Autodesk®
Inventor® 2024

A Tutorial Introduction

Includes video instruction

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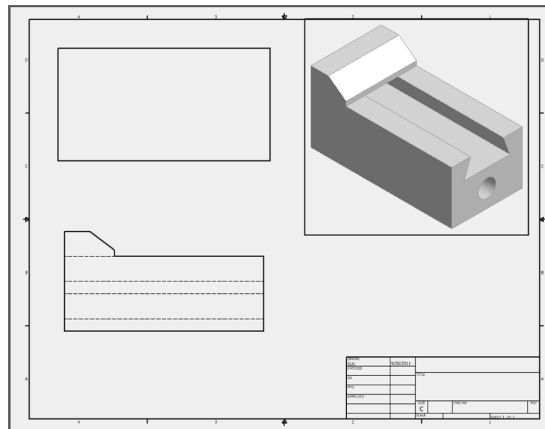
BARNES&NOBLE

CHAPTER 3

Learning to Create a Detail Drawing

Objectives:

1. Create a simple sketch using the Sketch Panel
2. Extrude a sketch into a solid using the 3D Model Panel
3. Create an Orthographic view using the Place Views/Drawing Views Panel
4. Extrude a sketch in the 3D Model Panel
5. Edit the appearance of a Solid Model using the Edit Views command

Chapter 3 includes instruction on how to design the parts shown.

1. Start Autodesk Inventor 2024 by referring to "Chapter 1 Getting Started."

2. After Autodesk Inventor 2024 is running, begin a new sketch.

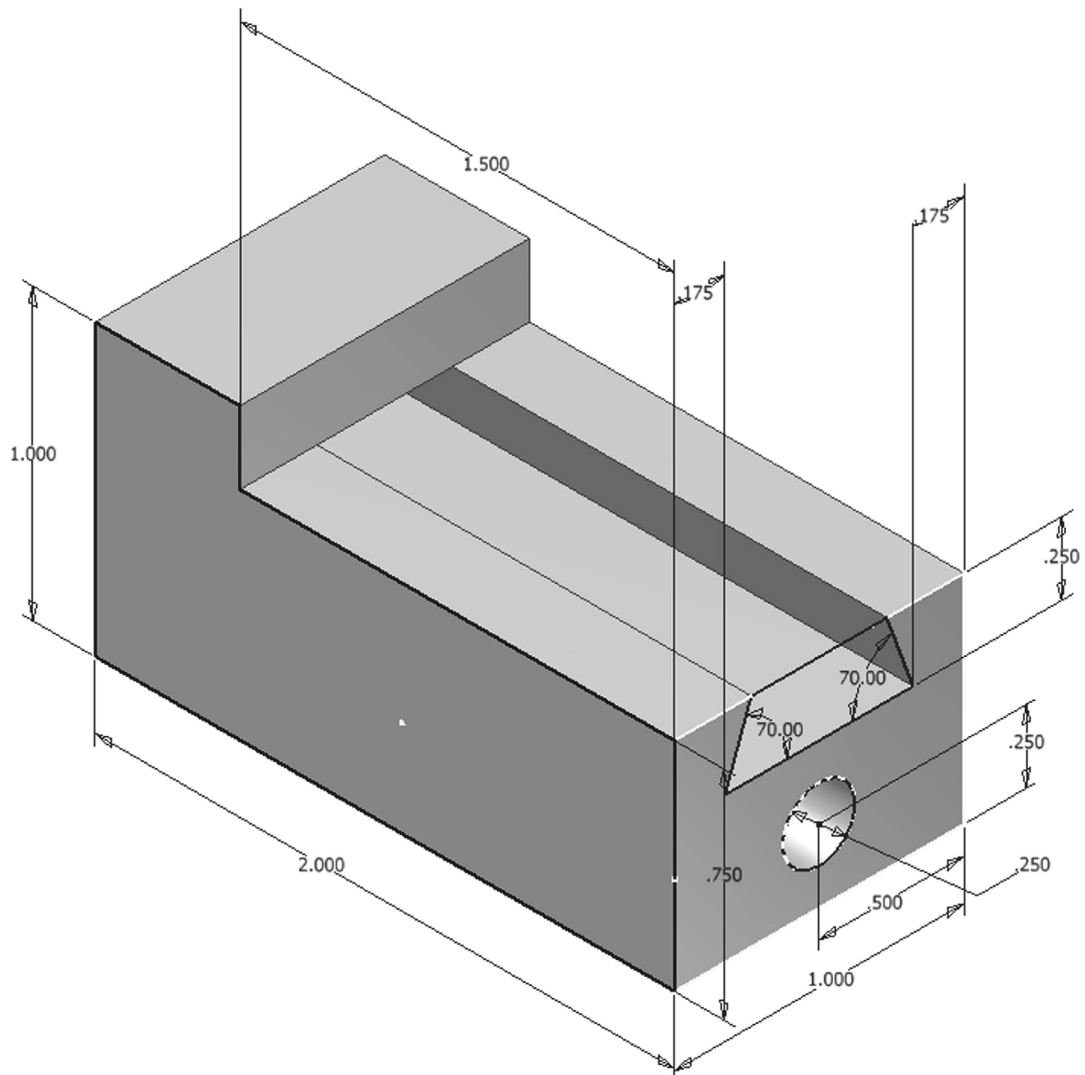3. Complete the drawing on the **XY Plane** shown in Figure 1.

Figure 1

4. Move the cursor to the right portion of the screen and left click on the 3D Model tab. Left click on the **Chamfer** icon as shown in Figure 2.
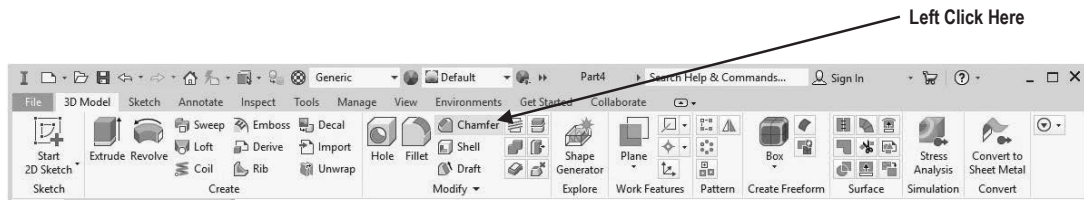
Left Click Here

Figure 2

5. After selecting **Chamfer**, the Chamfer dialog box will appear. Left click on the **Edge** icon. Left click on the "Two Distance Chamfer" icon as shown in Figure 3.
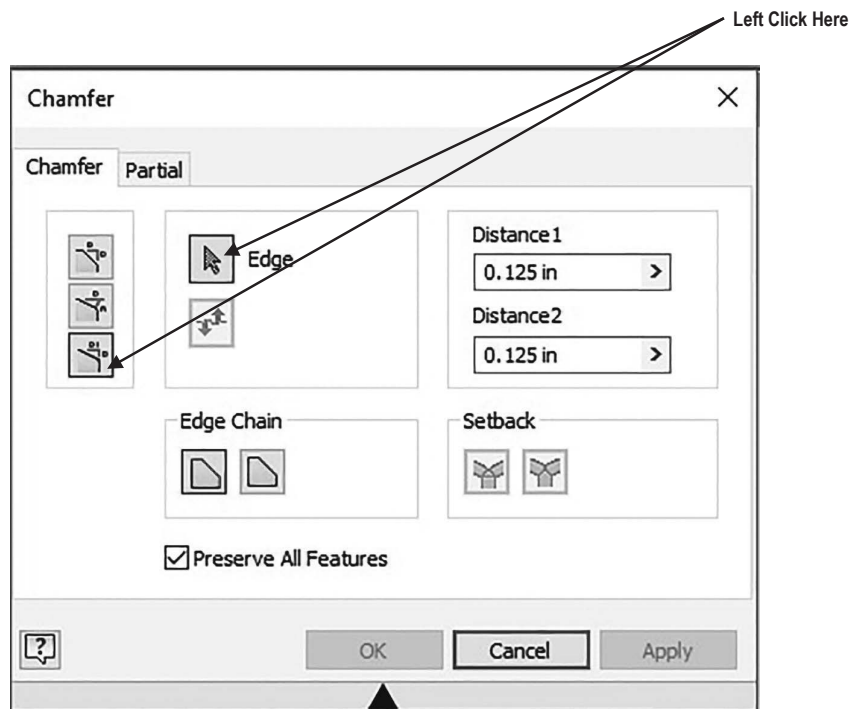
Left Click Here

Figure 3

6. Move the cursor to the front upper corner. A red line will appear as shown in Figure 4.
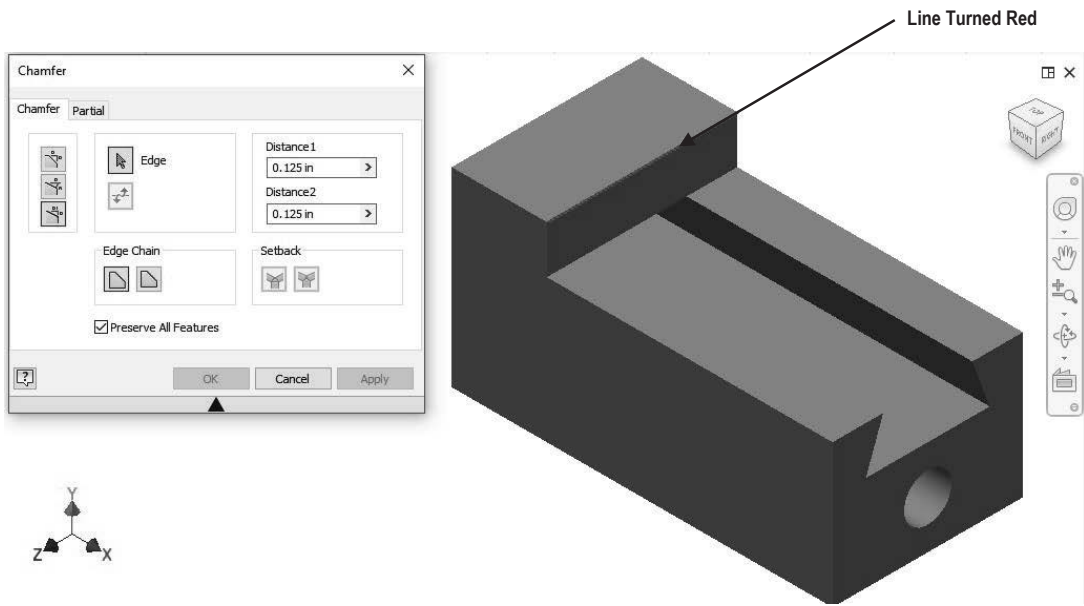

Figure 4

7. Inventor will provide a preview of the anticipated chamfer as shown in Figure 5.

Figure 5

8. Move the cursor to Distance 1 in the dialog box and highlight the text. Enter **.25** in the dialog box. Inventor will provide a preview of the chamfer as shown in Figure 6.
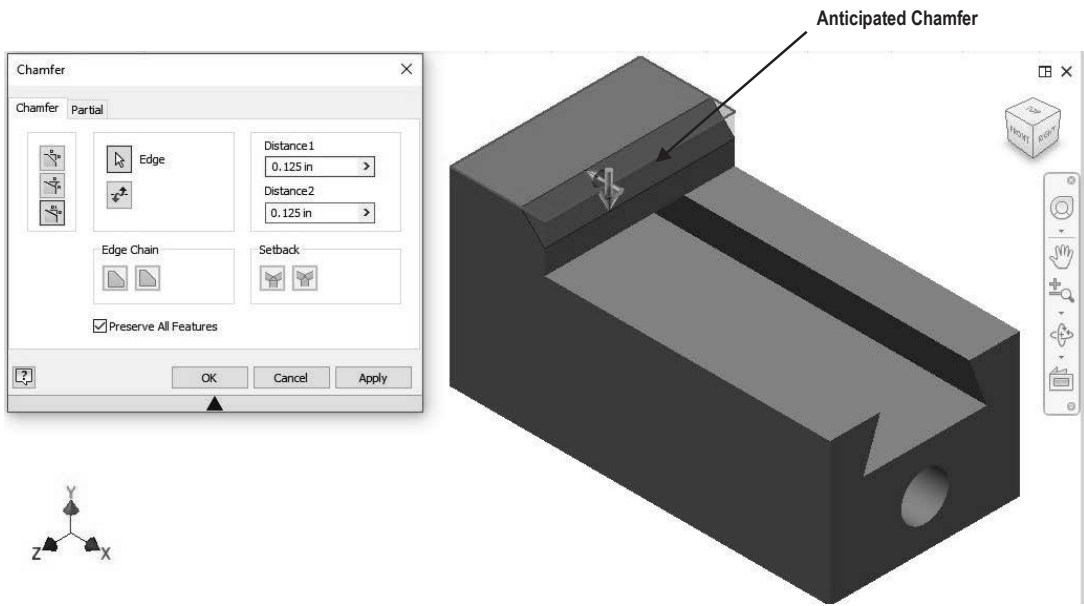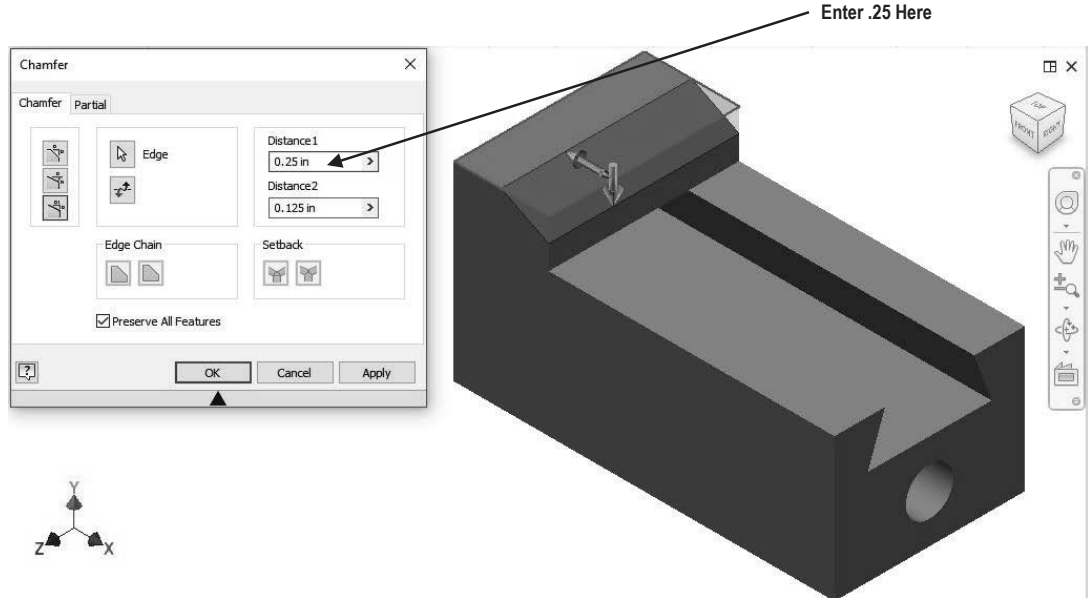
Enter .25 Here

Figure 6

9. Move the cursor to Distance 2 in the dialog box and highlight the text. Enter **.1875** in the dialog box. Inventor will provide a preview of the chamfer. To flip the direction of the chamfer simply left click on the Flip icon as shown in Figure 7.
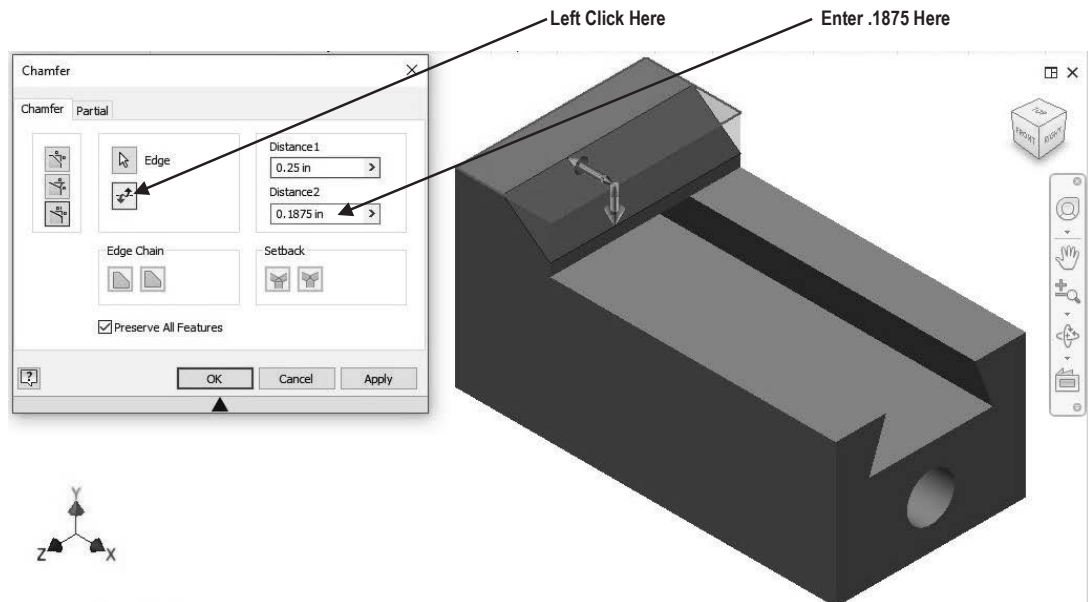
Left Click Here   **Enter .1875 Here**

Figure 7

10. Left click on **OK**. Your screen should look similar to Figure 8.
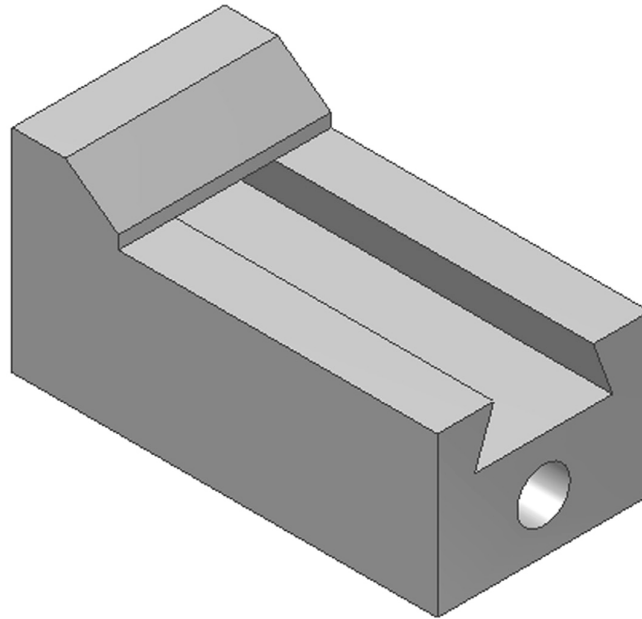

Figure 8

11. Save the part file for easy retrieval to be used in the following section. Do not close the part file.

Create an Orthographic view using the Drawing Views Panel

12. After the part file has been saved, move the cursor to the upper left portion of the screen and left click on the drop-down arrow to the right of the "New" icon. A drop-down menu will appear. Left click on **Drawing** as shown in Figure 9. Once the Drawing Views Panel appears, skip to step 14. If the Drawing Views Panel does not appear, move to step 13.
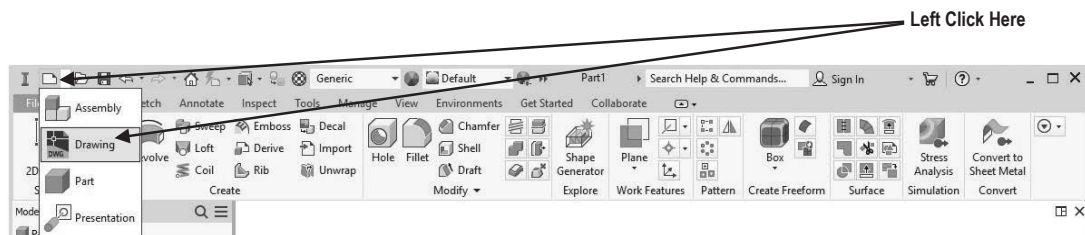
Left Click Here

Figure 9

13. The Create New File dialog box is shown below. Left click on the drop-down arrow next to en-US. Left click on the **English** folder. Left click on the **ANSI (in).idw.** Left click on **Create** as shown in Figure 10.
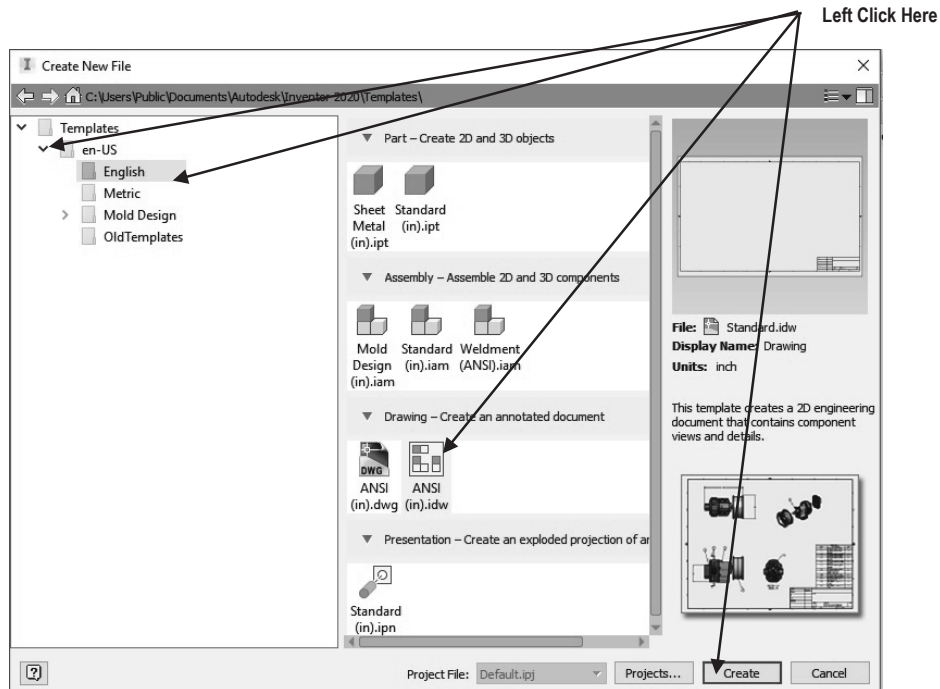

Figure 10

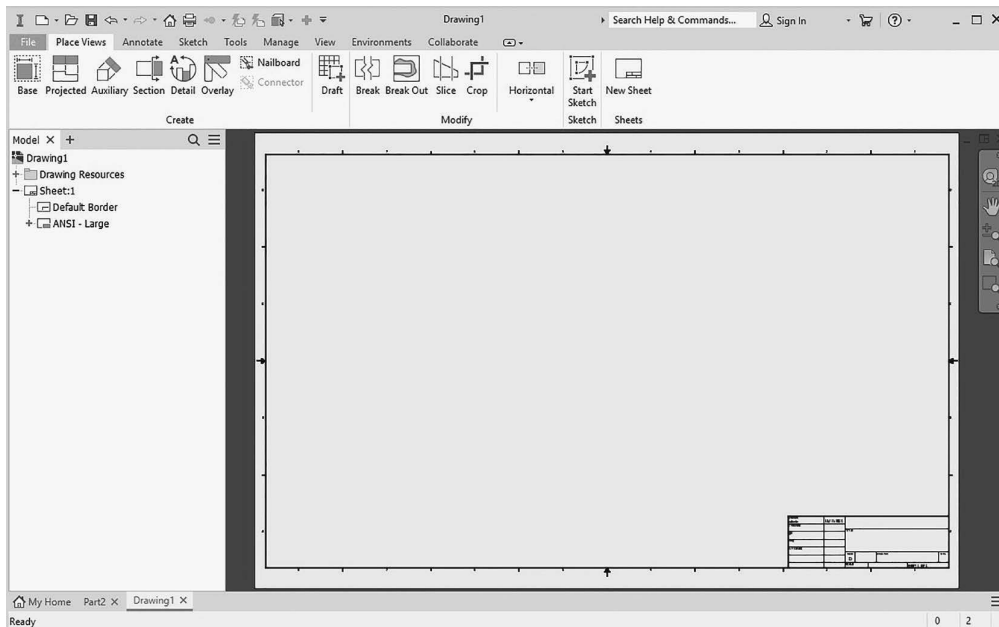**14.** Your screen should look similar to Figure 11.

Figure 11

15. Inventor is now in the Drawing Views Panel. Notice the commands at the top of the screen are now different.

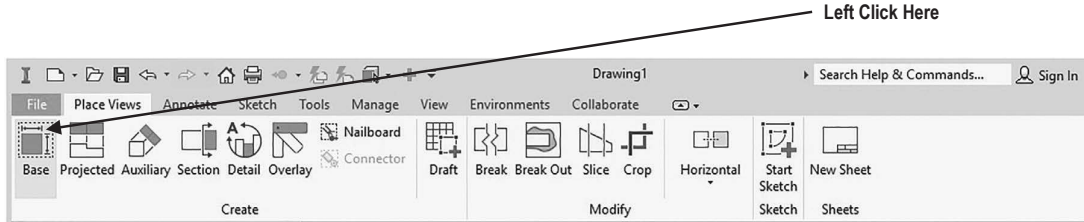**16.** Move the cursor to the upper left portion of the screen and left click on **Base** as shown in Figure 12.

Left Click Here

Figure 12

17. The drawing of the wedge block should appear attached to the cursor. If the part does not appear attached to the cursor, use the "Explore" icon to locate the part file as shown in Figure 13.
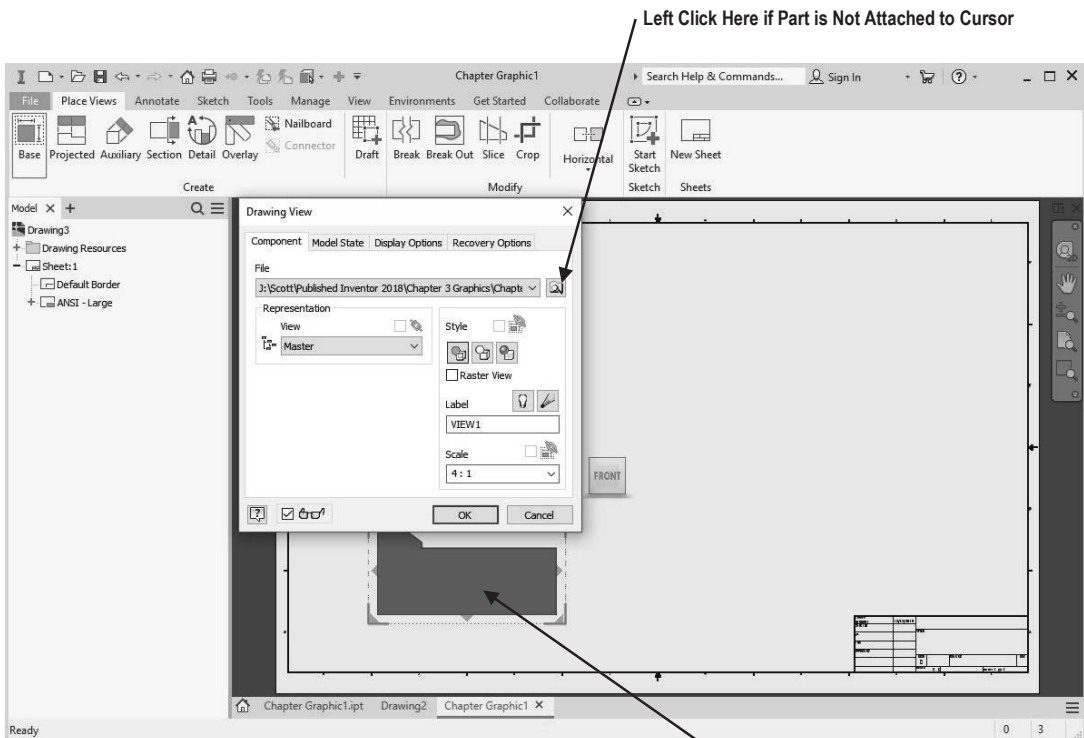
Left Click Here if Part is Not Attached to Cursor

Figure 13

Part Attached to Cursor

18. Different views can be selected for the front, top, and side views. Select the desired view from the view orientation cube as shown in Figure 14. To understand how the view orientation cube works, left click on the arrows that signify **Top, Left or Right** to have the top view, left view or right view as the front (base) view. Select the **Front** view for the base view as shown in Figure 14.
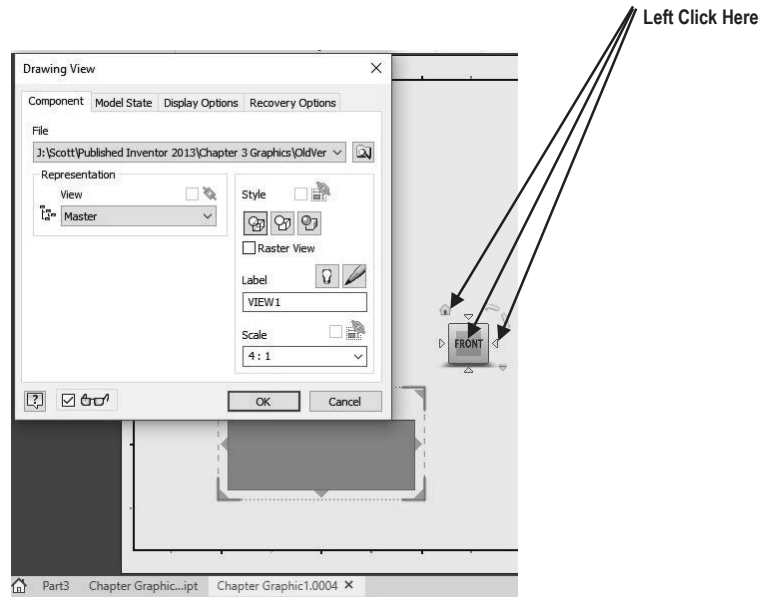
Left Click Here

Figure 14

19. Left click on the **Scale** drop-down box and set the drawing scale to **4:1** (if it is not already set to 4:1). The size of the wedge block will become larger as shown in Figure 15. Left click on **OK**.
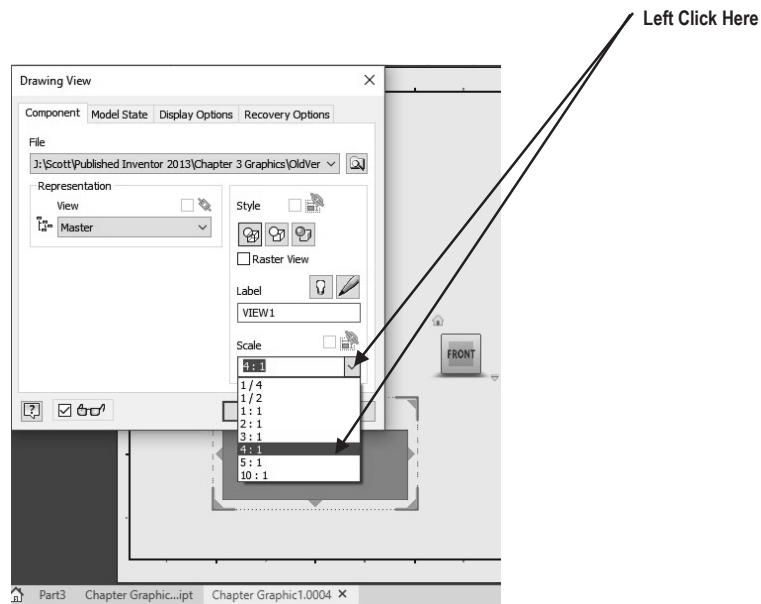
Left Click Here

Figure 15

20. Place the part just above the title block that is in the lower right corner of the screen and left click once. This will place the part as shown in Figure 16.
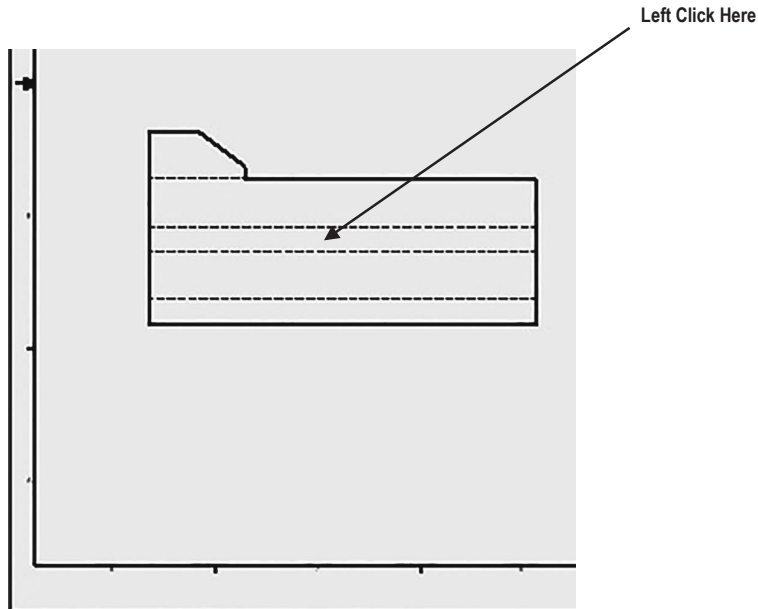
Left Click Here

Figure 16

21. If the part was inadvertently placed too low or too high, move the cursor over the dots that surround the part, left click (holding the mouse button down), and drag the part to the desired location.

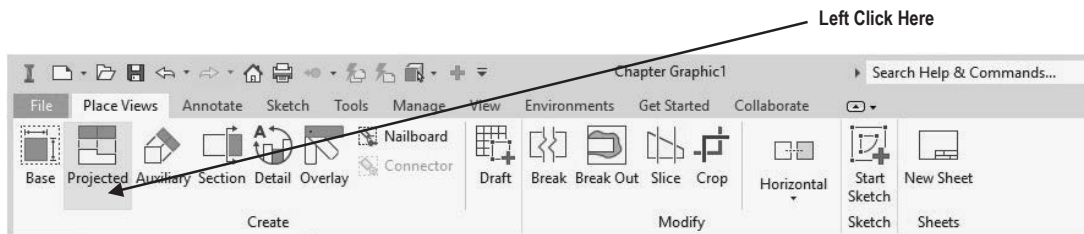**22.** Move the cursor to the upper left portion of the screen and left click on **Projected** as shown in Figure 17.

Left Click Here

Figure 17

23. The part will be attached to the cursor. Move the cursor upward and left click as shown in Figure 18.
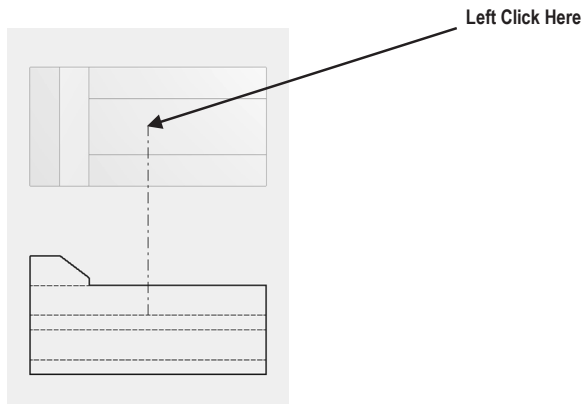
Left Click Here

Figure 18

24. Notice the black lines around the top view as shown in Figure 19. This indicates that the view has been placed.

Black Lines
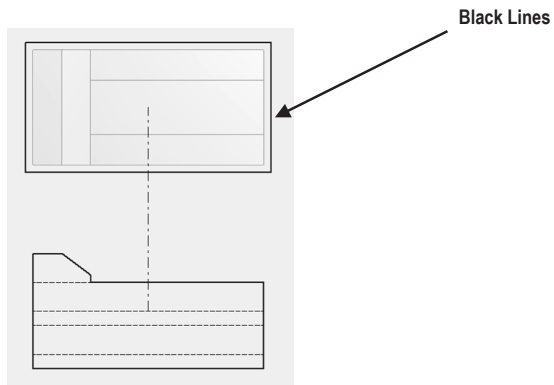
Figure 19

25. Move the cursor over to the upper right corner of the page and left click once as shown in Figure 20.

Left Click Here

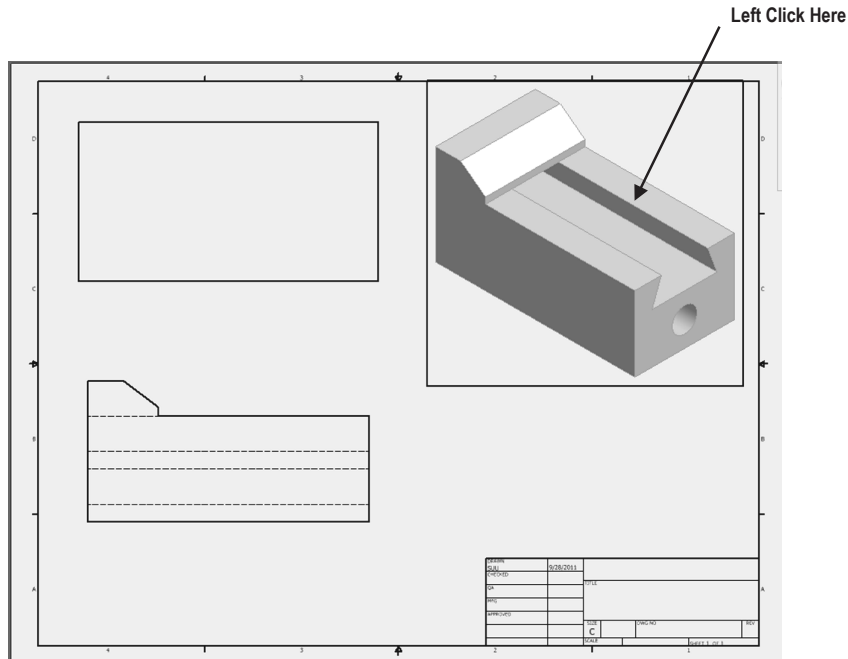**Figure 20**

26. Move the cursor down to where the side view will be located and left click once as shown in Figure 21.
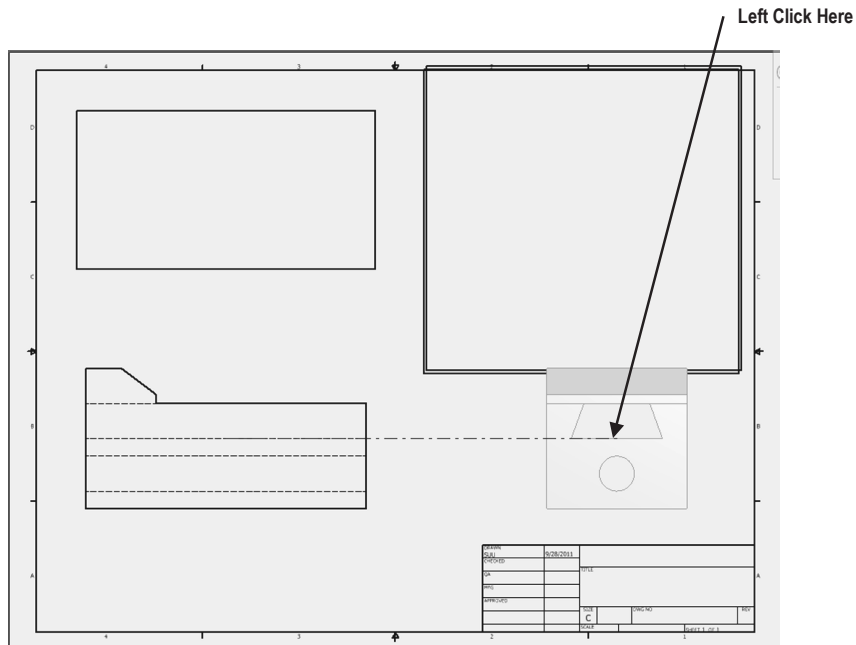
Left Click Here

Figure 21

27. Right click on the last view created (side view). A pop-up menu will appear. Left click on **Create** as shown in Figure 22.
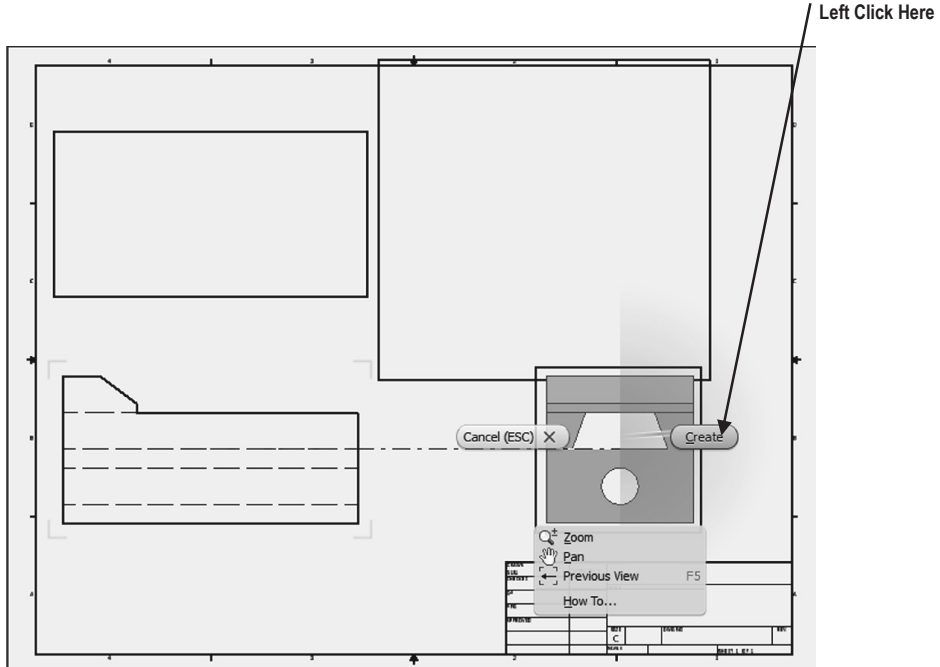
Left Click Here

Figure 22

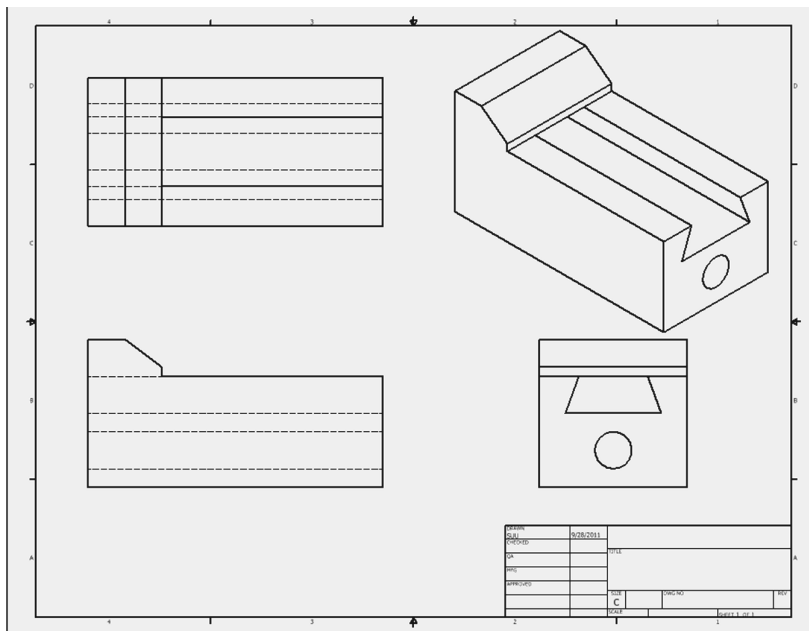**28.** Your screen should look similar to Figure 23.

Figure 23

Create a Solid Model using the Edit Views command

29. Move the cursor over the isometric view in the upper right corner of the drawing. Red dots will appear as shown in Figure 24.
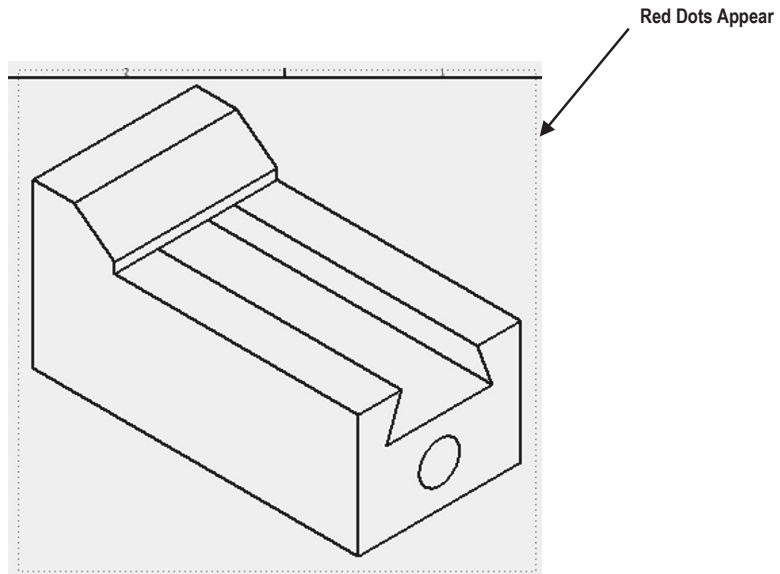
Red Dots Appear

Figure 24

30. After the red dots appear, right click once. A pop-up menu will appear. Left click on **Edit View** as shown in Figure 25.
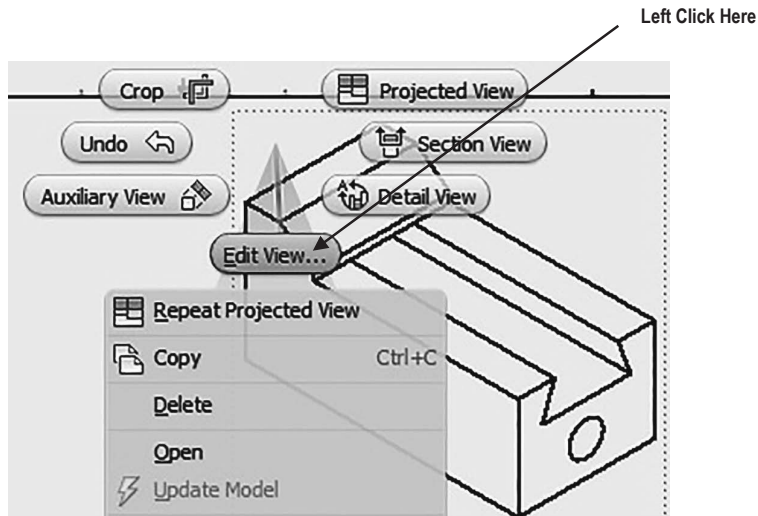
Left Click Here

Figure 25

31. The Drawing View dialog box will appear. Left click on the "blue sphere." Left click on **OK** as shown in Figure 26.

Figure 26
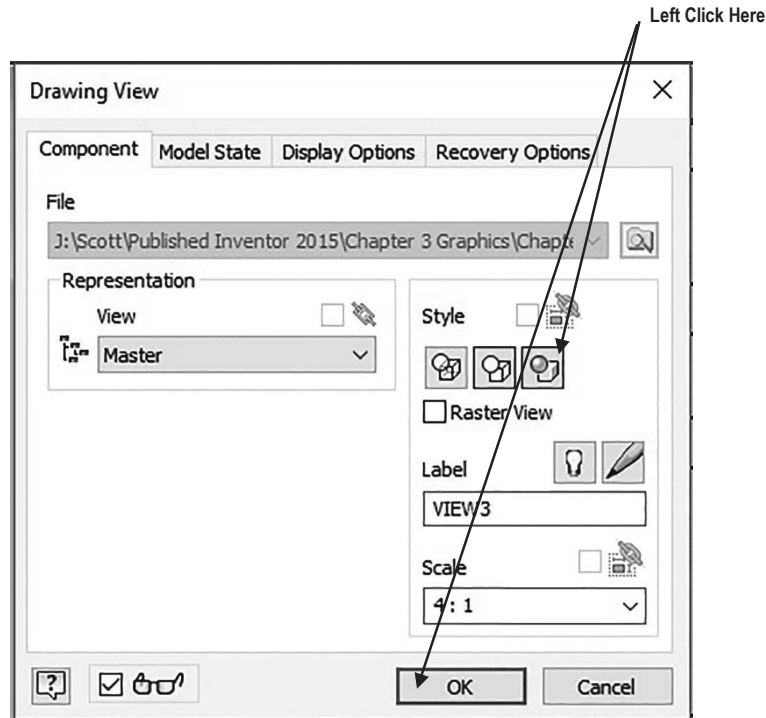
32. The isometric view will become a miniature solid model as shown in Figure 27.
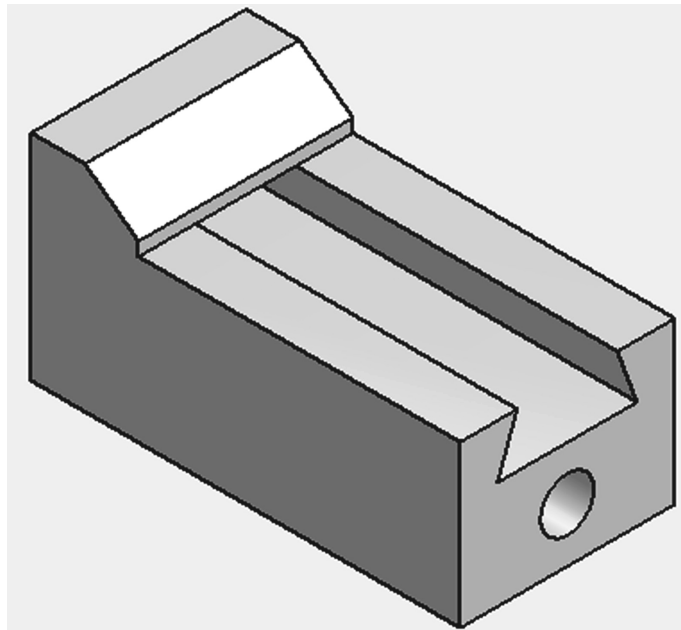

Figure 27

33. Your screen should look similar to Figure 28.
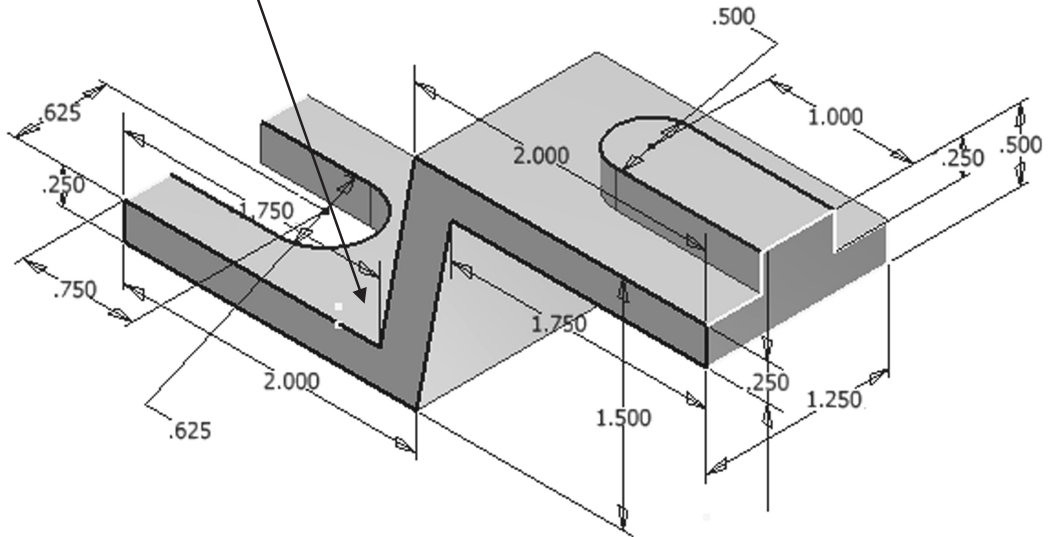

Figure 28
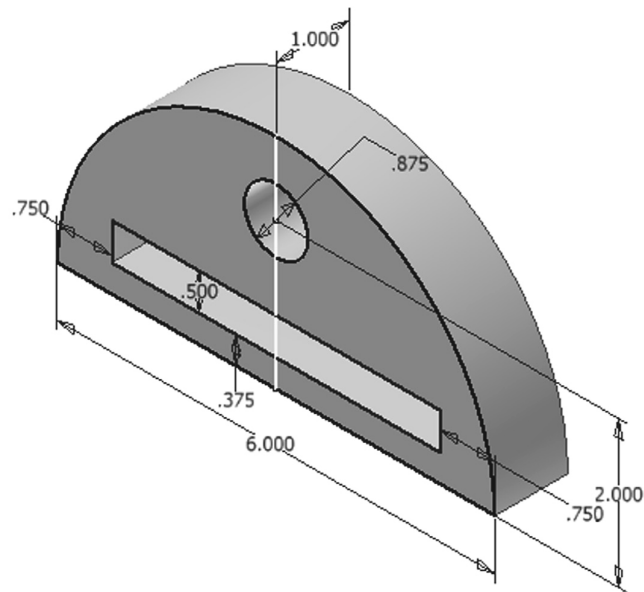
Chapter Problems

Create 3 view/multi view drawings of the following parts.

Problem 3-1   Hint: Dimension the angle between the lower and upper section to 105 degrees.

.625
.250
.750
2.000
.625
1.750
.500
2.000
1.000
.250
.500
1.750
1.500
.250
1.250

Problem 3-2

1.000
.875
.750
.500
.375
6.000
2.000
.750

Problem 3-3

Hint: Extrude to .25 inches

1.000
.250
1.000
3.000
1.500
1.500
.750
.250 .500
.250
.500
2.000
4.000
2.000
2.000
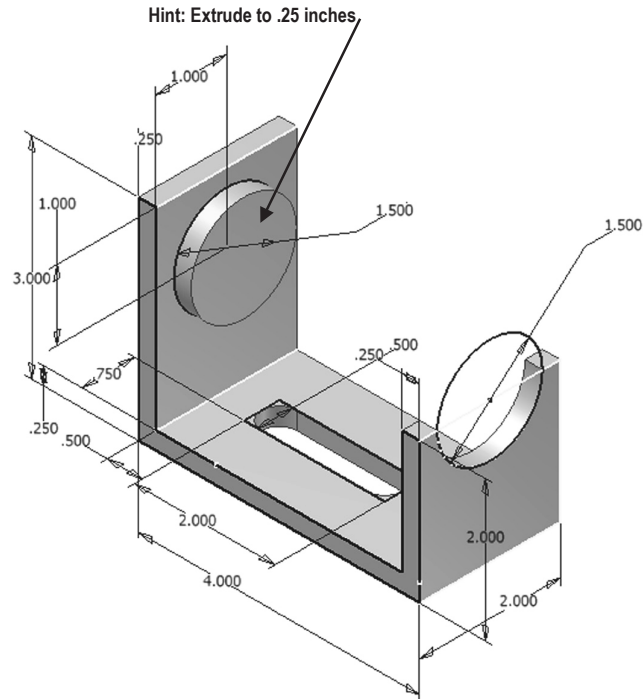
Problem 3-4

Hint: Use the 2 directional Extrude-Cut command to create the 1.625 Diameter Hole

1.000
.375 .375
.375
.500 .750
.500
1.625
.375
3.000
1.500
.818
1.000
95.00
4.000
.875
.500
2.000
.750 .500 2.000

3-18

Problem 3-5

Extrude .50 inches
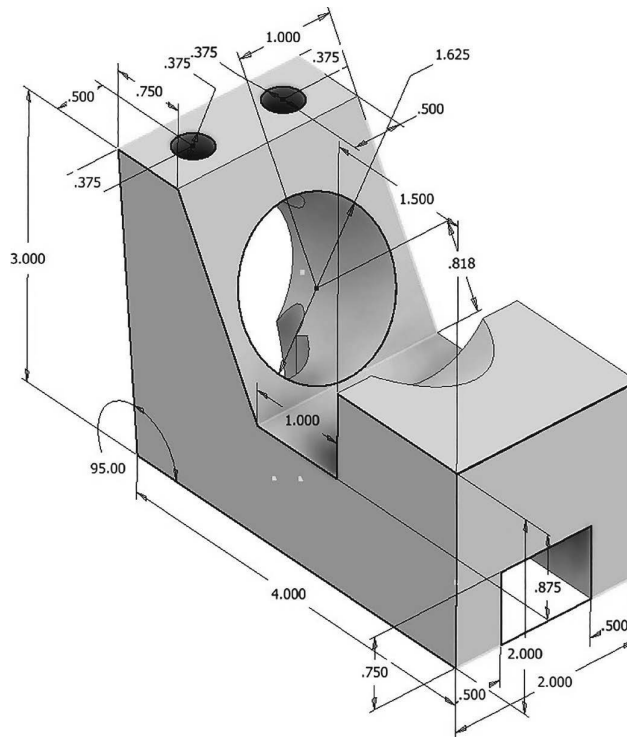Center to Center
Distance is 1.5 Inches

.875

.750   1.500   .750

.500

4.000

.750

1.500

.500

1.500

2.500

.500

.500

1.500   1.500

2.500   .500

1.875

3.000

1.500

1.375   .500

2.500

1.500   .500   1.375

1.500

3.000

1.500

2.500

.375

.688

Problem 3-6

1.500   1.375

1.000

.500

3.000

2.000

.500

85.00   .625

1.500   1.750

.500

1.500   .500

3.000   4.000

Problem 3-7

Extrude to .25 off of Base
Extrude to .50 off of Base
Extrude to .75 off of Base

.500

.500

.688

.500

2.500

.275

.375

.625

.275

.250

1.000

.438

1.100

Problem 3-8
1 inch Fillets

Hint: Start by drawing the sketch first.   Use the Chamfer command

.750

.500

1.000

1.500

2.000

.375

.500 .500

1.000

.750

.375

1.500

.500

4.000

1.500

.375

2.000