

Autodesk® Inventor® 2019

A Tutorial Introduction
Includes video instruction

New
Includes chapters on
Weldments & Content Center



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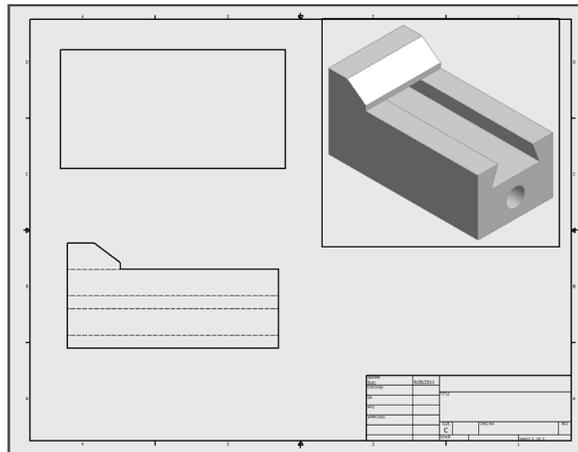
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CHAPTER 3

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 Model/Part Features 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 2019 by referring to “Chapter 1 Getting Started.”
2. After Autodesk Inventor 2019 is running, begin a new sketch.
3. Complete the drawing shown in Figure 1.

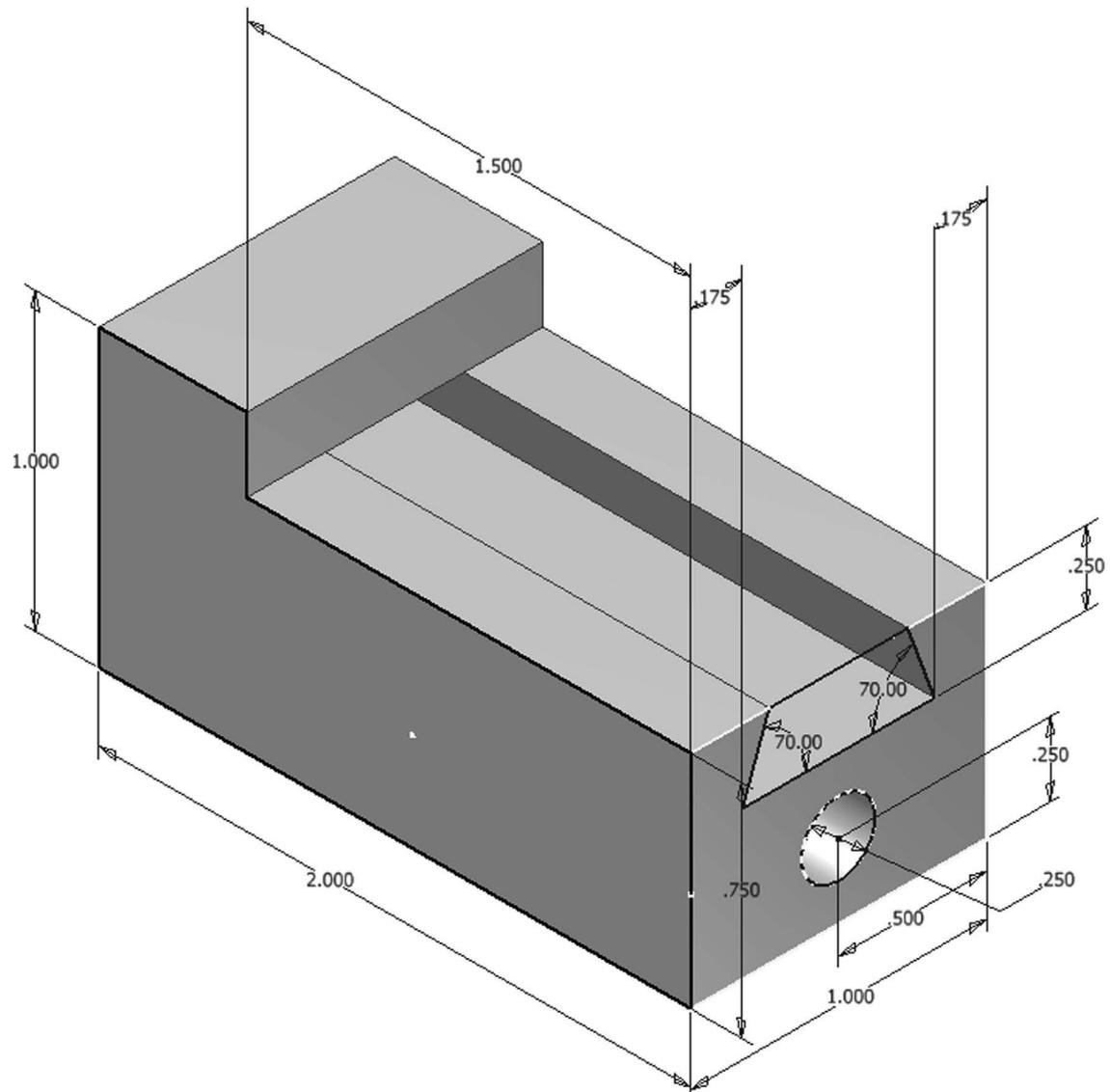


Figure 1

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.

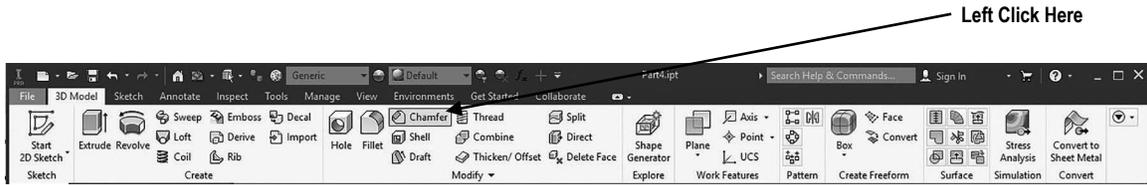


Figure 2

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.

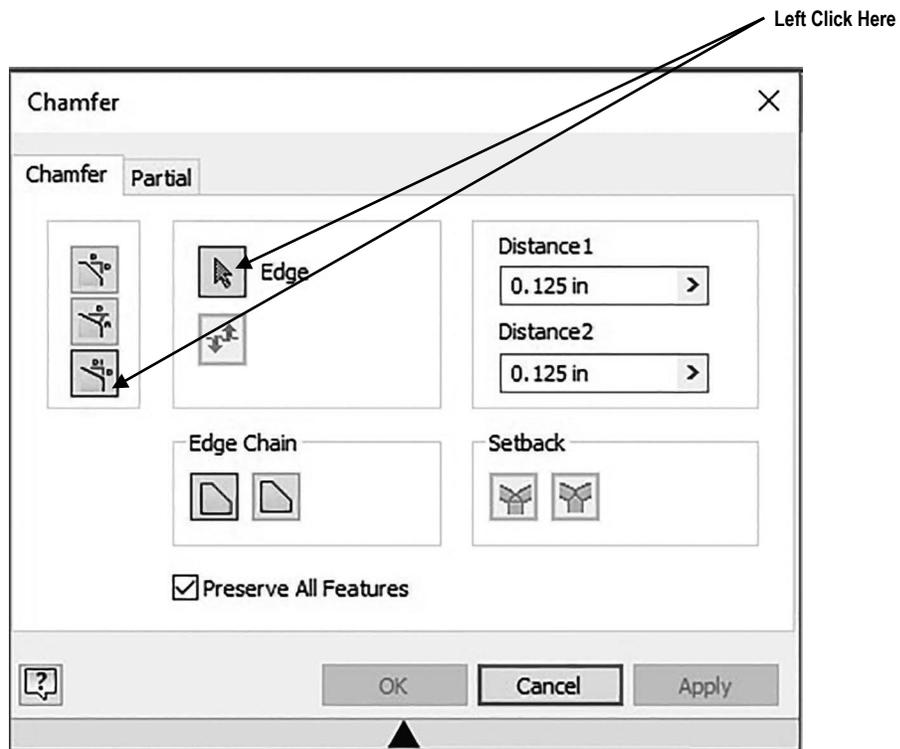


Figure 3

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

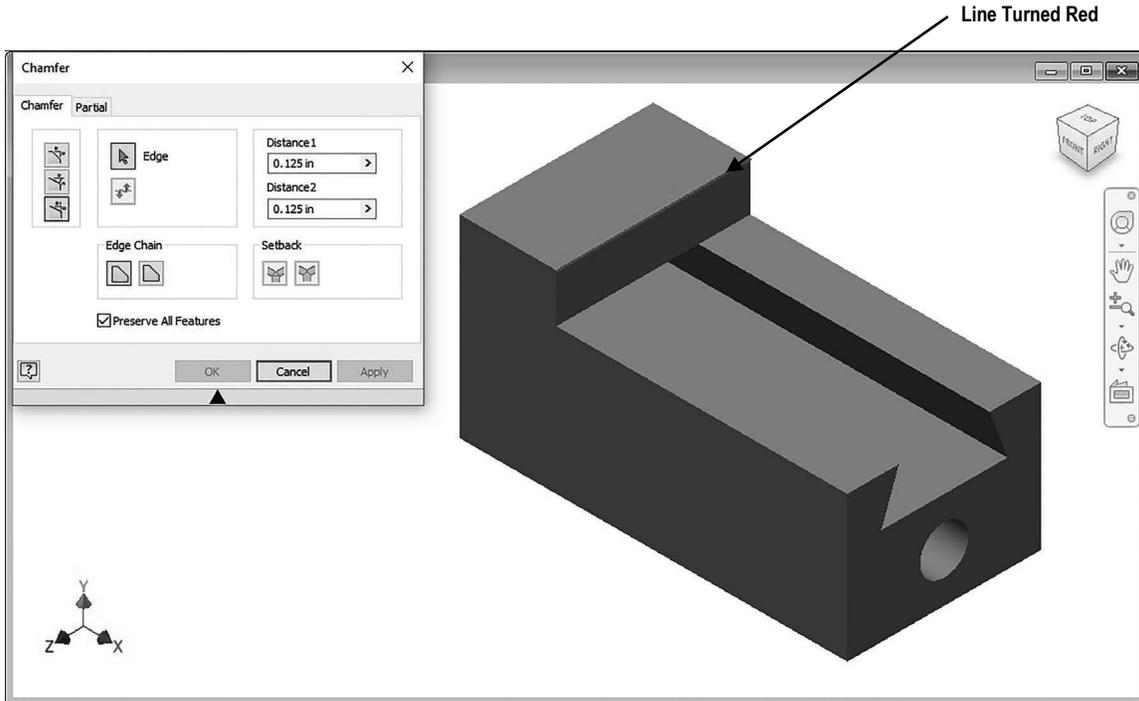


Figure 4

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

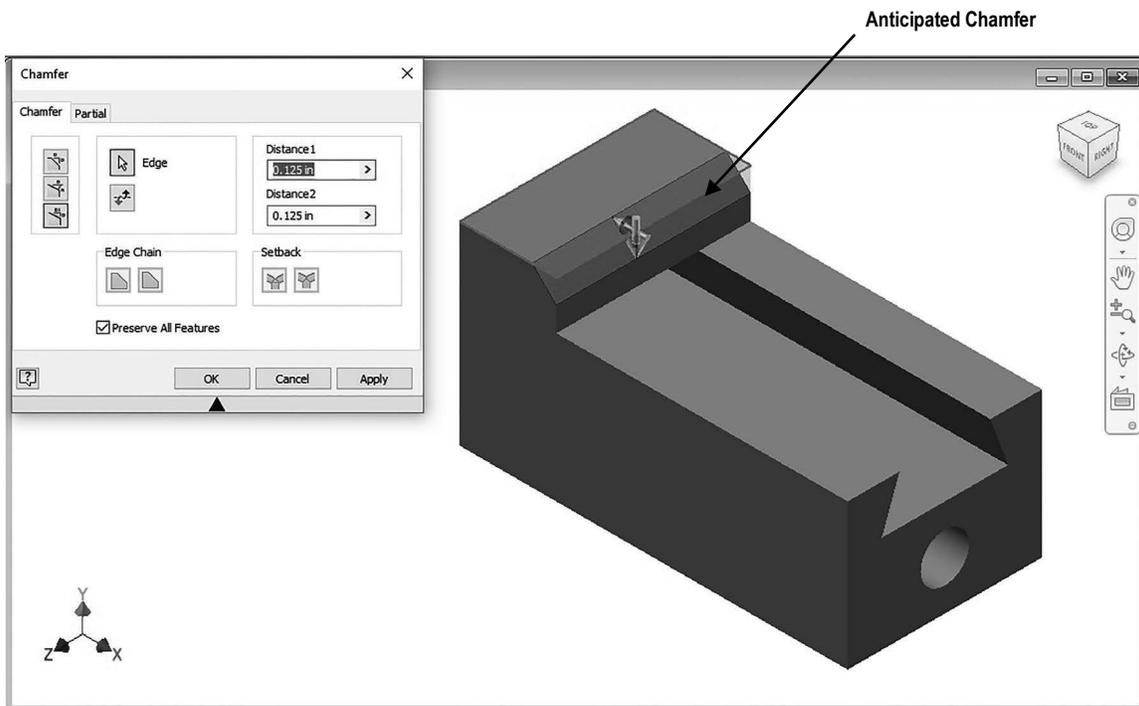


Figure 5

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.

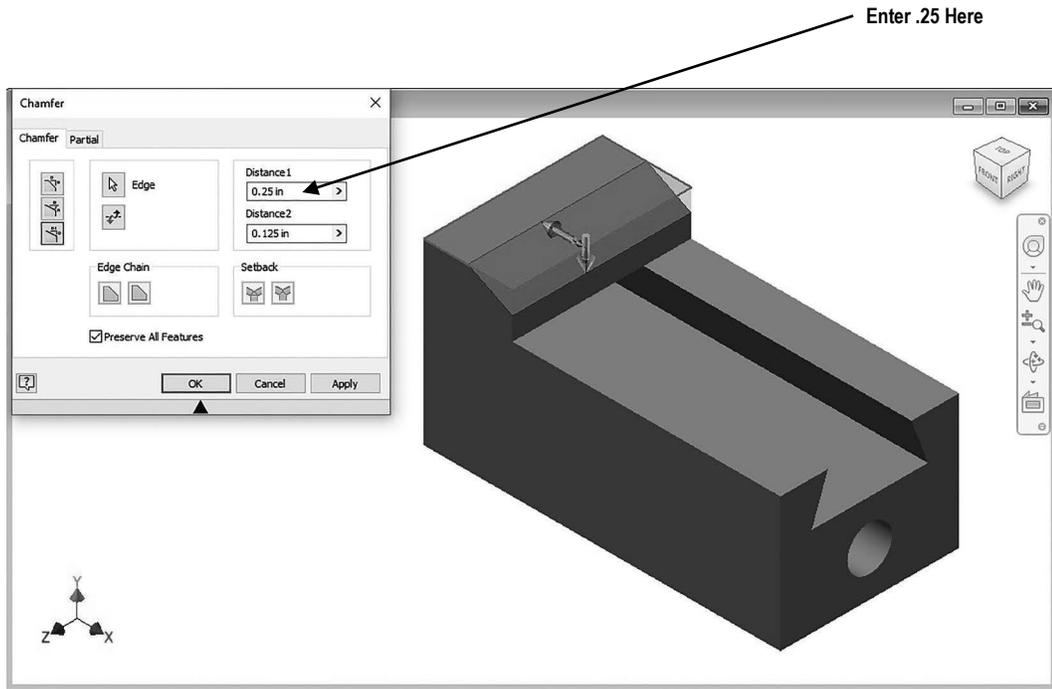


Figure 6

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

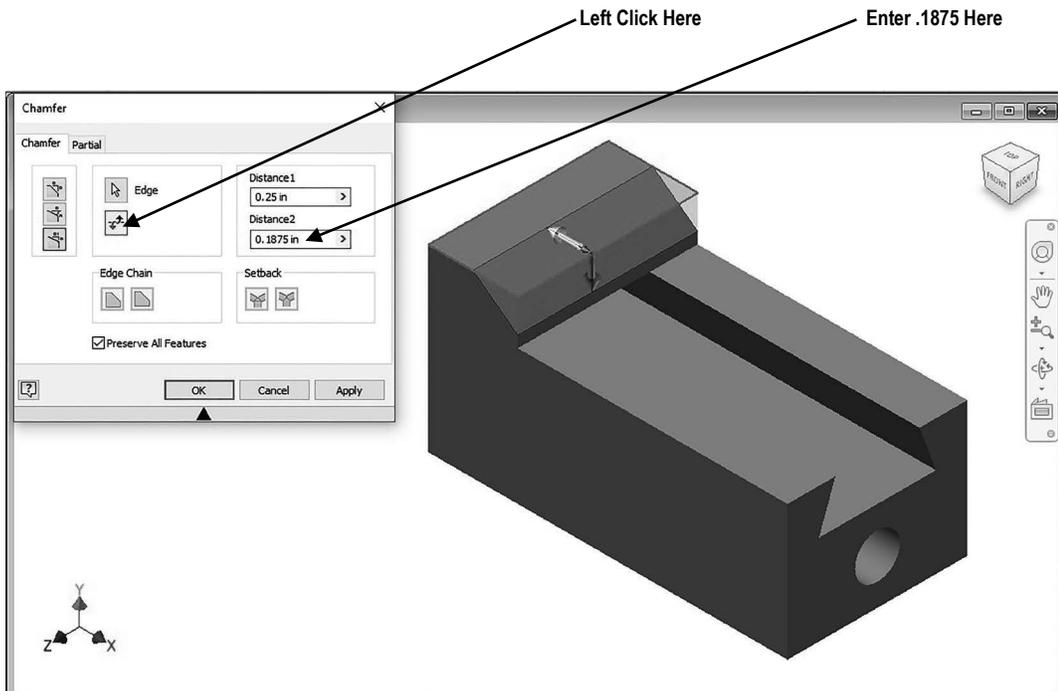


Figure 7

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

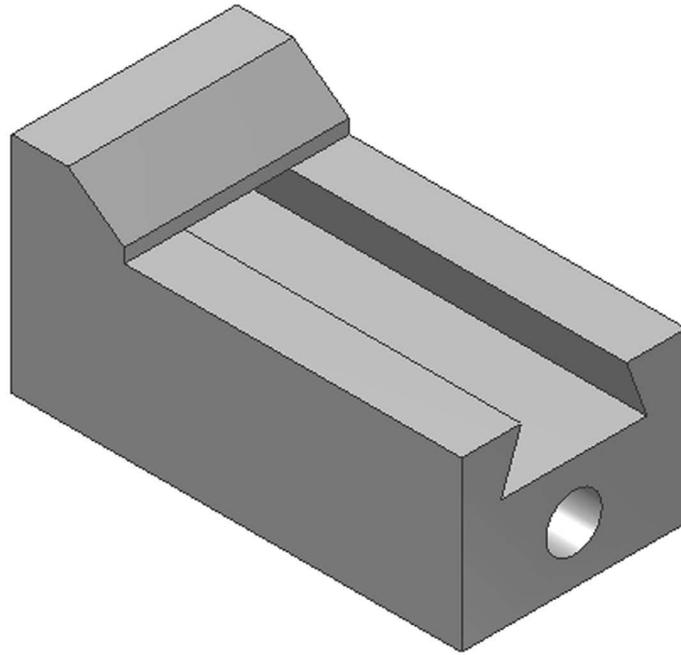


Figure 8

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

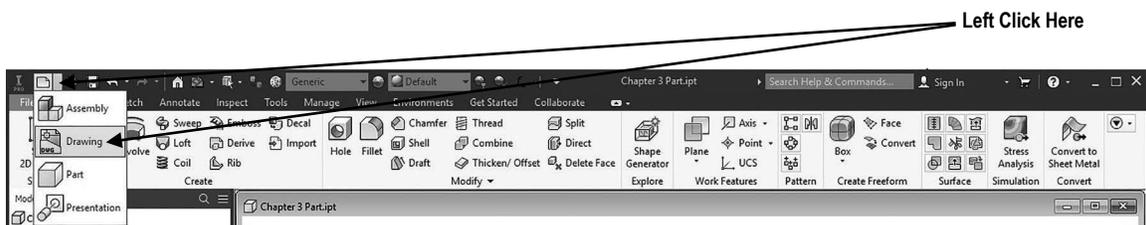


Figure 9

Create an Orthographic view using the Drawing Views Panel

13. The Create New File dialog box will appear. Left click on the **English** folder. Left click on the **ANSI (in).idw**. Left click on **Create** as shown in Figure 10.

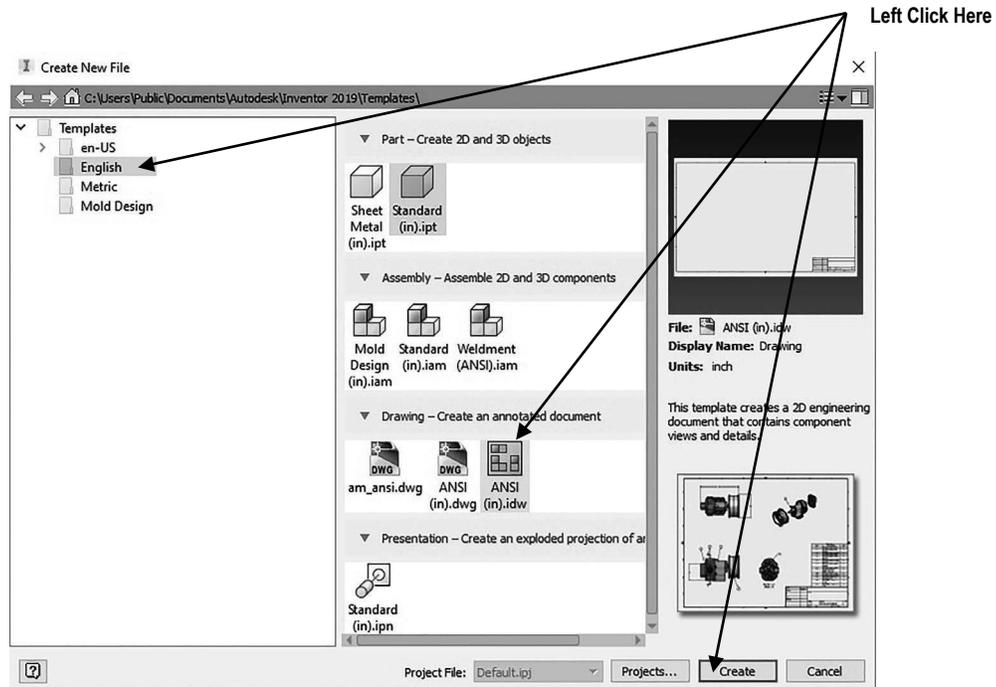


Figure 10

14. Your screen should look similar to Figure 11.

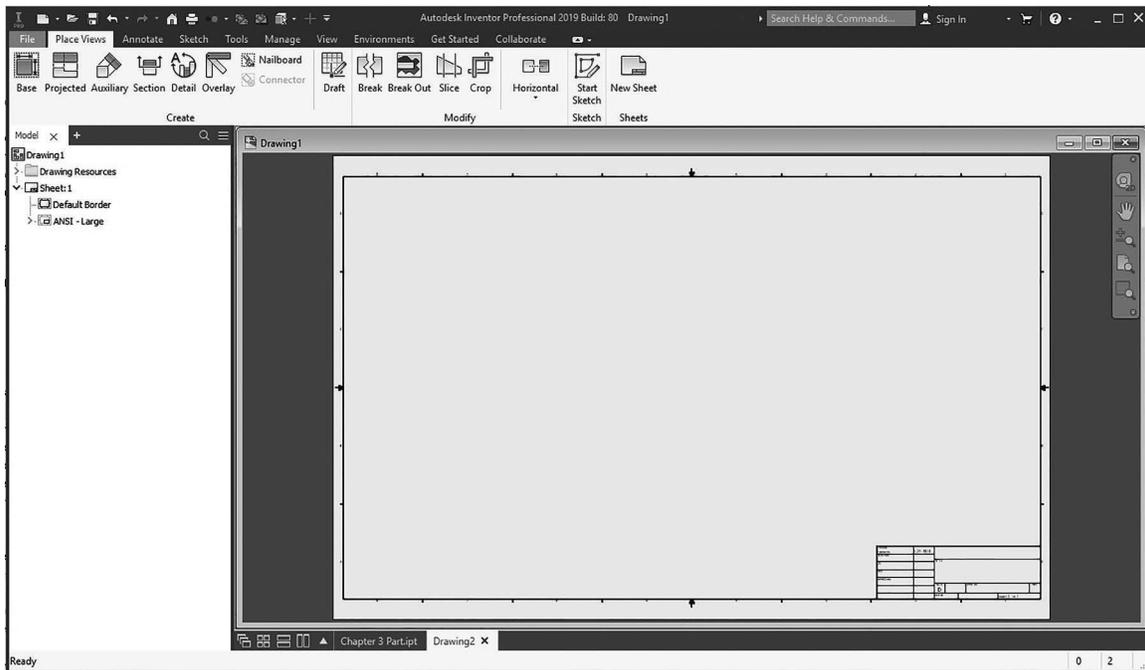


Figure 11

15. Inventor is now in the Place Views Panel. Notice the commands at the top of the screen are now different. The width of the screen has been reduced to add instructional clarification.
16. Move the cursor to the upper left portion of the screen and left click on **Base** as shown in Figure 12.

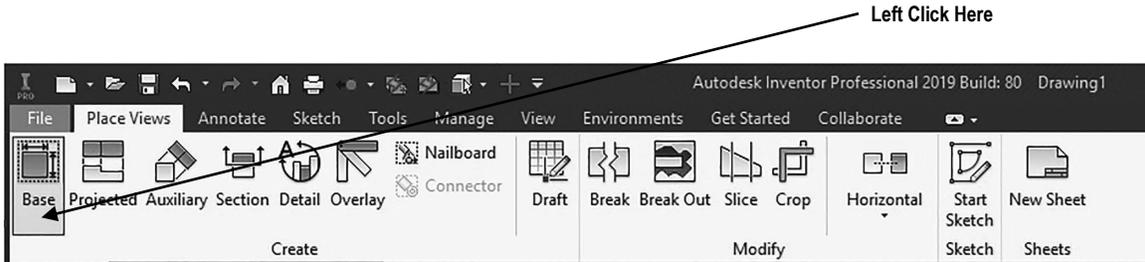


Figure 12

17. The drawing of the wedge block should appear attached to the cursor. Move the cursor around to verify it is attached. If the part does not appear attached to the cursor, use the "Explore" icon to locate the part file as shown in Figure 13.

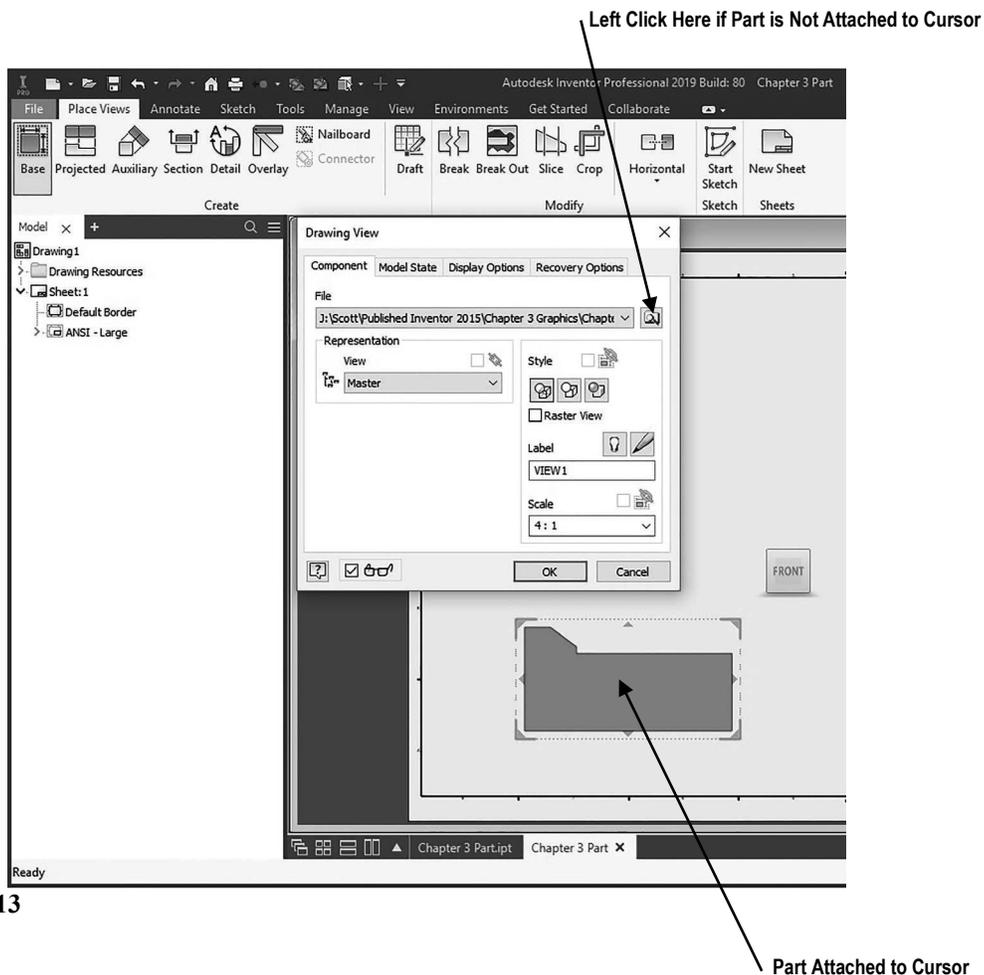


Figure 13

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.

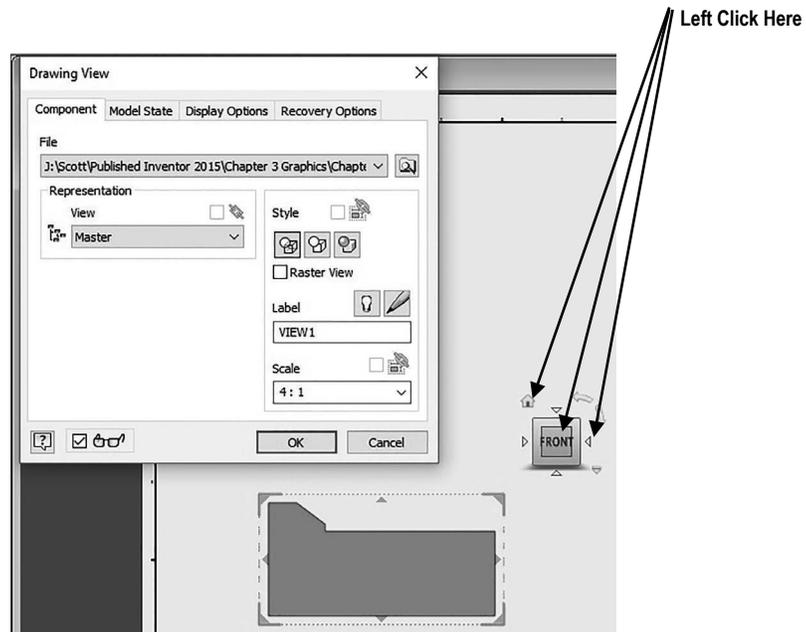


Figure 14

19. Left click on the **Scale** drop down box and set the drawing scale to **4:1**. The size of the wedge block will become larger as shown in Figure 15. Left click on **OK**.

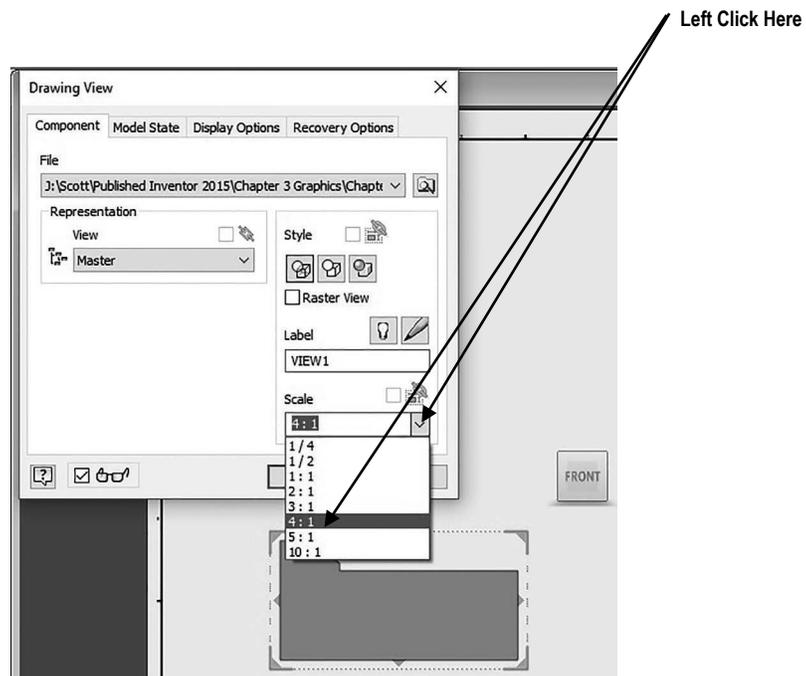


Figure 15

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.

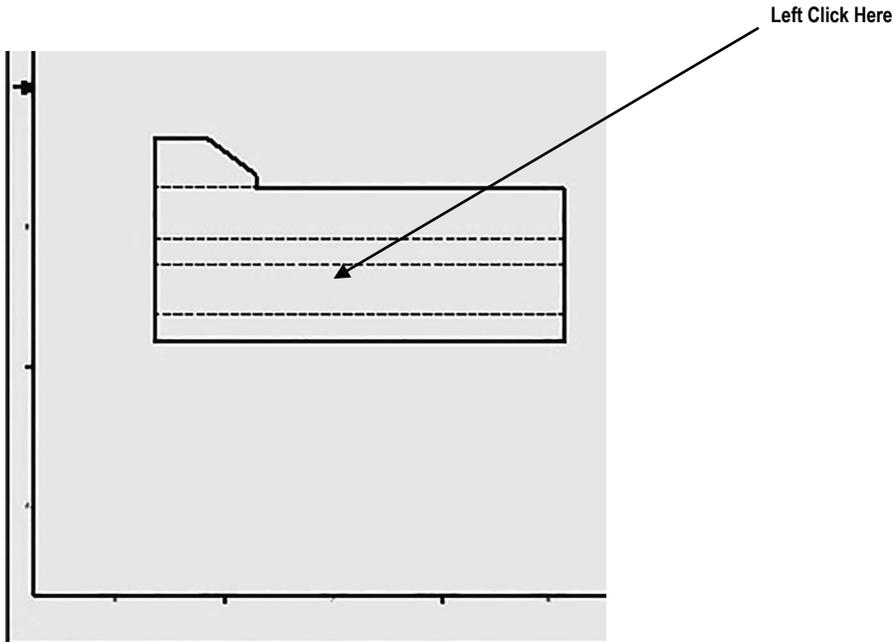


Figure 16

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.

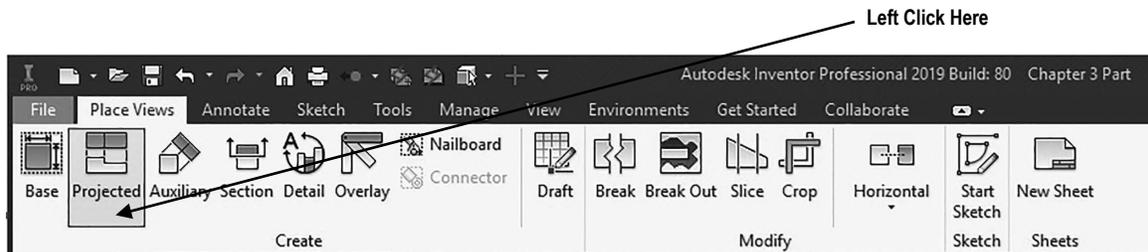


Figure 17

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

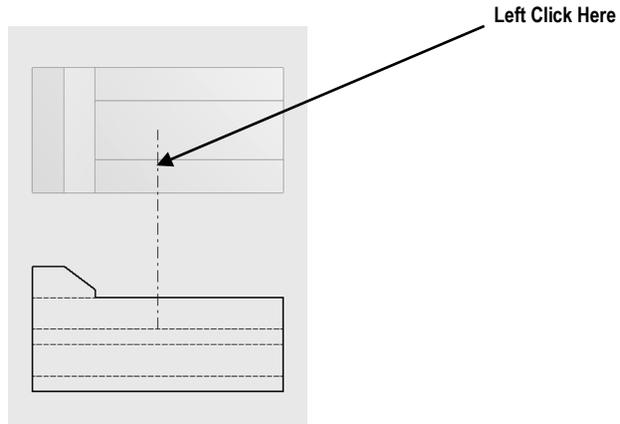


Figure 18

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

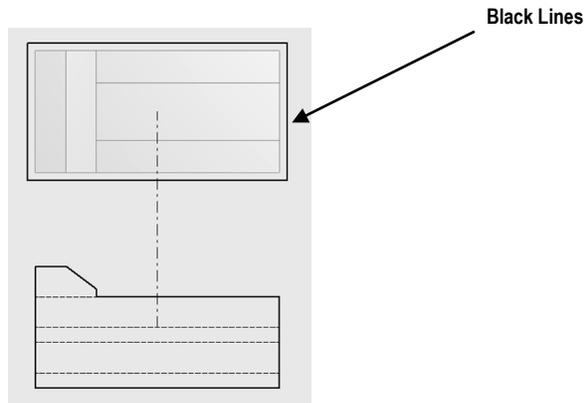


Figure 19

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

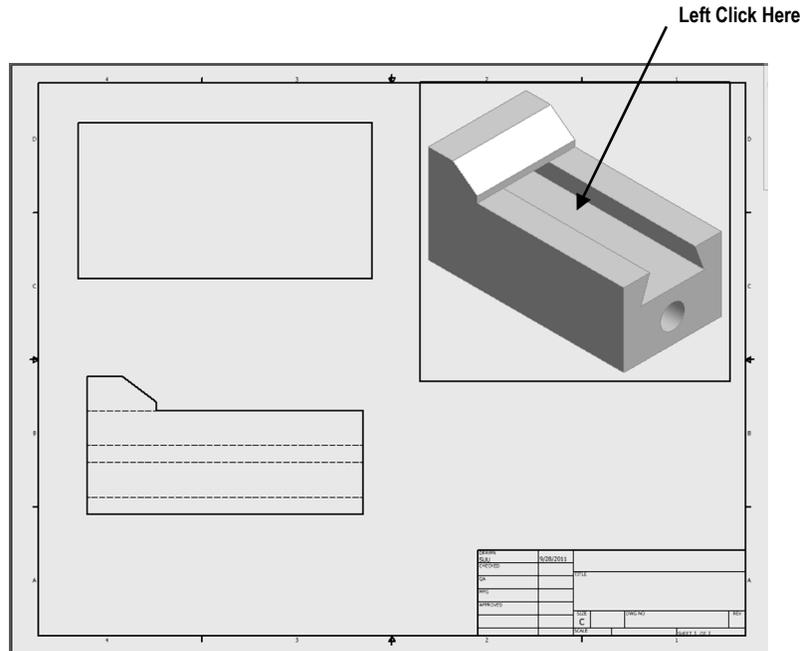


Figure 20

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

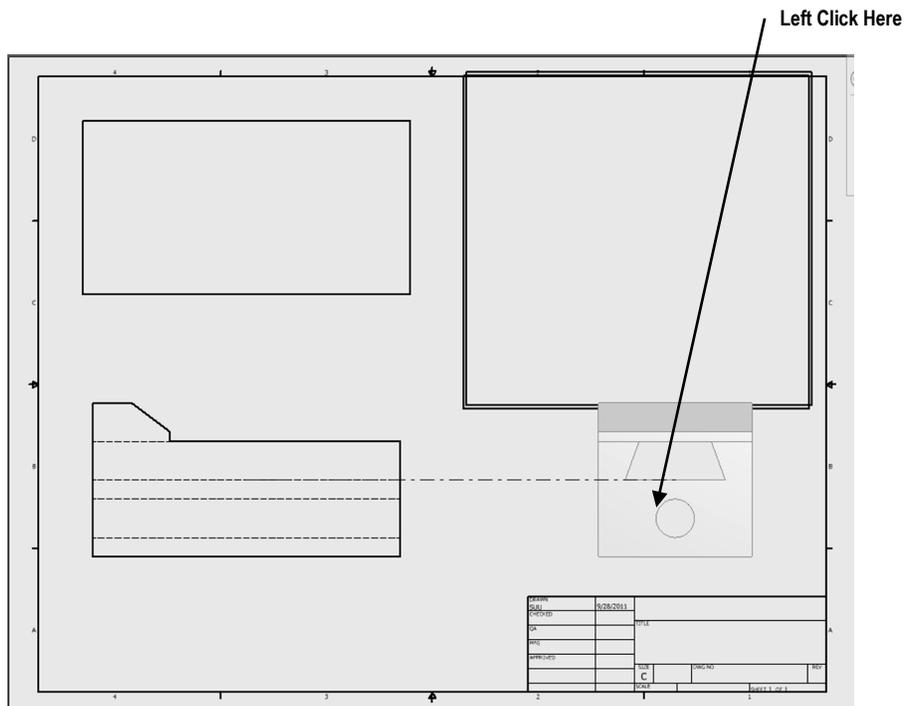


Figure 21

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.

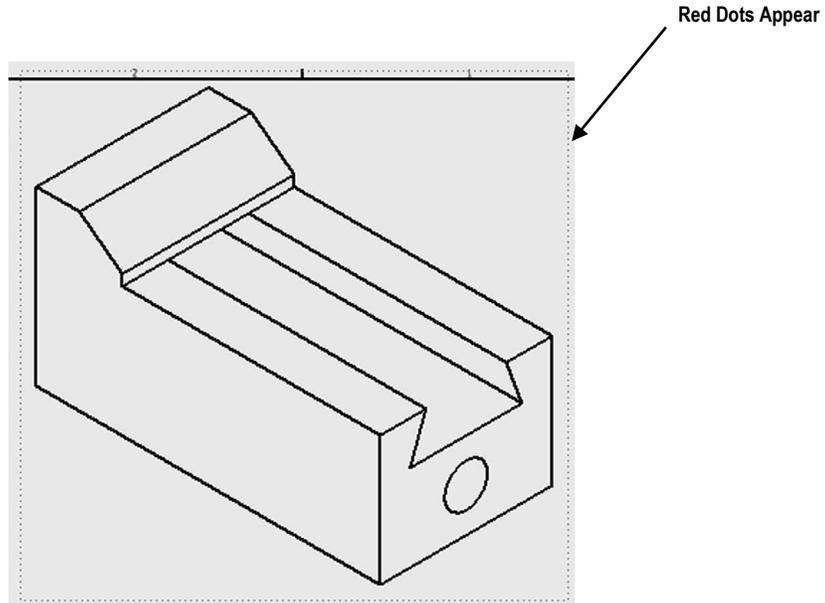


Figure 24

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.

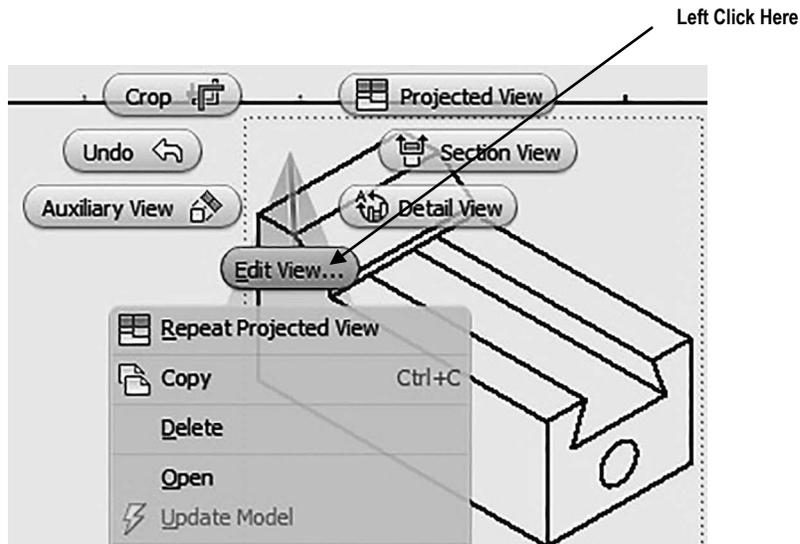


Figure 25

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

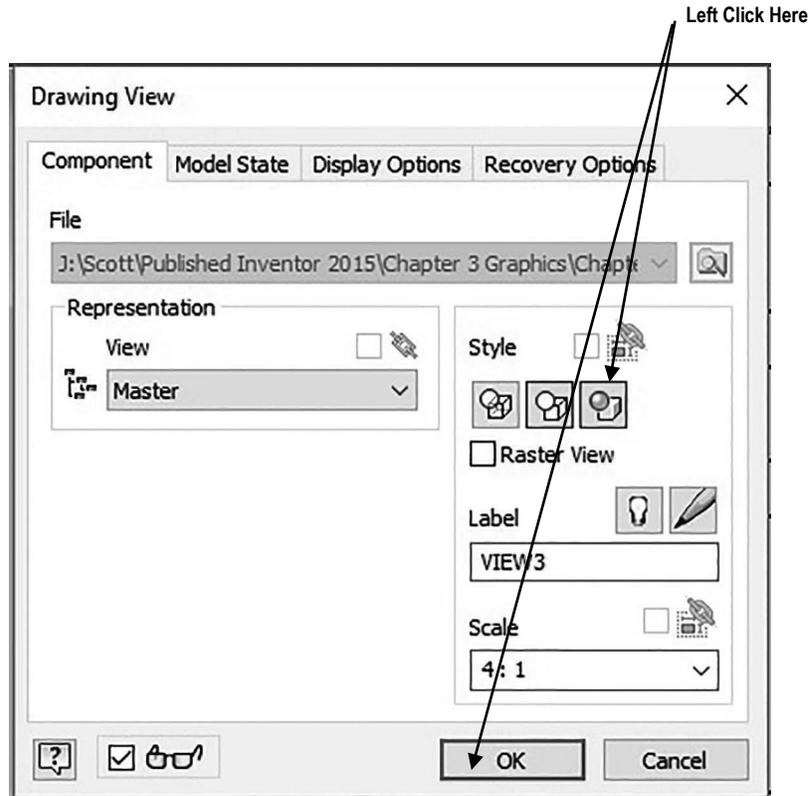


Figure 26

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

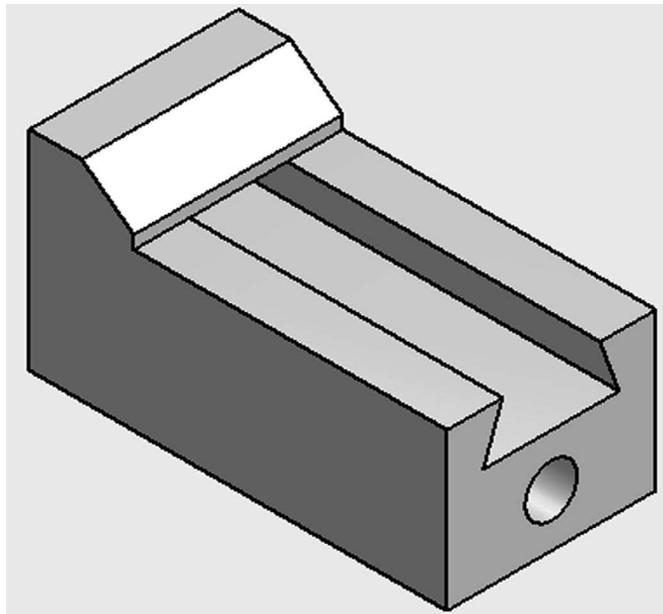
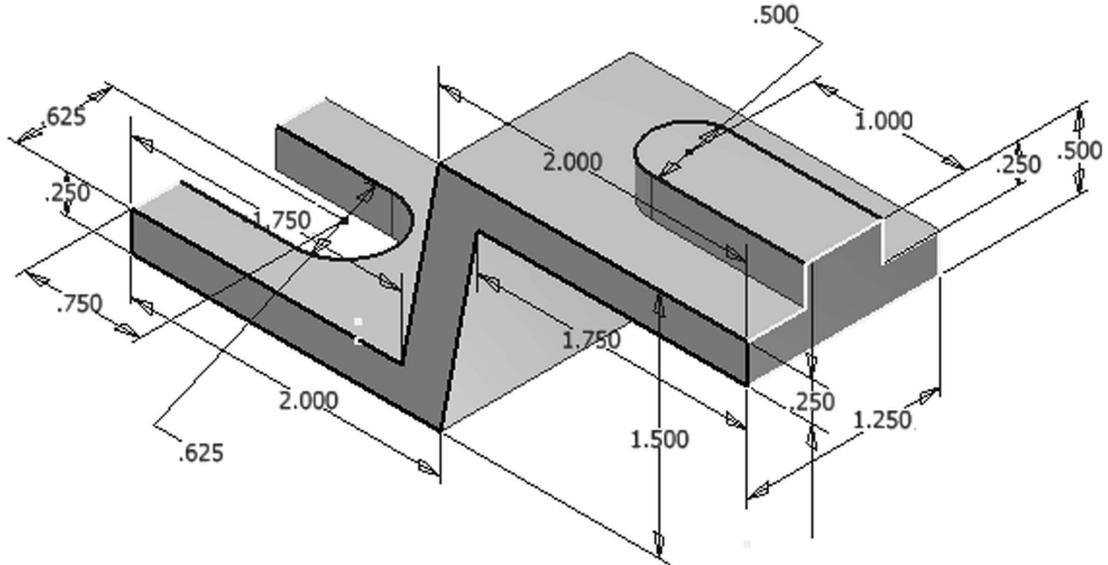


Figure 27

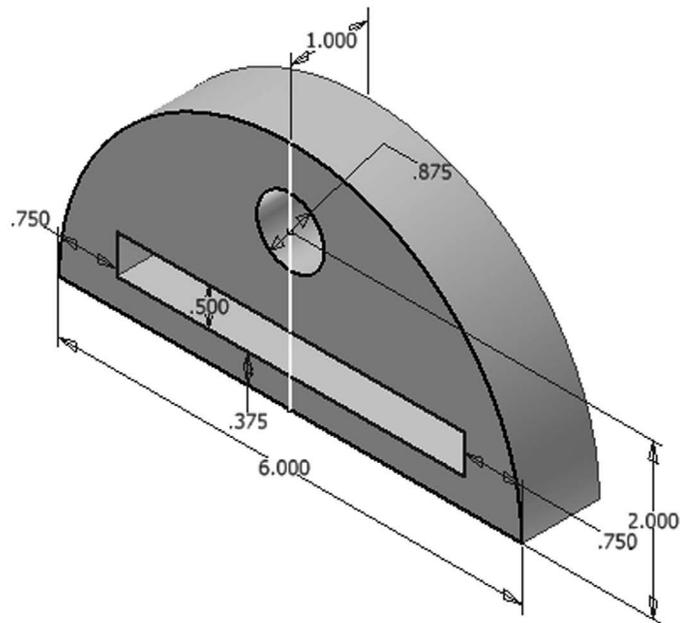
Chapter Problems

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

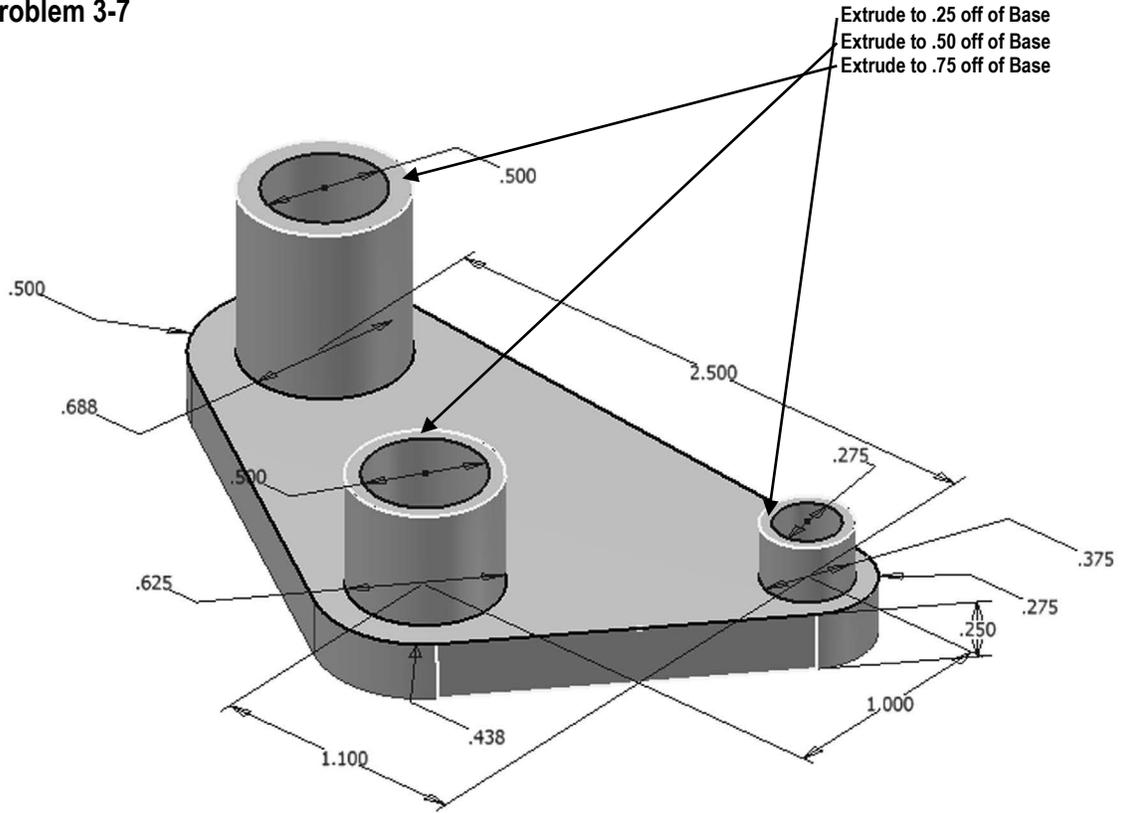
Problem 3-1



Problem 3-2



Problem 3-7



Problem 3-8

