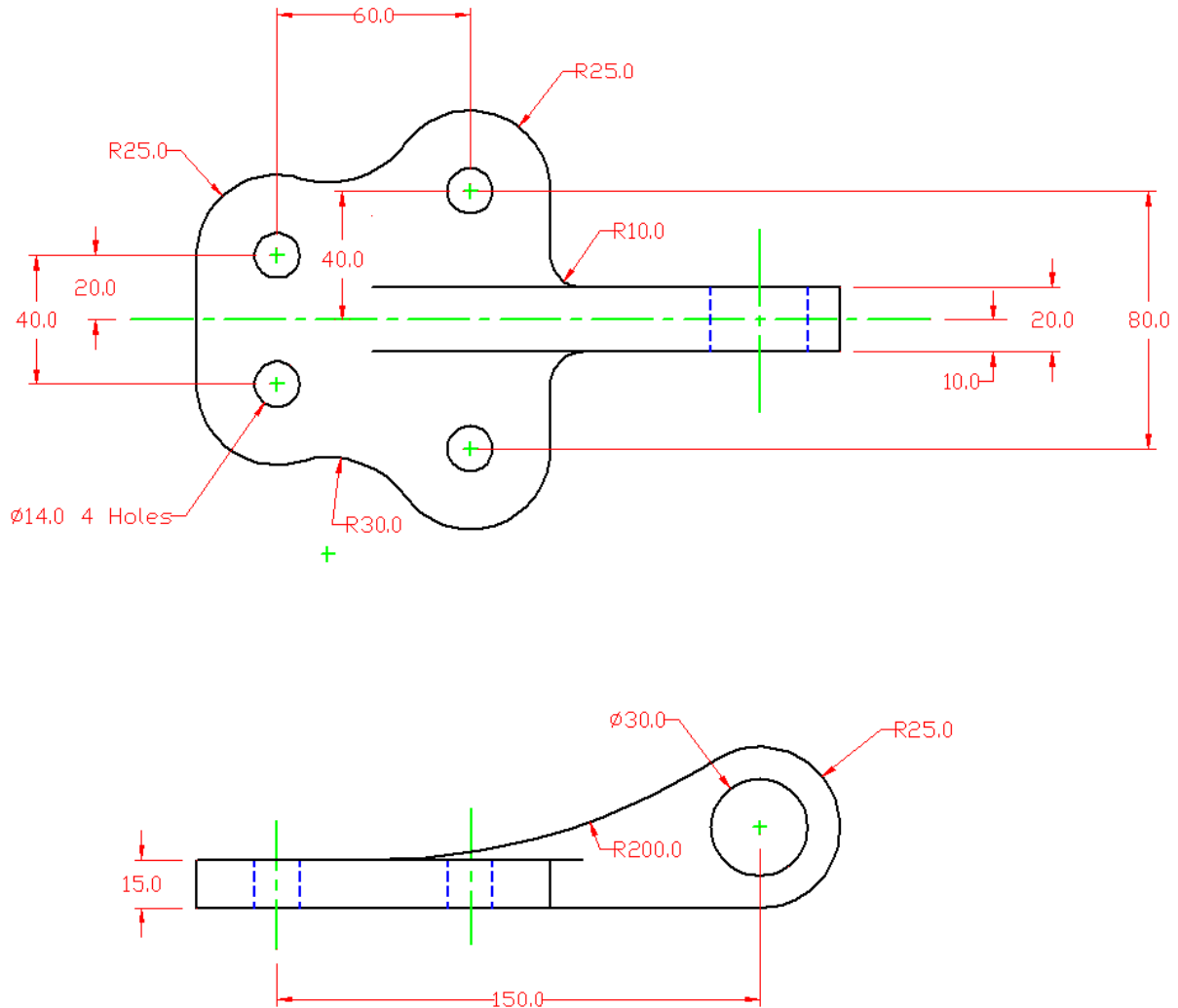
Figure 11.10 – The Bracket Problem with Dimensions

* World Class CAD Challenge 100-22 * - Create a New file using the Metric template, and draw the 66 entities shown in Figure 11.09 in 5 minutes 30 seconds. Continue this drill four times, each time completing the drawing under 5 minutes 30 seconds to maintain your World Class ranking.