

# Chapter 11

## Dimensioning the Bracket Problem

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

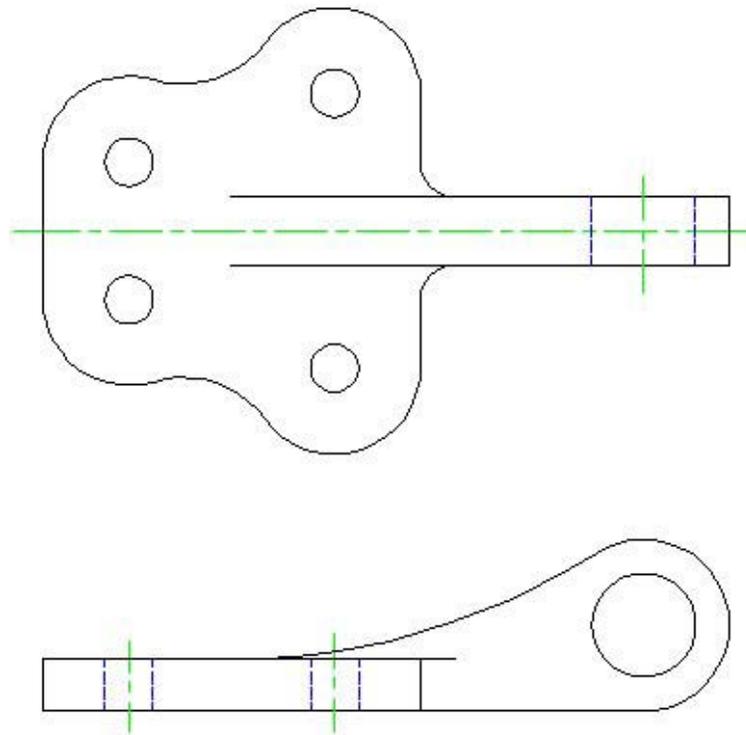
- 1. Dimensioning a Multiple View Drawing**
- 2. Placing Center Marks on a Multiple View Drawing**
- 3. Placing Linear Dimensions on a Multiple View Drawing**
- 4. Placing Radius Dimensions on a Multiple View Drawing**
- 5. Placing Diameter Dimensions on a Multiple View Drawing**
- 6. Using the Properties Tool to Modify a Dimension Variable**
- 7. Discussing 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.

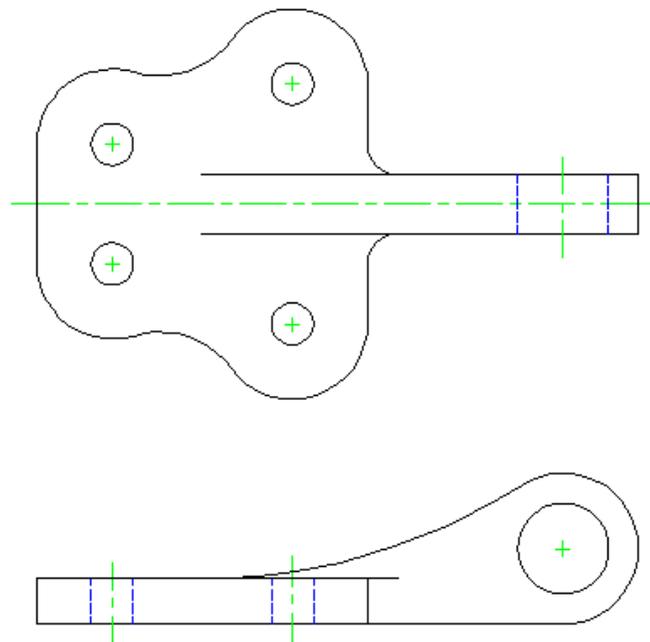


**Figure 11.01 – Dimension the Bracket Problem**

## 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 7.0 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 7.0 circle on the lower left corner of the part (See Figure 11.02). A green center mark appears 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 Osnap Settings to Endpoint only for inserting Linear Dimensions without having 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 a 10, 20 and 40 mm dimension that is coming off the centerline of the top view. Then there is a 40, 60 and 80 mm dimension between hole and hole. Add the 15 and 20 mm dimensions showing the thick ness 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.



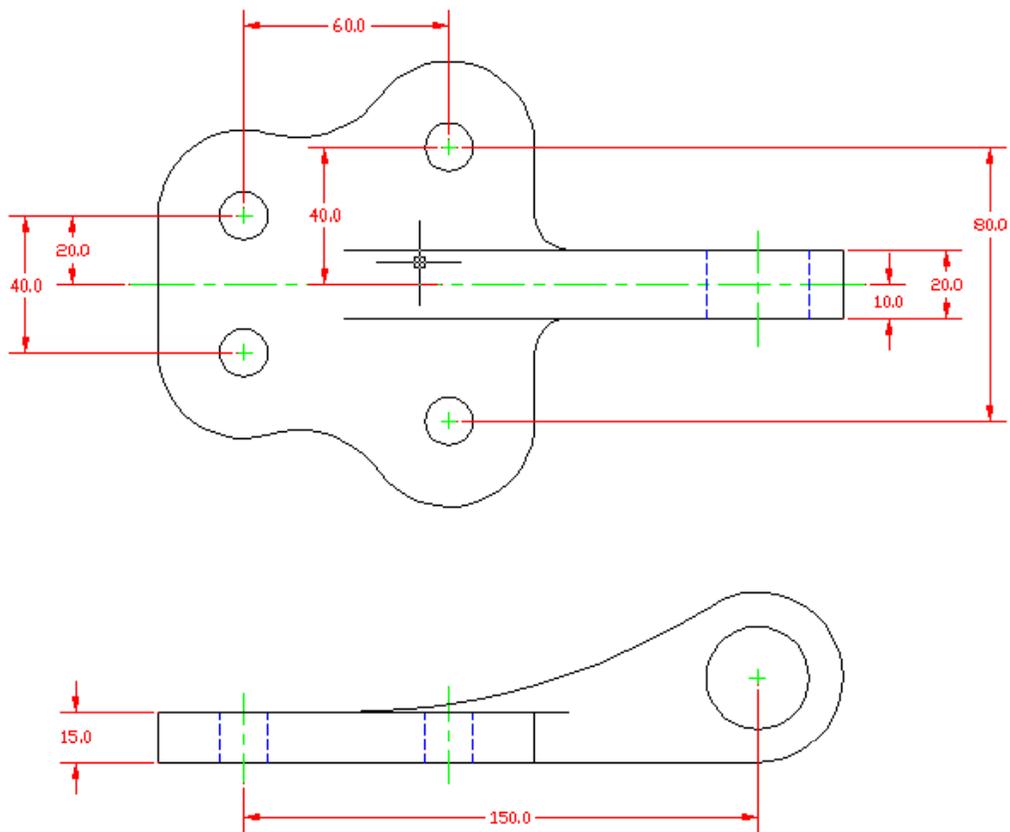


Figure 11.03 – Using the Linear Dimension Tool

## Placing Radius Dimensions on a Multiple View Drawing

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, 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. In Figure 11.03, you can see the 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, since there are so many curved surfaces that not labeling every arc on one side of the centerline, then the detail could be mistaken for another dimension. 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, since 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 200 mm 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.

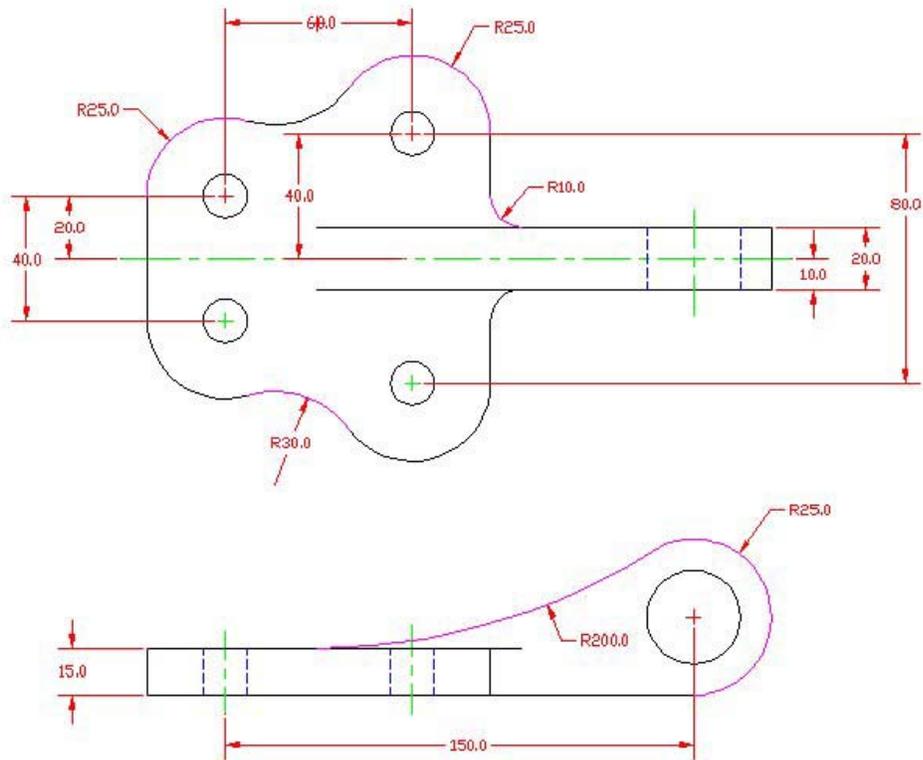


Figure 11.03 – Using the Linear Dimension Tool

## Placing Diameter Dimensions on a Multiple View Drawing

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.

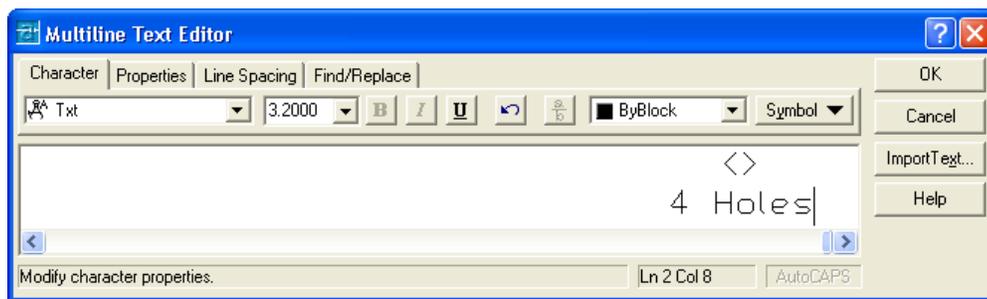
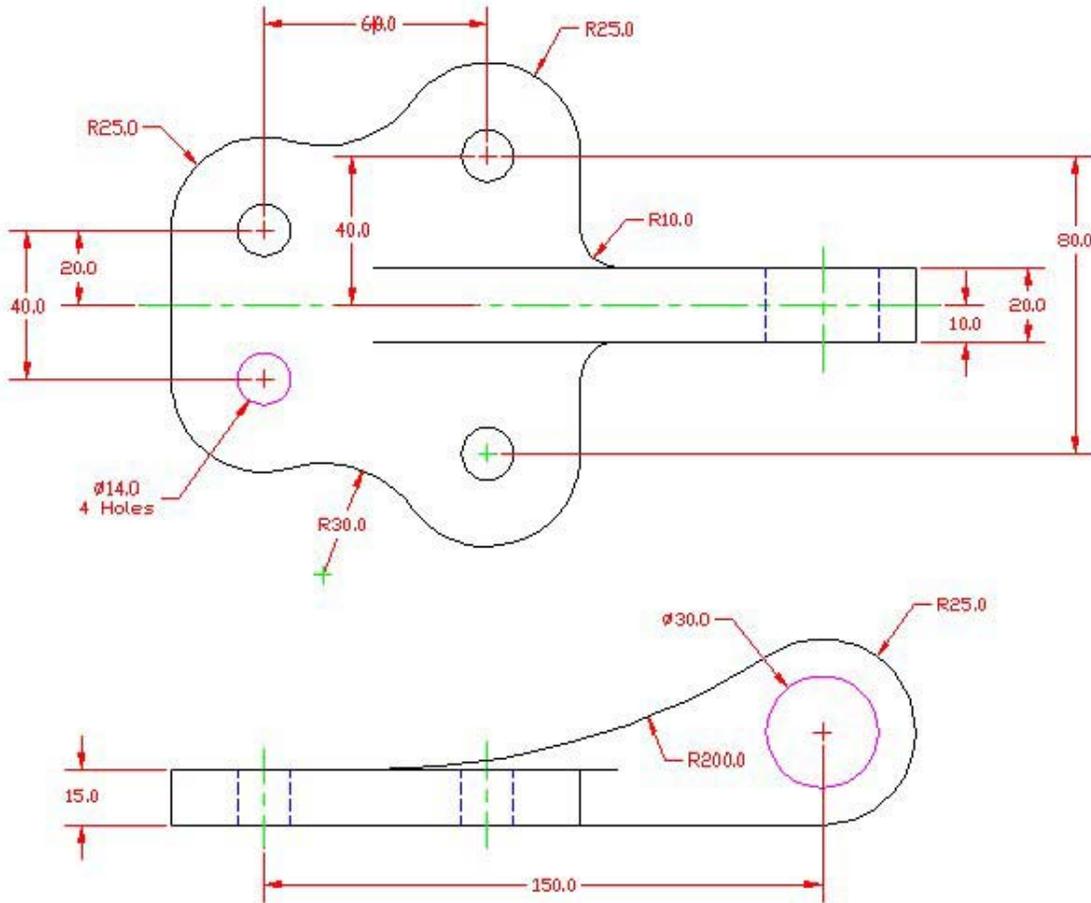


Figure 11.04 – Adding a Note to a Dimension

To add text the  $\varnothing 14$  mm dimension stating that this description applies to “4 holes”, select the Edit Text tool on the Modify II toolbar. The Multiline Text Editor window will appear in the graphical screen. The “ $\langle \rangle$ ” symbol represent the auto dimension, so place the cursor behind the “ $\langle \rangle$ ” symbol, and ENTER. On the line below the auto dimension symbol, type “4 holes” and select “OK” to close the window.



**Figure 11.05 – Using the Diameter Dimension and Edit Text Tool**

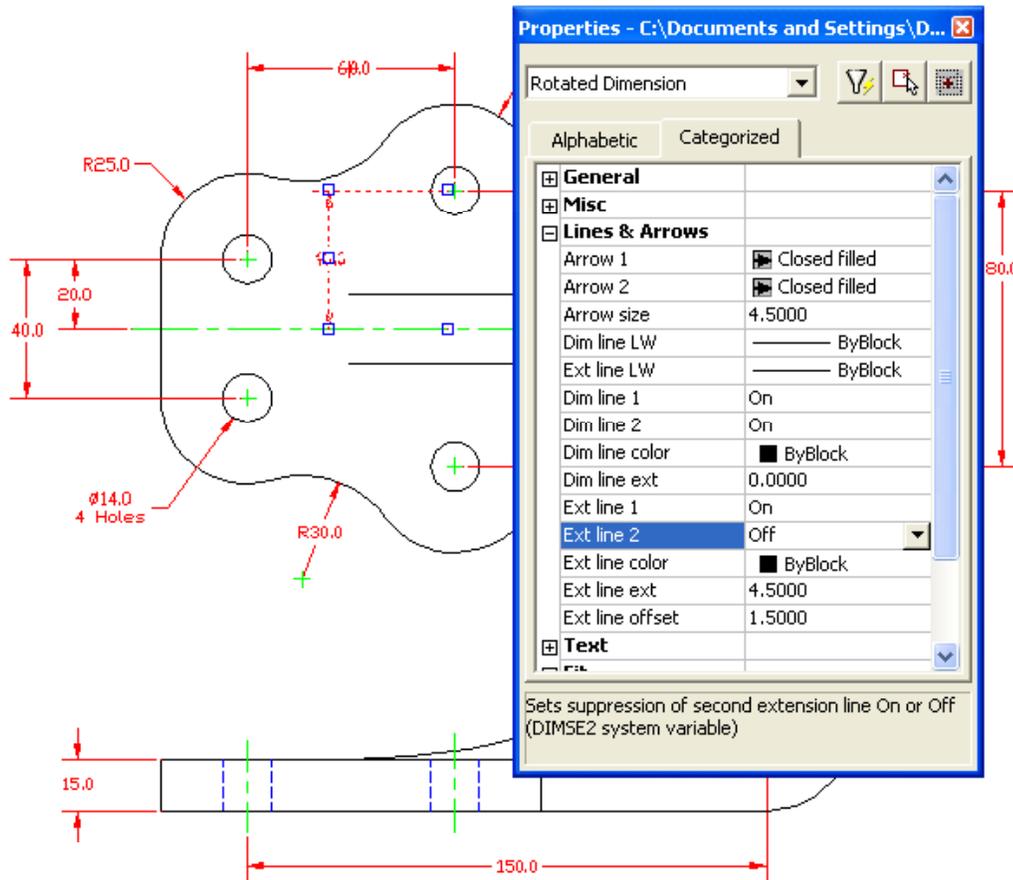
In the Layer Control list box, select the Center Layer. Then you can add an additional center mark to the drawing at the R30 dimension on the top view.

## Using the Properties Tool to Modify a Dimension Variable

Select the Properties tool on the Standard toolbar and select the 40 mm dimension. 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 double click over the “On” text to the right of the Ext Line 2 label to turn the extension line off. (See Figure 11.06)



**Figure 11.06 – Modifying a Dimension with the Properties Tool**

Next, to turn off the center marks on the radius and diameter dimensions, zoom out to view the entire Bracket Problem and select all the radius and diameter dimensions. Double click on the Center mark label to change “Mark” and the entity will change to “Line”, double click on the label and the entity will change to “None”. (See Figure 11.07) The red center marks now will disappear just leaving the green marks on the Center layer.

When the 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 the drawing. Then having 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.

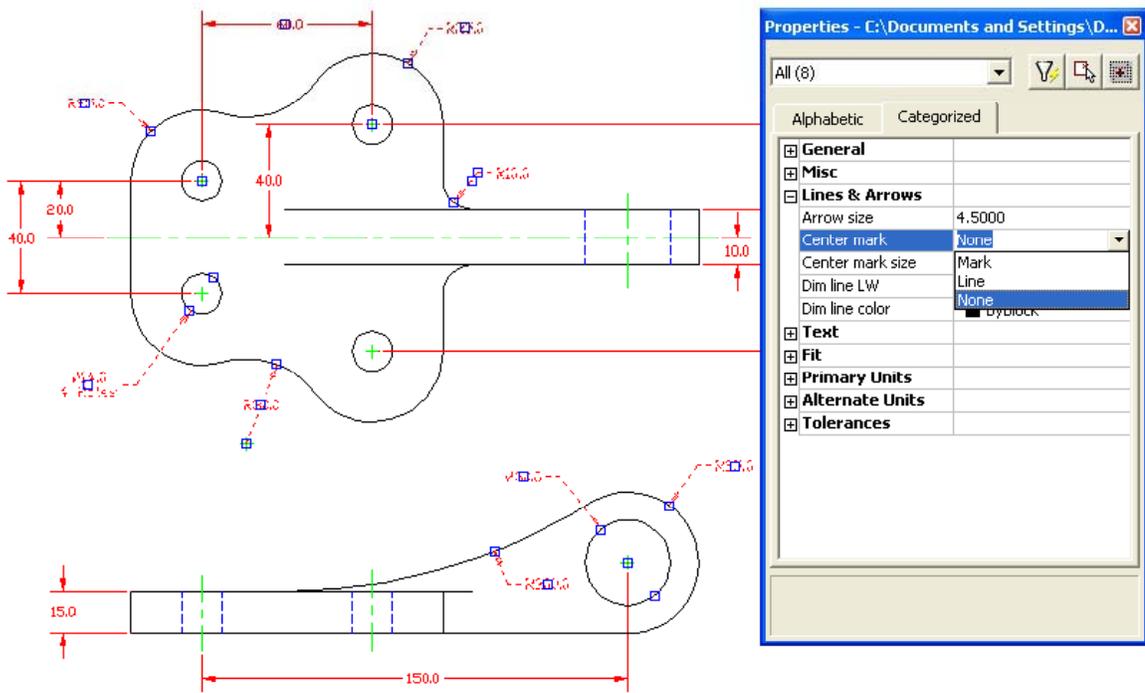


Figure 11.07 – Removing Dimension Center Marks with the Properties Tool

## Discussing the Difference Between Model and Paper Space

Now that you are 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 as shown Figure 11.08 displays that you are in Model Space.

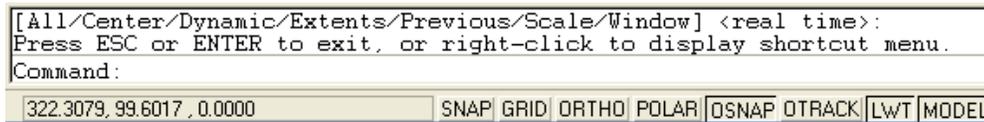


Figure 11.08 – The AutoCAD Command Line and Status Bar

When you begin the next chapter, “Placing the Bracket Problem in a B Size Border”, then you will be working in Paper Space or as you will notice the Status Bar list you in Layout 1. 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 for the showing the part or assembly and allowing you to dimension and note the 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. Next chapter, you will be able to plot a drawing in the “B” size border and release the signed print to a production facility.

