

# Chapter 11

## Dimensioning the Bracket Problem

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

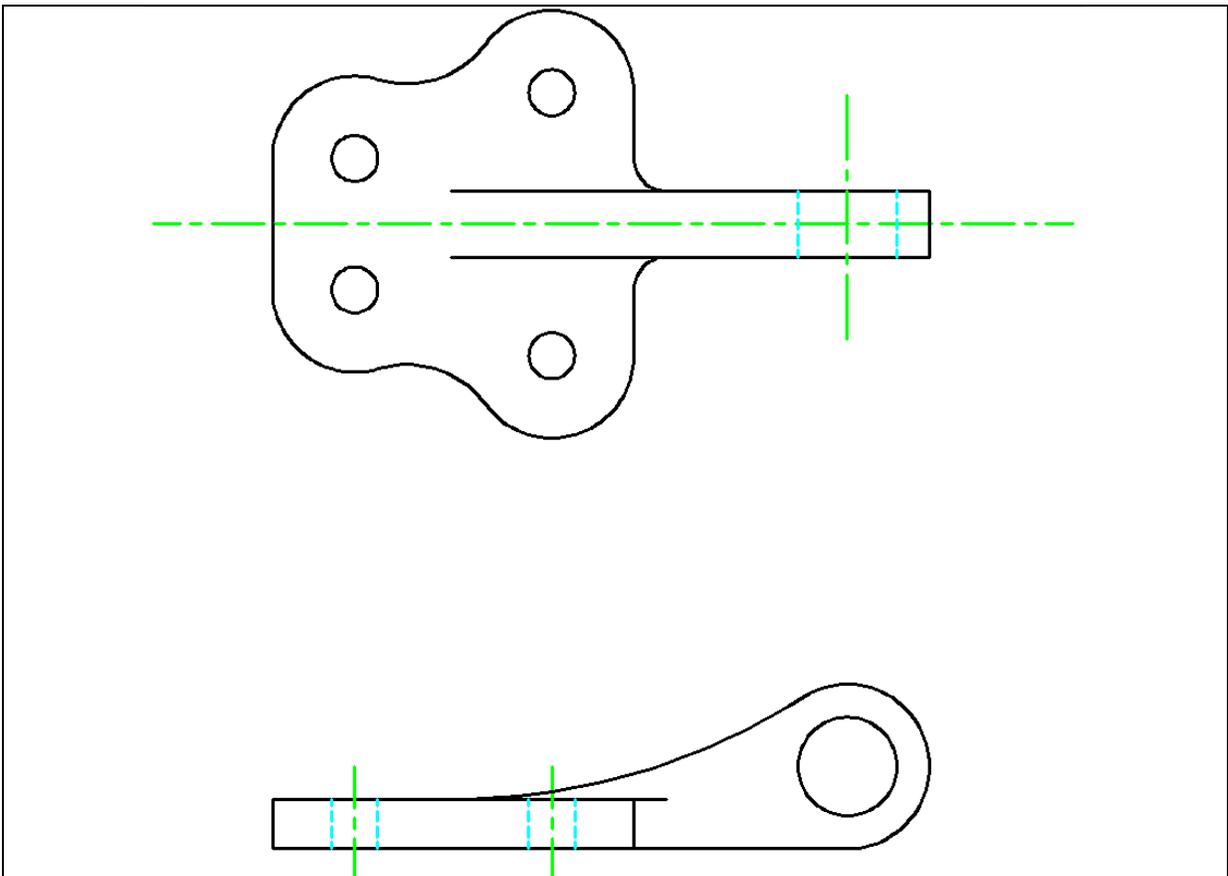
- **Dimensioning a multiple view drawing**
- **Placing center marks on a multiple view drawing**
- **Placing linear dimensions on a multiple view drawing**
- **Placing radial dimensions on a multiple view drawing**
- **Placing diameter dimensions on a multiple view drawing**
- **Using the Properties tool to modify a dimension variable**
- **Discussing the difference between Model and Paper space**

## Dimensioning a Multiple View Drawing

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A properly dimensioned multiple view drawing portrays 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 is 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 that the pattern maker can easily read the drawing and make the correct production choices. Check every circle, line, and arc at the end of the dimensioning process to make sure you haven't missed any measurements.

CAD operators with previous drafting experience may recognize 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 texts 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.

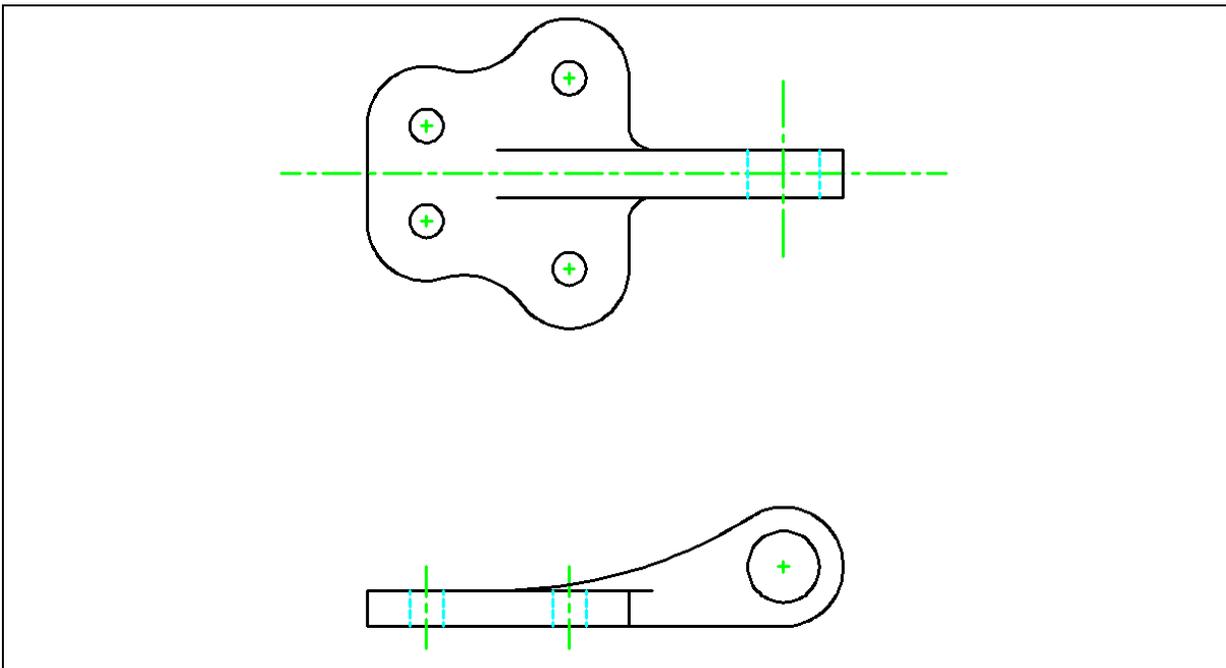


**Figure 11.01 – The Bracket problem ready for dimensioning**

## Placing Center Marks on a Multiple View Drawing

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Select the Center Layer from the Layer Control list box. Now when you add center marks to the drawing they will reside on the Center layer. You need to place center marks inside the four 7.0 circles and inside the loop detail in the front view. Select the Center Mark tool on the Dimensions panel under the Annotate tab. The mouse arrow will turn into a pick box. Select the 7.0 circle on the lower left corner of the part (Figure 11.02). A green center mark will appear inside the circle. Hit ENTER to repeat the command for each of the circles. 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 Osnap settings to Endpoint only in order to insert Linear Dimensions without other Object Snap modes interfering. In the Layer Control list box, select the Dimension Layer. Now, pick the Linear dimension tool on the Dimensions panel and insert dimensions as shown in Figure 11.03. Draw a 10, 20 and 40 mm dimension from the centerline of the top view. Then draw 40, 60 and 80 mm dimensions between the two holes. Add the 15 and 20 mm dimensions showing the thickness of the material. Finally, add the 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.



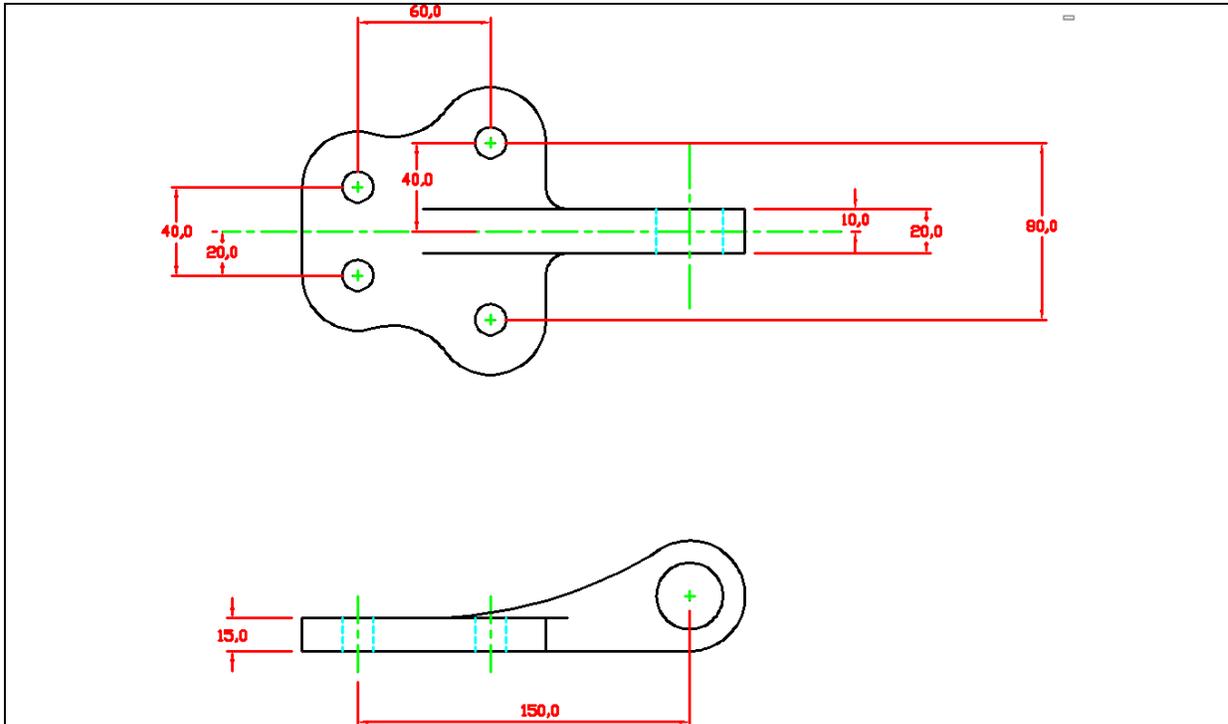


Figure 11.03 – Using the linear dimension tool

## Placing Radial Dimensions on a Multiple View Drawing

There are six radial dimensions to be placed on the Bracket problem: four on the top view and two in the front view. Select the Radial dimension tool on the Dimension panel and the mouse pointer will change from an arrow to a pick box to allow you to choose the arc or circle you wish to dimension. In Figure 11.04, you can see the arcs you need to dimension 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.



You probably want to dimension both 25 mm arcs on the top view. Not labeling every arc on one side of the centerlines could cause confusion because there are so many curved surfaces. 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 someone's ability to read the drawing. You can place the 200 mm dimension inside the part. Placing the measurement on the outside of the arc will display a long leader from the center of the radius, which may be in the middle of the top view. Place the dimension as shown.

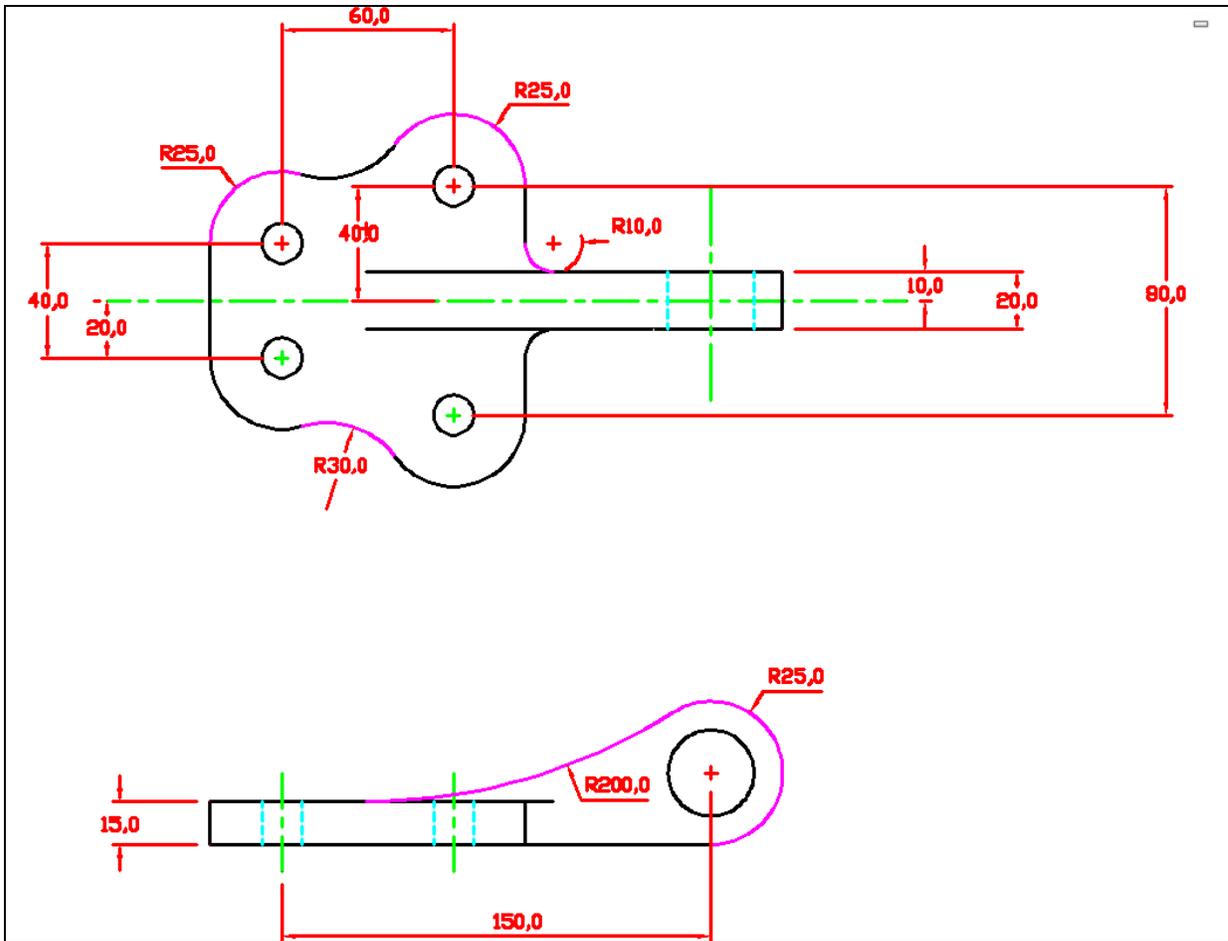
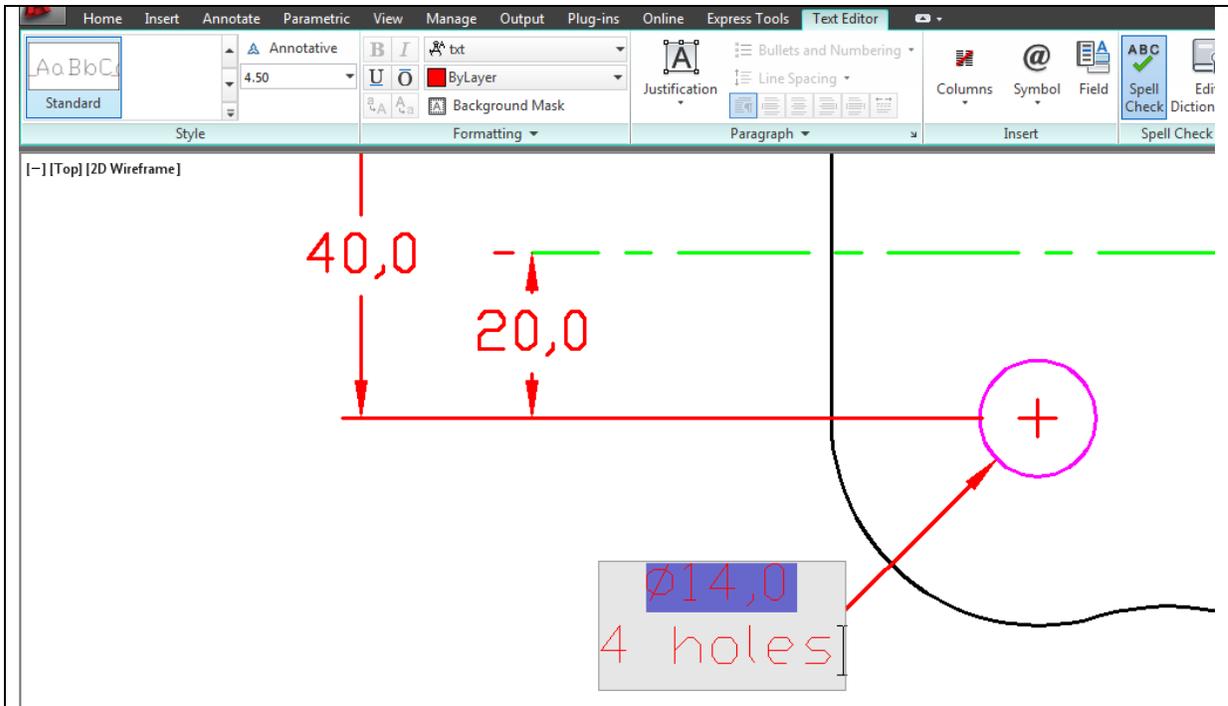


Figure 11.04 – Using the linear dimension tool

## Placing Diameter Dimensions on a Multiple View Drawing

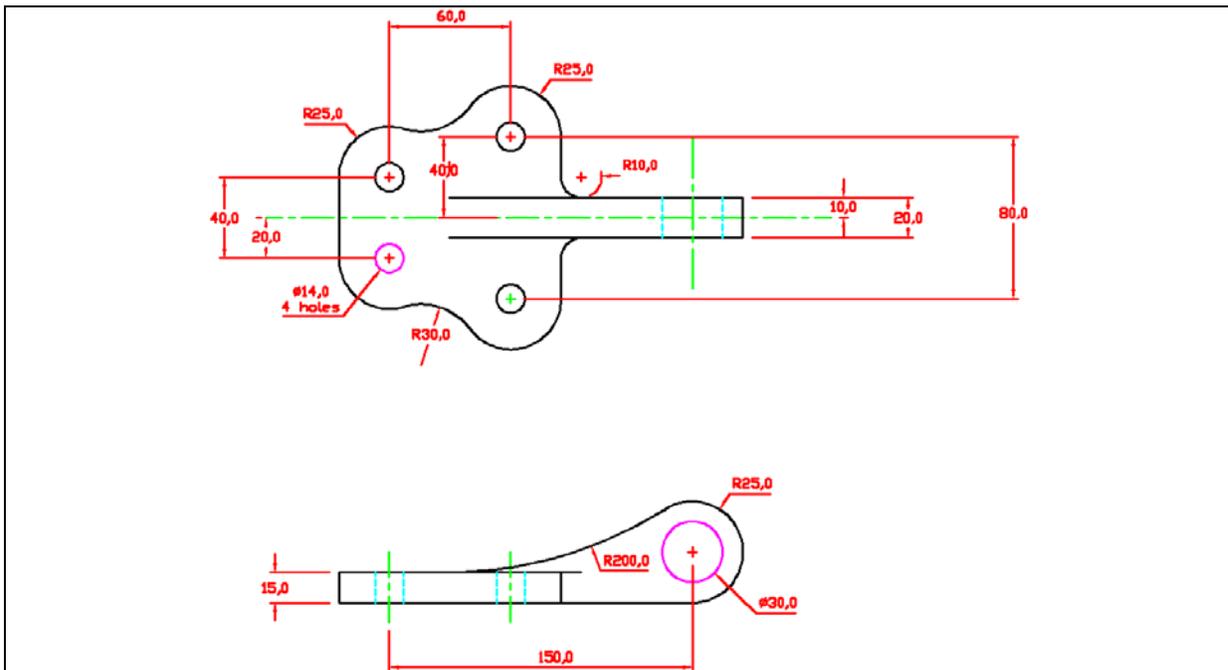
There are two diameter dimensions on the Bracket problem: one on the top view and the other in the front view. Select the Diameter dimension tool on the Dimension toolbar and the mouse pointer will change from an arrow to a pick box to allow you to choose the circle you wish to dimension. You can see the two circles shown in magenta in Figure 11.06. There are 14 and 30 mm circles to dimension. After placing both dimensions in the drawing, you need to add the text “4 holes” to the 14 mm hole diameter.





**Figure 11.05 – Adding a note to a dimension**

To add text to the  $\varnothing$  14 mm dimension stating that this description applies to “4 holes,” Double-click the 14 mm dimension label. The Text tab and a text box will appear. Go to the end of the line that contains the 14.0 and hit ENTER to start a new line. Type “4 holes” and then close the text editor.

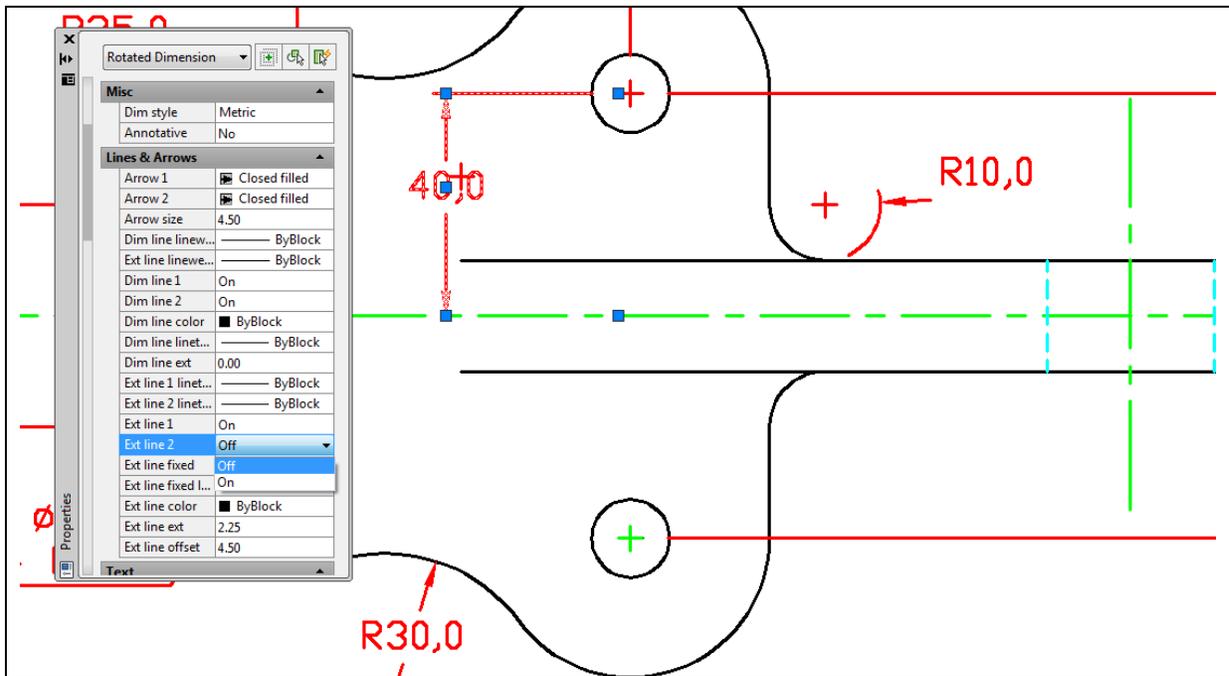


**Figure 11.06 – Using the diameter dimension and edit text tool**

## Using the Properties Tool to Modify a Dimension Variable

Select the 40 mm dimension, right click, and select Properties. The variables that represent the 40 mm dimension list the categories General, Misc, Lines and Arrows, Text, Fit, Primary Units, Alternate Units and Tolerances in the Properties window.

To remove the dimension extension line that is covering the centerline, select the Lines and Arrows category and toggle the Ext Line 2 from “On” to “Off” to turn the extension line off (Figure 11.06).



**Figure 11.06 – Modifying a dimension with the properties tool**

Next, turn off the center marks on the radius and diameter dimensions by zooming out to view the entire Bracket Problem and selecting all the radius and diameter dimensions. In the Properties menu, change the center mark label from “Mark” to “None” (Figure 11.07). The red center marks now will disappear leaving just the green marks on the Center layer.

When a CAD operator prints a drawing in black and white, one layer or one color overlapping the other may not be noticeable. Today many offices utilize color Laser printers or use color monitors on the shop floor to view drawings. Dimension extension lines lying on the top of object lines can cause trouble in interpreting the print.

In another World Class CAD textbook, you can learn about using the Visual Basic computer language to automatically make a drawing in seconds. The variables in the Properties window are manageable using the VBA coding language, so learning about how to change an entity using manual changes will help when you are advancing in your computer aided design skills.

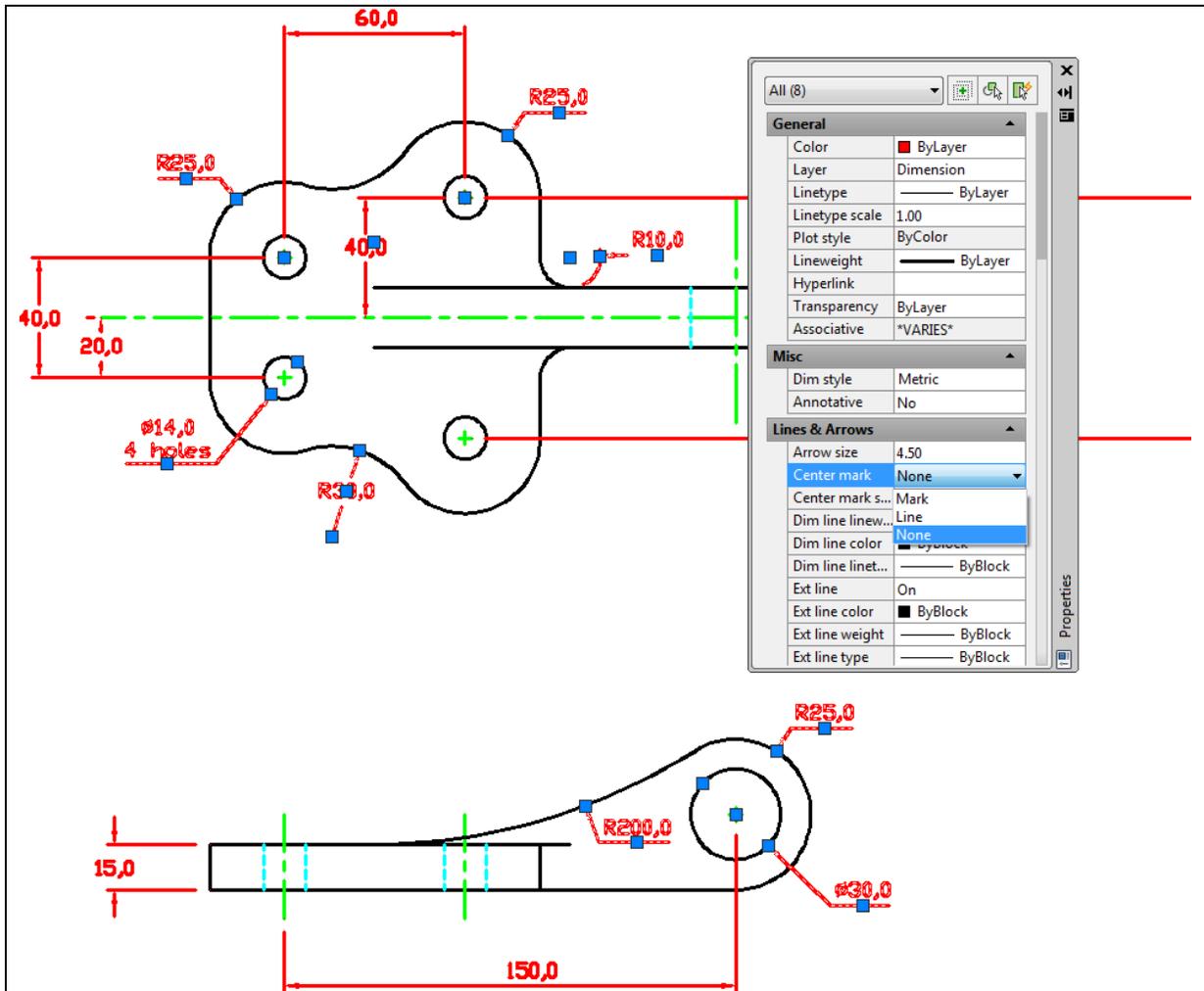


Figure 11.07 – Removing dimension center marks with the properties tool

## Discussing the Difference Between Model and Paper Space

Now that you are finished dimensioning 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.

When you begin the next chapter, you will make the transition from model space to paper space. In the next textbook, you will be working in both Model and Paper Space. Model Space is for 3D parts or assemblies. Paper Space for displaying the part or assembly and allowing you to dimension and note 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 display those orthographic views next to each other in the correct projections

just as if you were drawing them manually. In the next chapter, you will plot this drawing in a border as if you were going to 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.

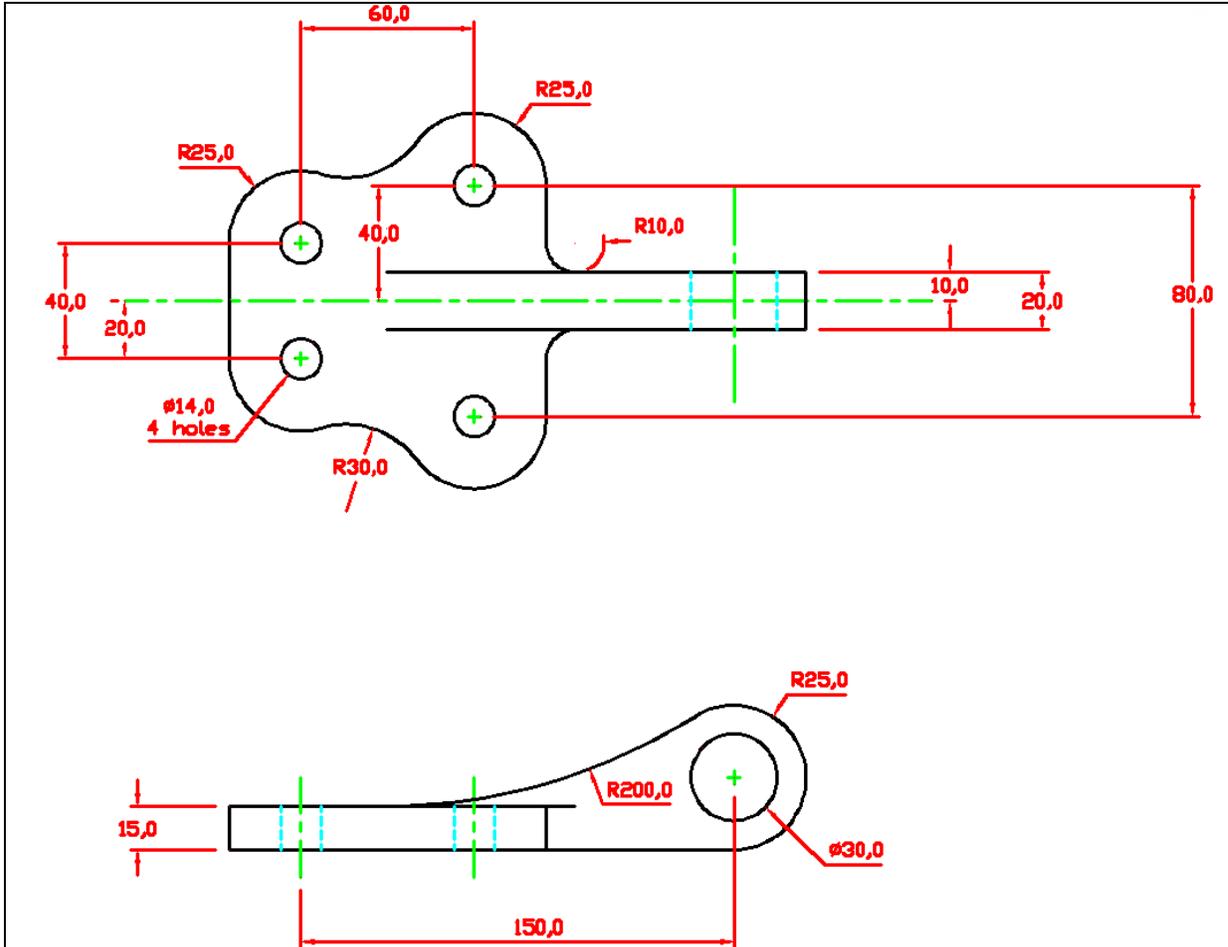


Figure 11.08 – The Bracket Problem with dimensions

\* World Class CAD Challenge 02-22 \* - Create a New file using the Metric template, and draw the 66 entities shown in Figure 11.08 in 10 minutes. Continue this drill four times, each time completing the drawing in less than 10 minutes to maintain your World Class ranking.