

Autodesk[®] Inventor[®] 2015

A Tutorial Introduction



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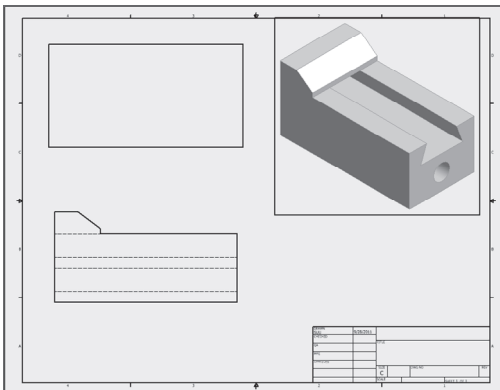
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CHAPTER 3

Learning to Create a Detail Drawing

Objectives:

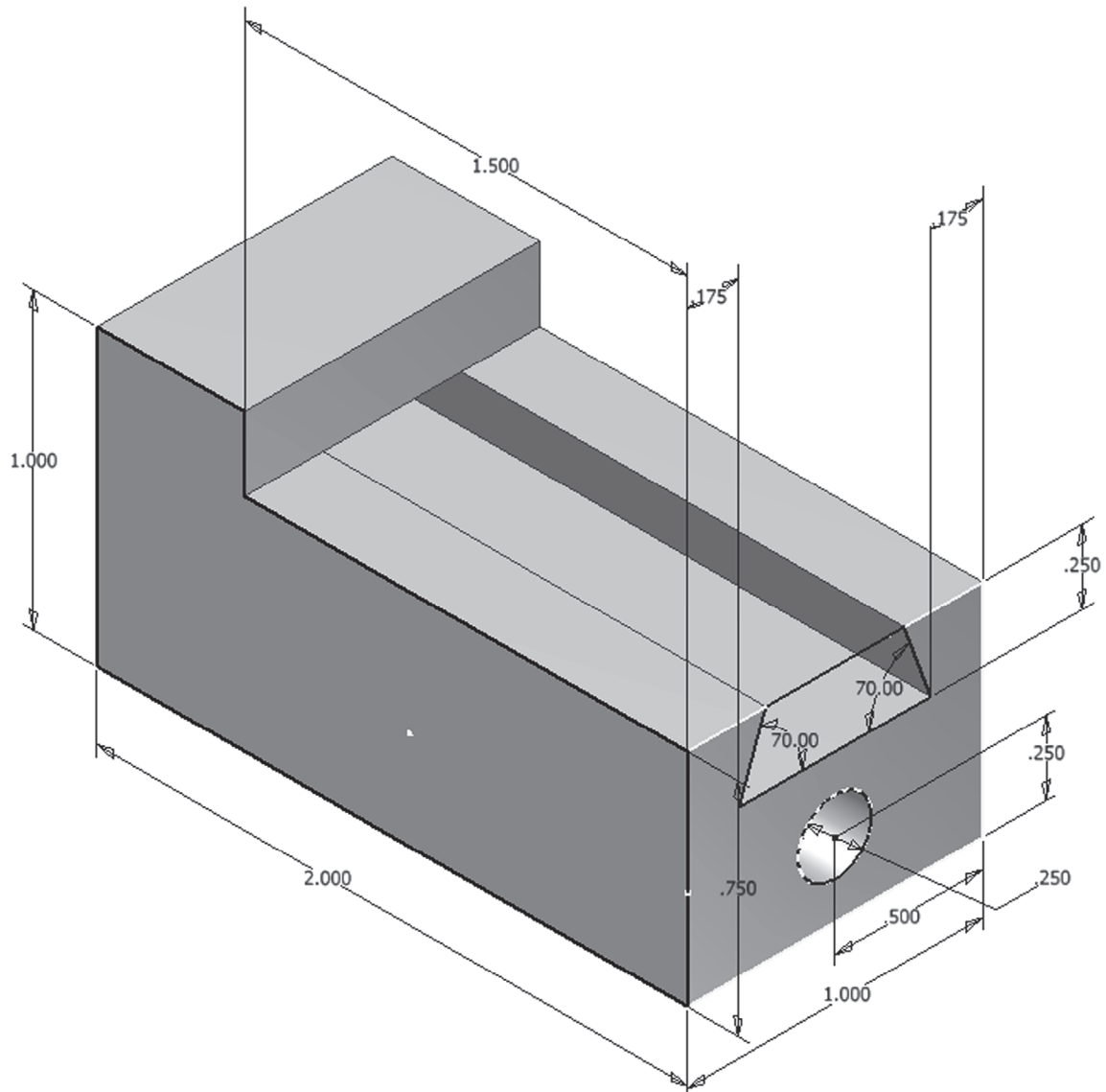


1. Create a simple sketch using the Sketch Panel
2. Extrude a sketch into a solid using the Model/Part Features Panel
3. Create an Orthographic view using the Place Views/Drawing Views Panel Extrude a sketch in the Model/Part Features
4. Edit the appearance of a Solid Model using the Edit Views command

Chapter 3 includes instruction on how to design the parts shown

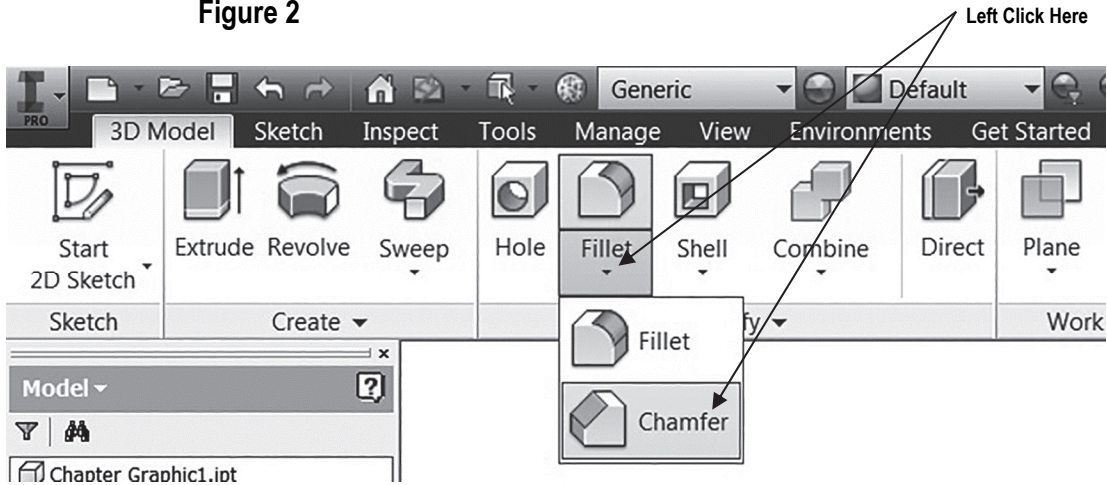
1. Start Autodesk Inventor 2015 by referring to “Chapter 1 Getting Started”.
2. After Autodesk Inventor 2015 is running, begin a new sketch.
3. Complete the drawing shown in Figure 1.

Figure 1



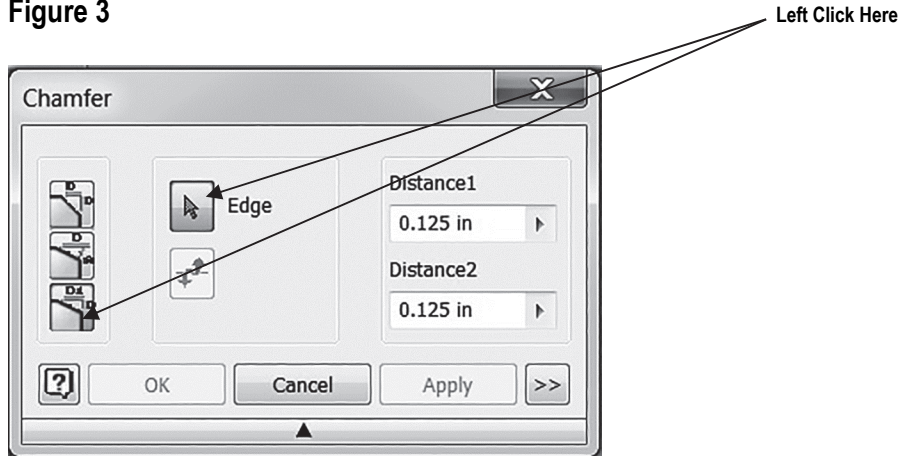
4. Move the cursor to the upper middle portion of the screen and left click on the drop down arrow under Fillet as shown in Figure 2.

Figure 2



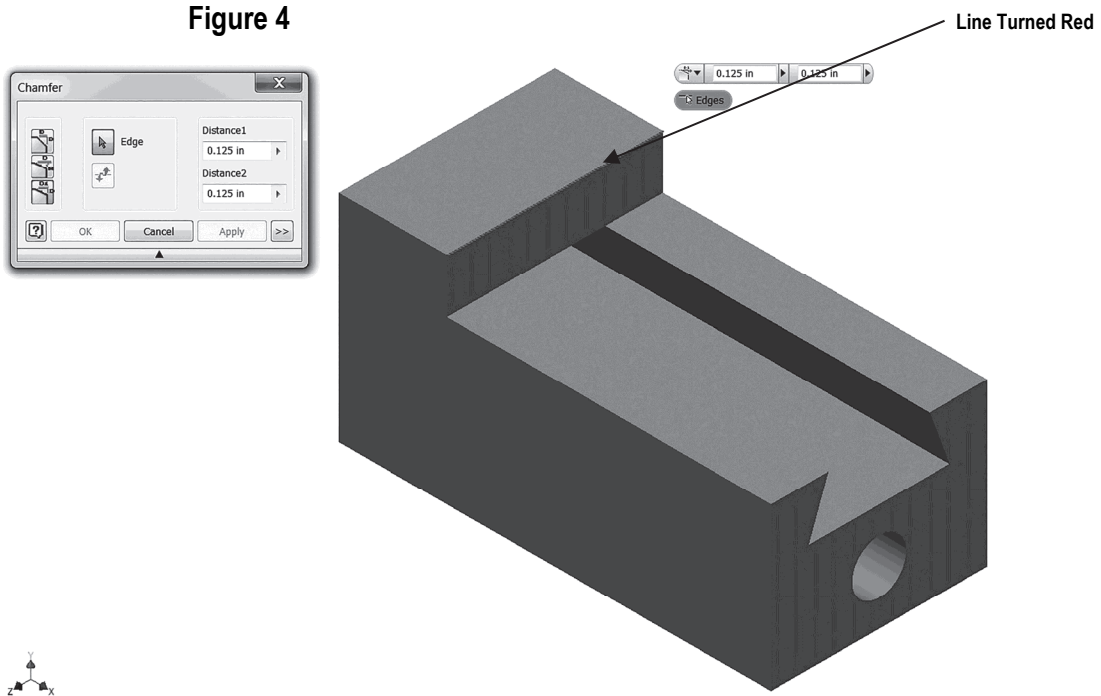
5. After selecting **Chamfer**, the Chamfer dialog box will appear. Left click on the “Two Distance Chamfer” icon. Left click on the **Edge** icon as shown in Figure 3.

Figure 3



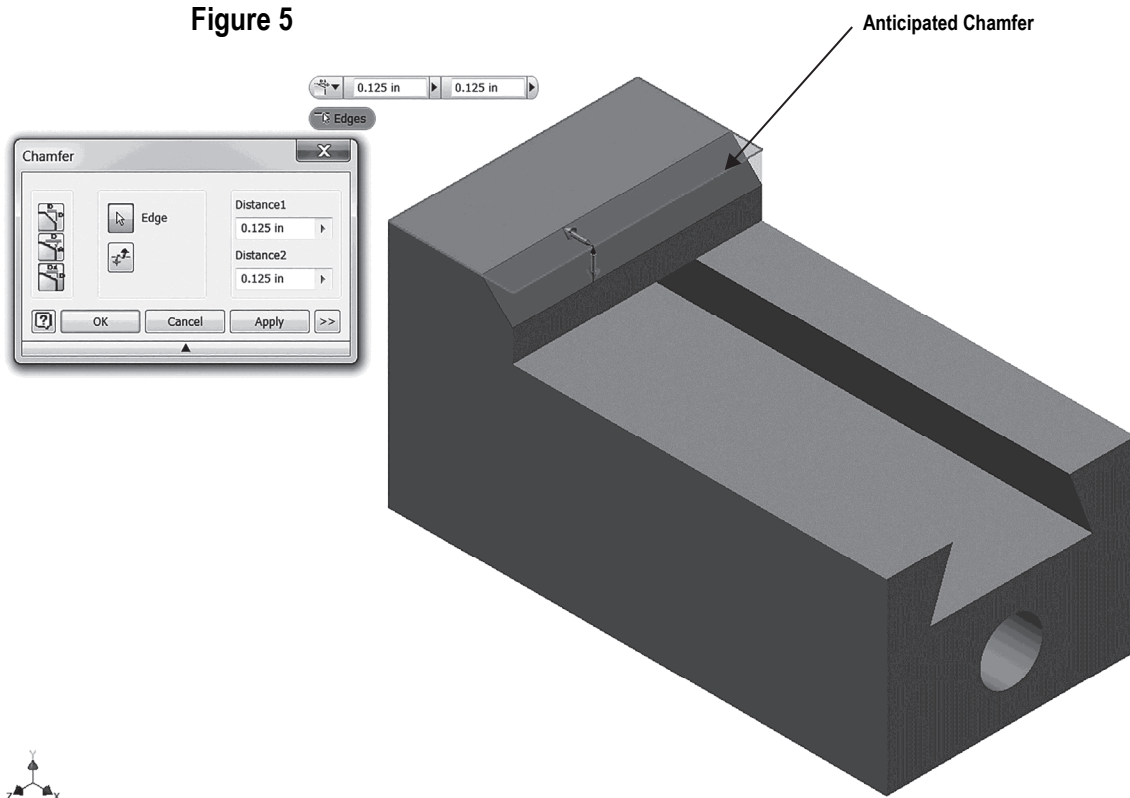
6. Move the cursor to the front upper corner. A red line will appear as shown in Figure 4.

Figure 4

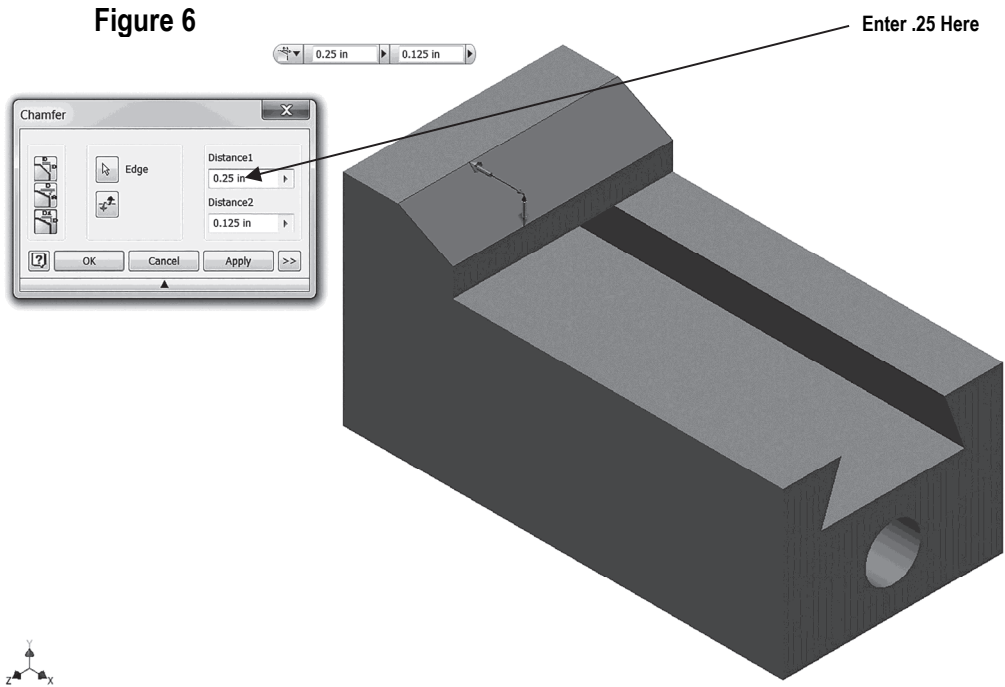


7. Inventor will provide a preview of the anticipated chamfer as shown in Figure 5.

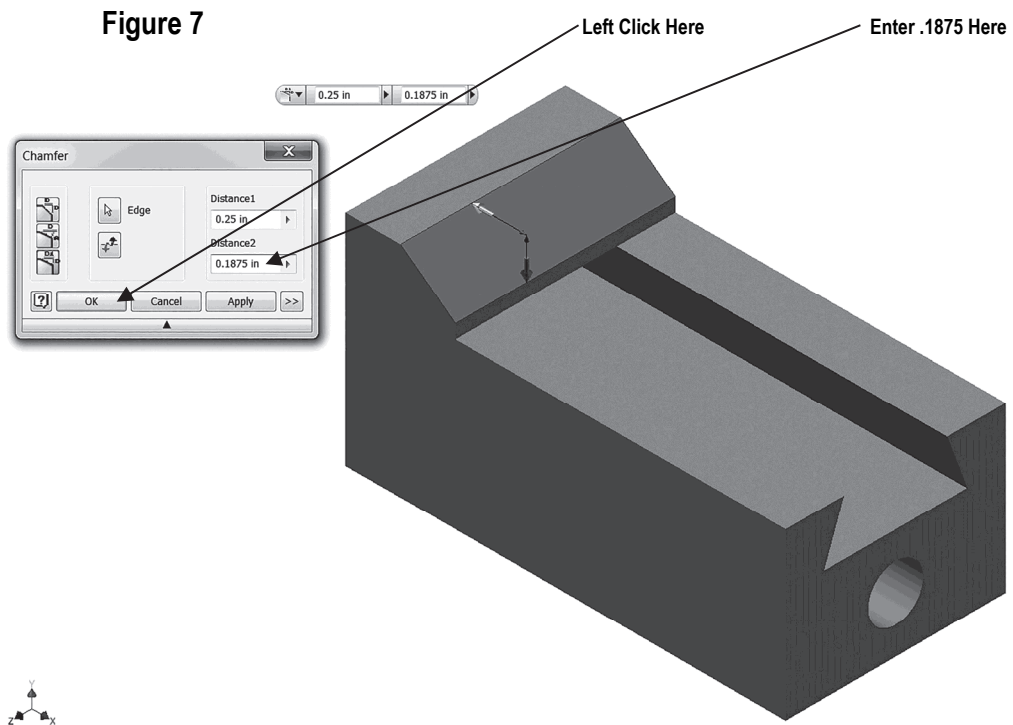
Figure 5



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.

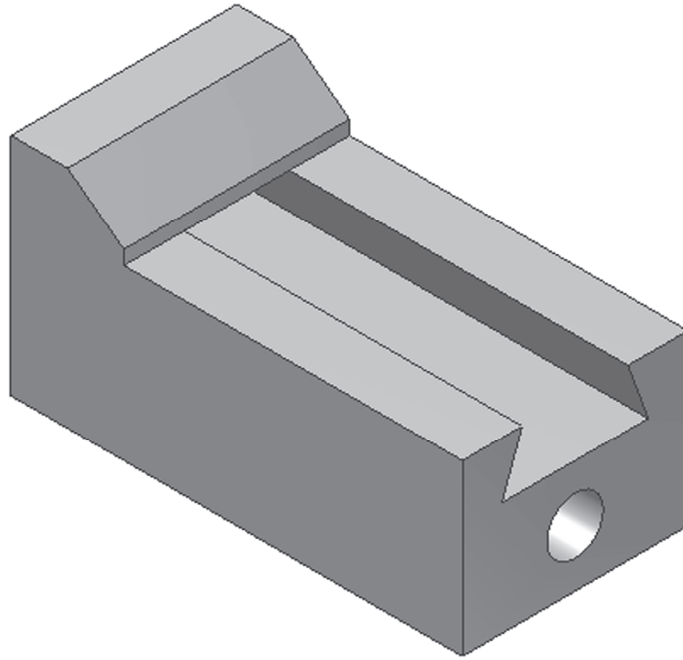


9. Move the cursor to Distance 2 in the dialog box and highlight the text. Enter **.1875** in the dialog box. Inventor will provide a preview of the chamfer as shown in Figure 7.



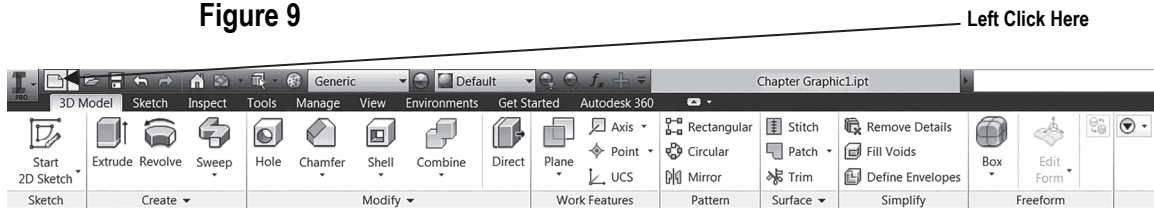
10. Left click on **OK**. Your screen should look similar to Figure 8.

Figure 8



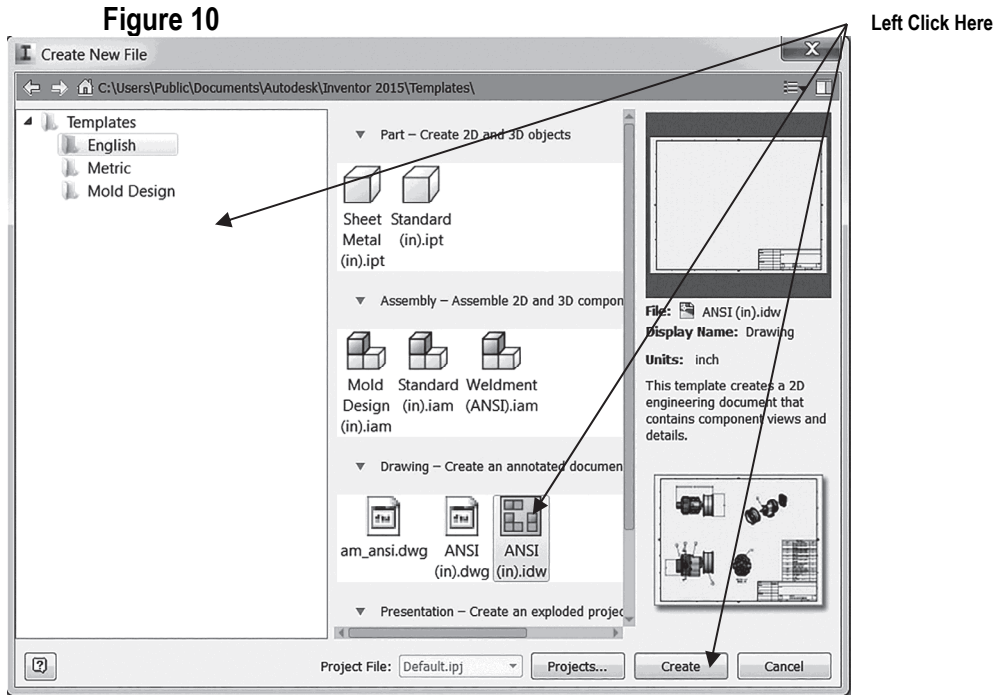
11. Save the part file for easy retrieval to be used in the following section. Do not close the part file.
12. After the part file has been saved, move the cursor to the upper left portion of the screen and left click on the “New” icon as shown in Figure 9.

Figure 9



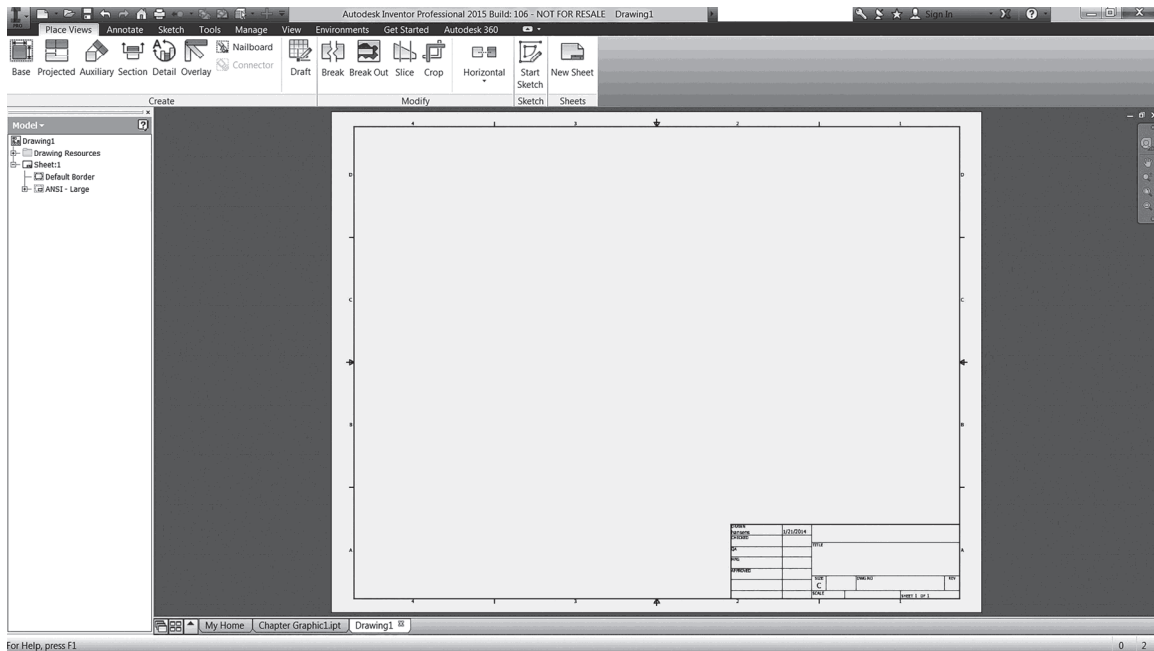
Create an Orthographic view using the Drawing Views Panel

- 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.



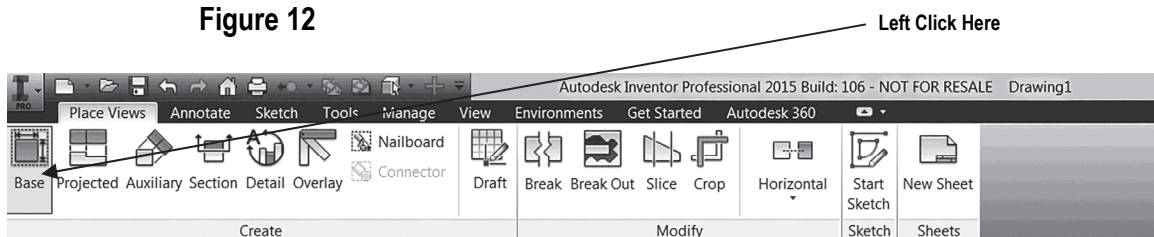
- Your screen should look similar to Figure 11.

Figure 11



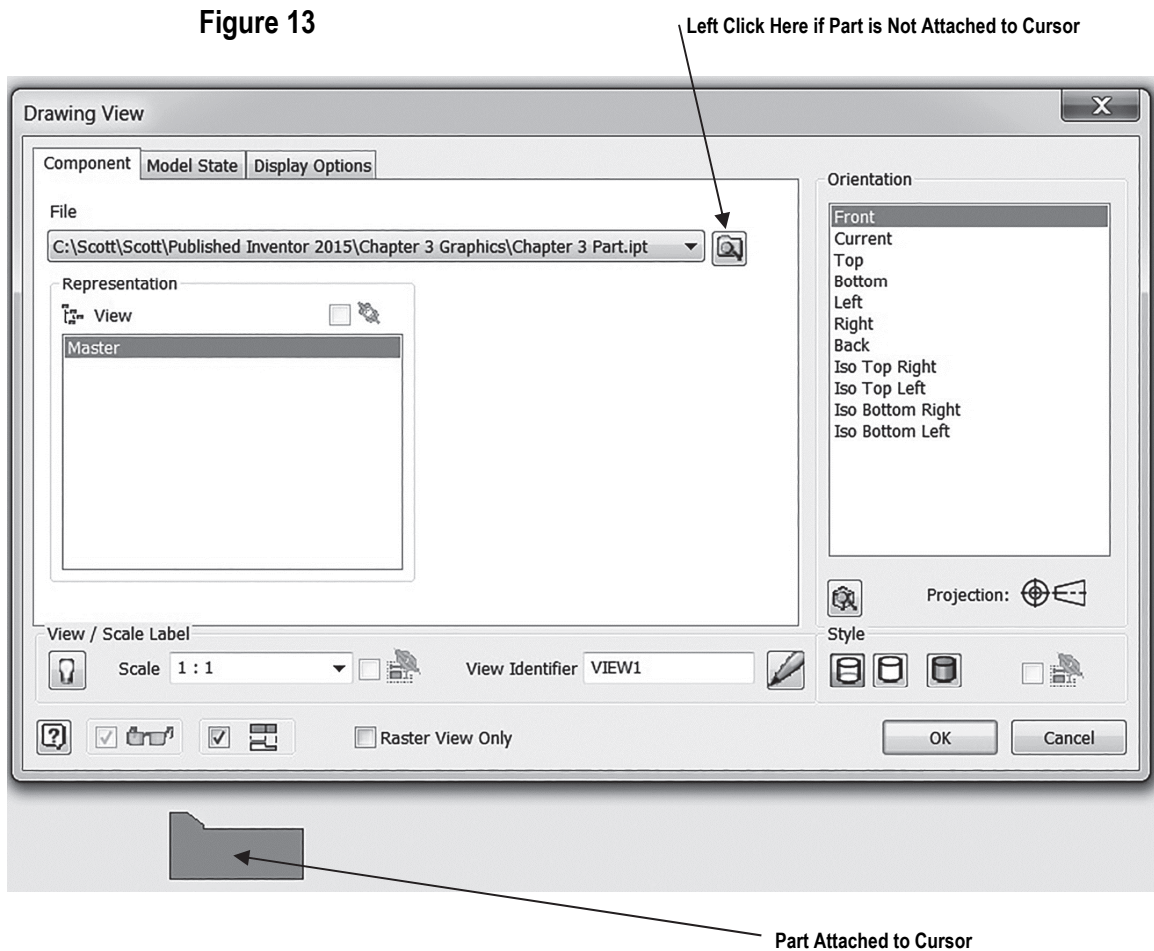
15. Inventor is now in the Place Views Panel. Notice the commands at the top of the screen are now different. The width of the screen has been reduced to add instructional clarification.
16. Move the cursor to the upper left portion of the screen and left click on **Base** as shown in Figure 12.

Figure 12

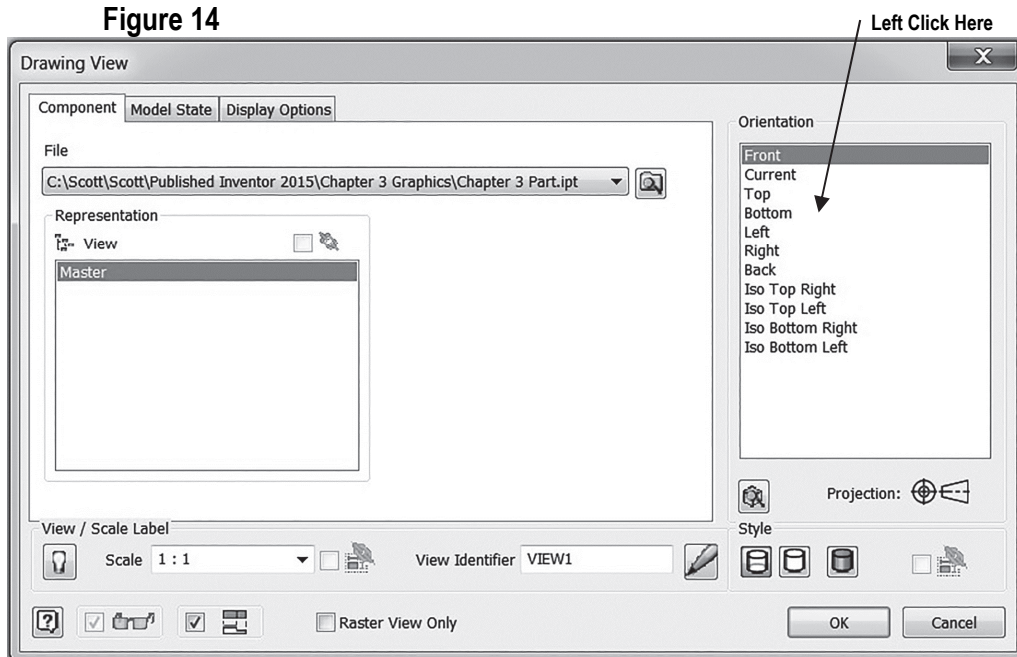


17. The drawing of the wedge block should appear attached to the cursor. Move the cursor around to verify it is attached. If the part does not appear attached to the cursor, use the "Explore" icon to locate the part file as shown in Figure 13.

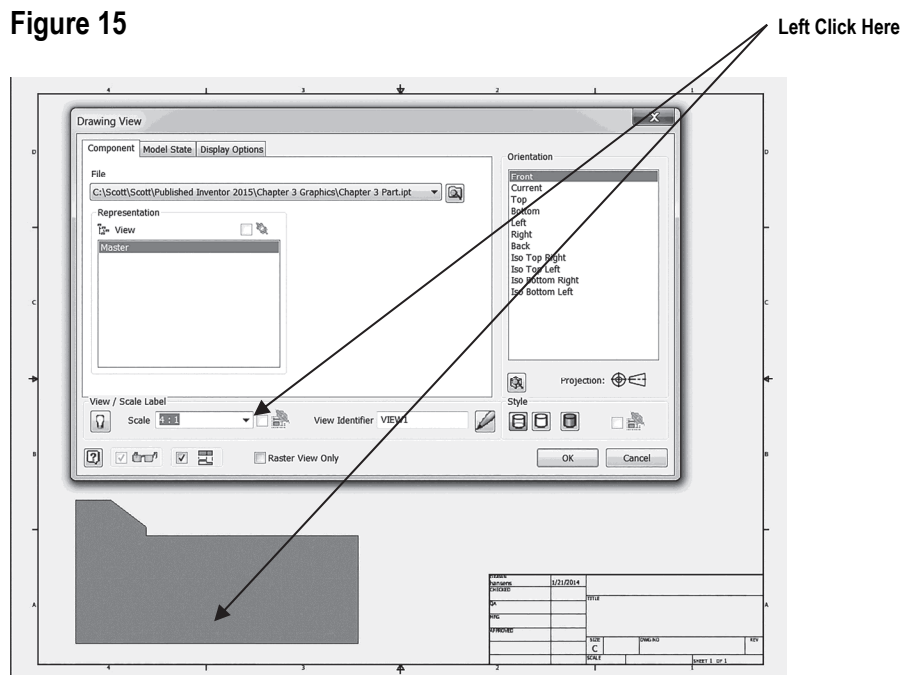
Figure 13



- Different views can be selected for the front, top, and side views. Select the desired view from the Orientation selection box as shown in Figure 14. To understand how the orientation selection works, left click on **Top** or **Left** to have the top view or left view as the front (base) view.

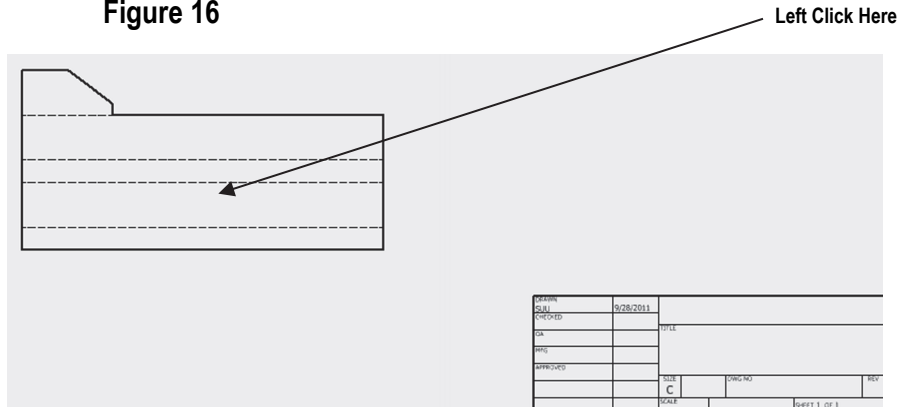


- Select the **Front** view for the base view. Left click on the **Scale** drop down box and set the drawing scale to **4:1**. The size of the wedge block will become larger as shown in Figure 15.



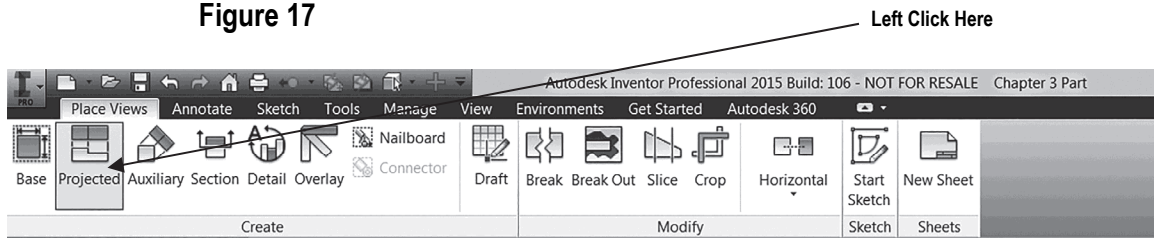
20. Place the part just above the title block that is in the lower right corner of the screen and left click once. This will place the part as shown in Figure 16.

Figure 16



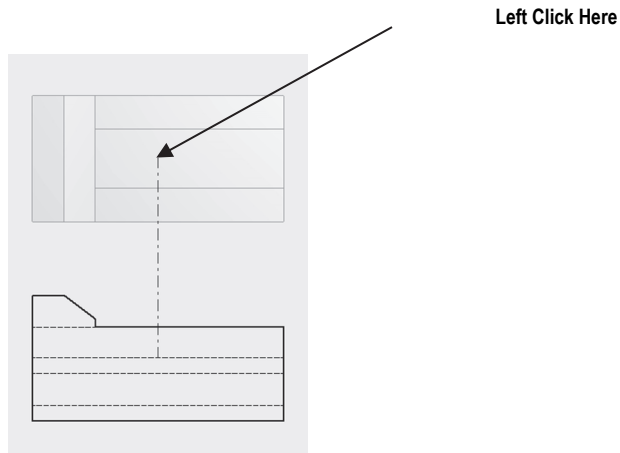
21. If the part was inadvertently placed too low or too high, move the cursor over the dots that surround the part, left click (holding the mouse button down), and drag the part to the desired location.
22. Move the cursor to the upper left portion of the screen and left click on **Projected** as shown in Figure 17.

Figure 17



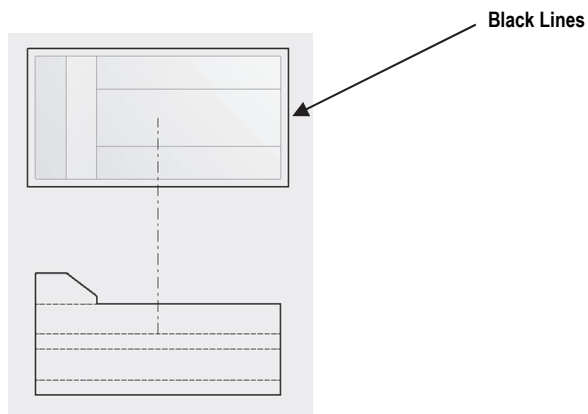
23. The part will be attached to the cursor. Move the cursor upward and left click as shown in Figure 18.

Figure 18



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

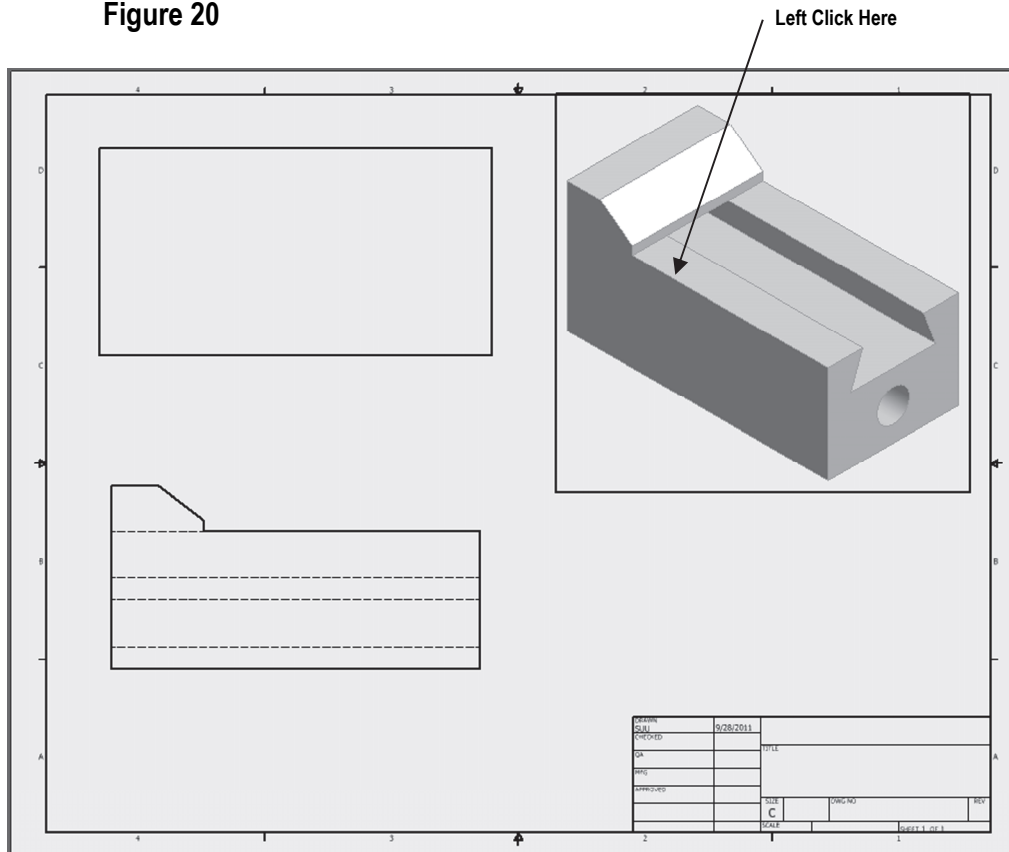
Figure 19



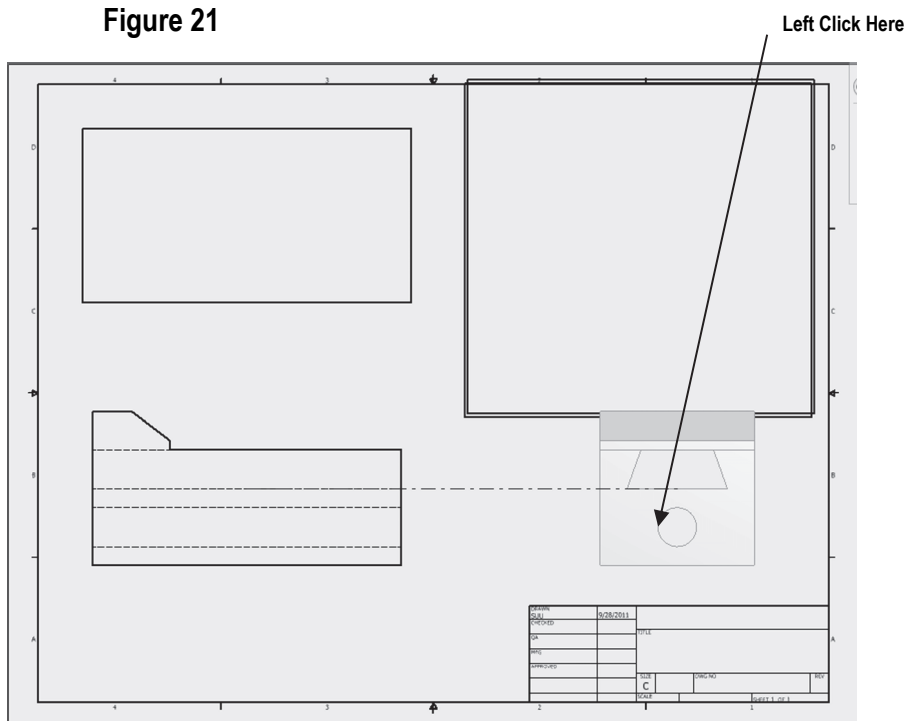
Create a Solid Model using the Edit Views command

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

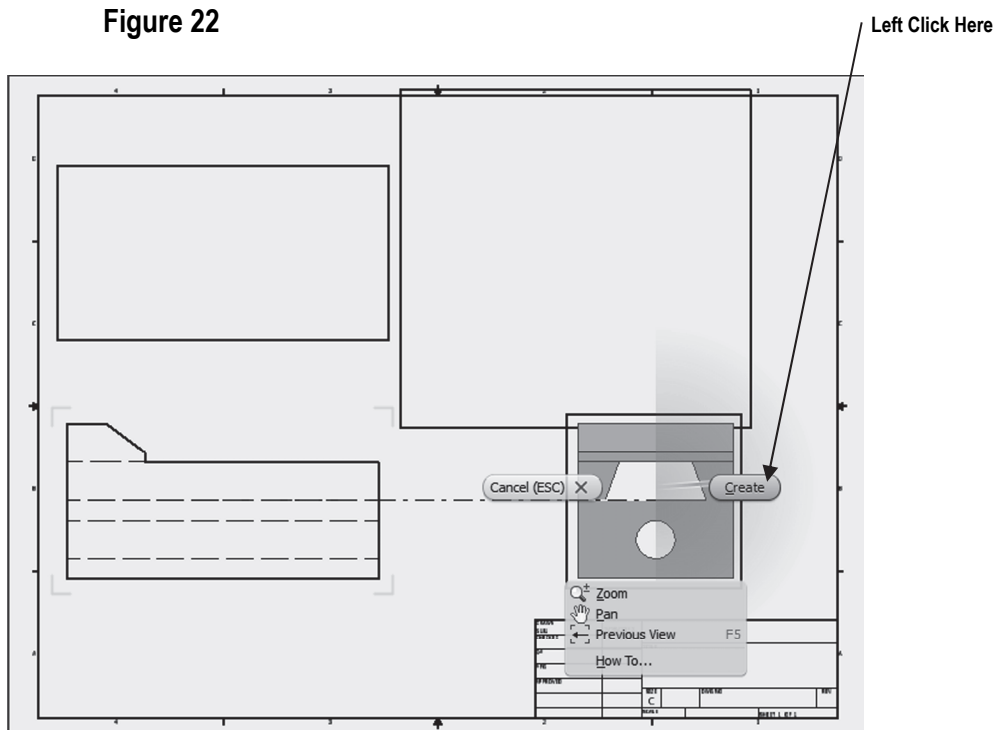
Figure 20



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

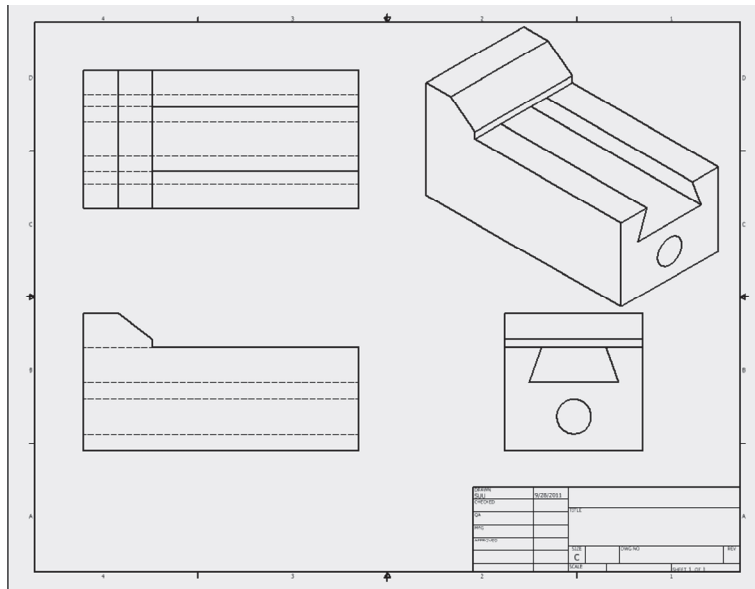


27. Right click on the last view created (side view). A pop up menu will appear. Left click on **Create** as shown in Figure 22.



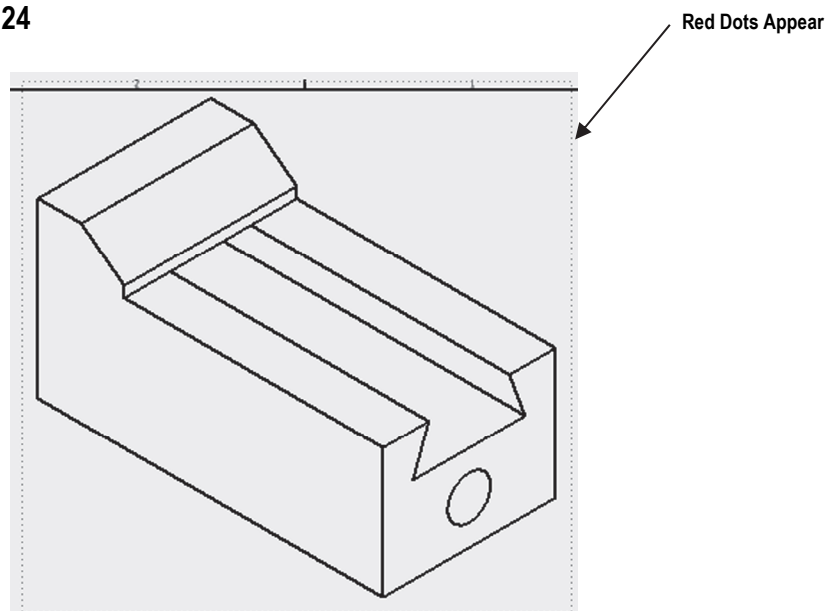
28. Your screen should look similar to Figure 23.

Figure 23



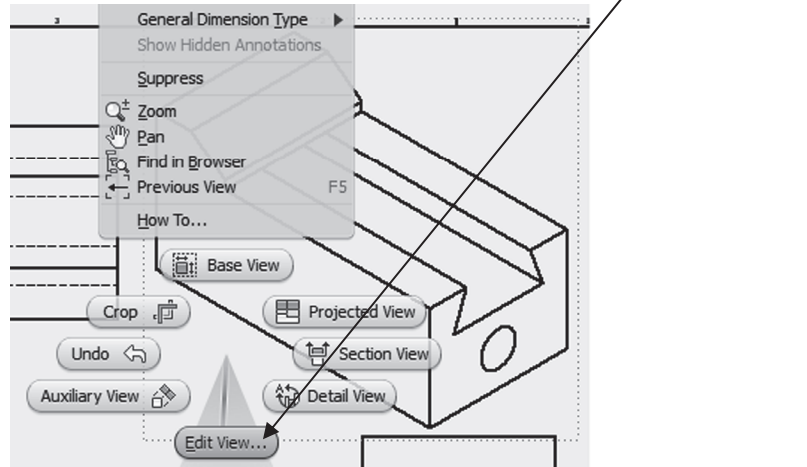
29. Move the cursor over the isometric view in the upper right corner of the drawing. Red dots will appear as shown in Figure 24.

Figure 24



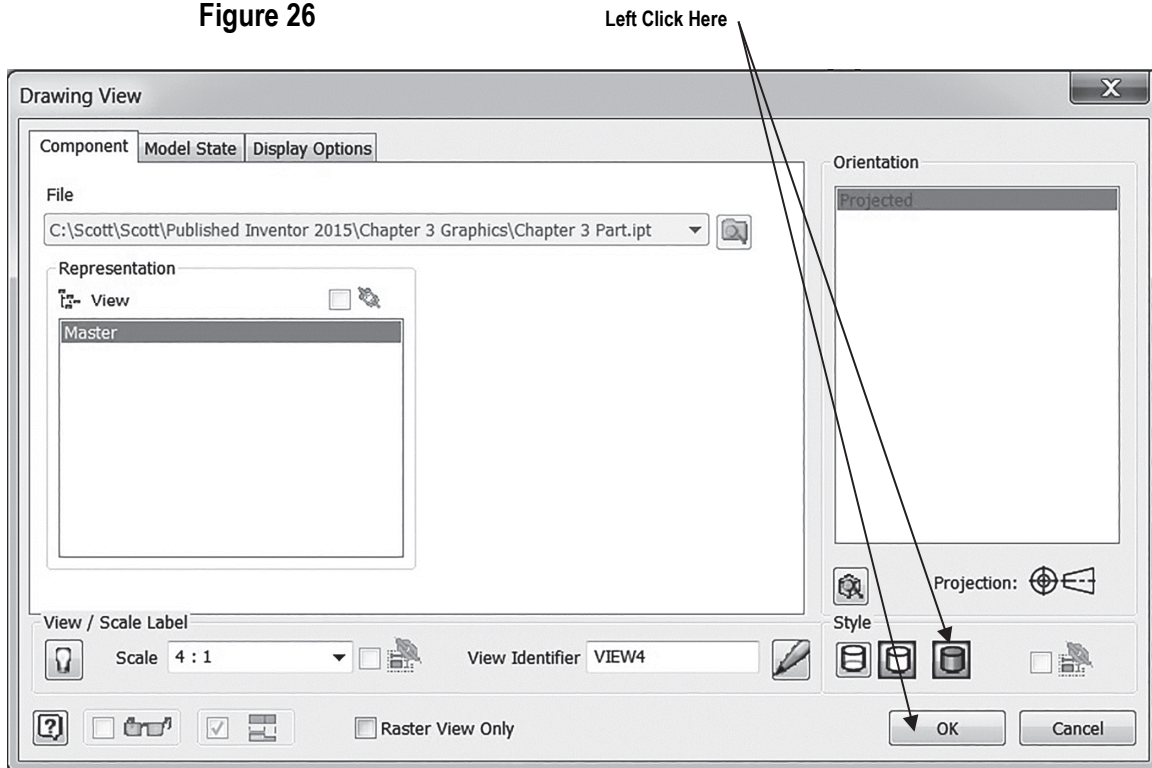
30. After the red dots appear, right click once. A pop up menu will appear. Left click on **Edit View** as shown in Figure 25.

Figure 25



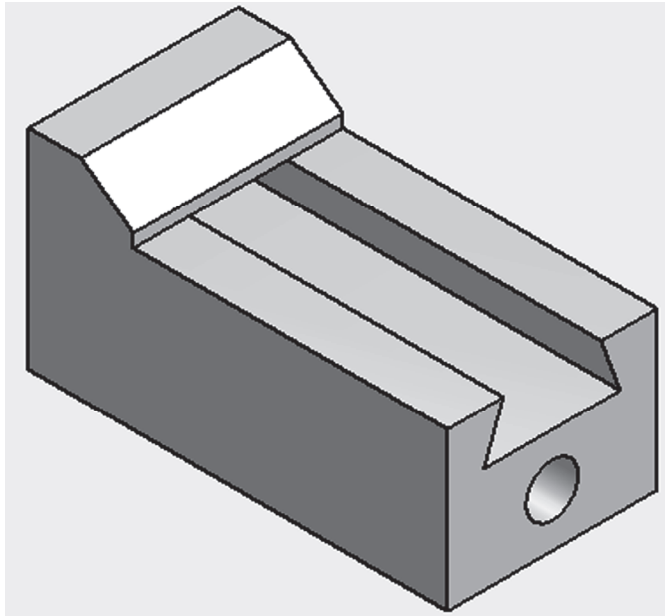
31. The Drawing View dialog box will appear. Left click on the “blue barrel” to the right under Style. Left click on **OK** as shown in Figure 26.

Figure 26



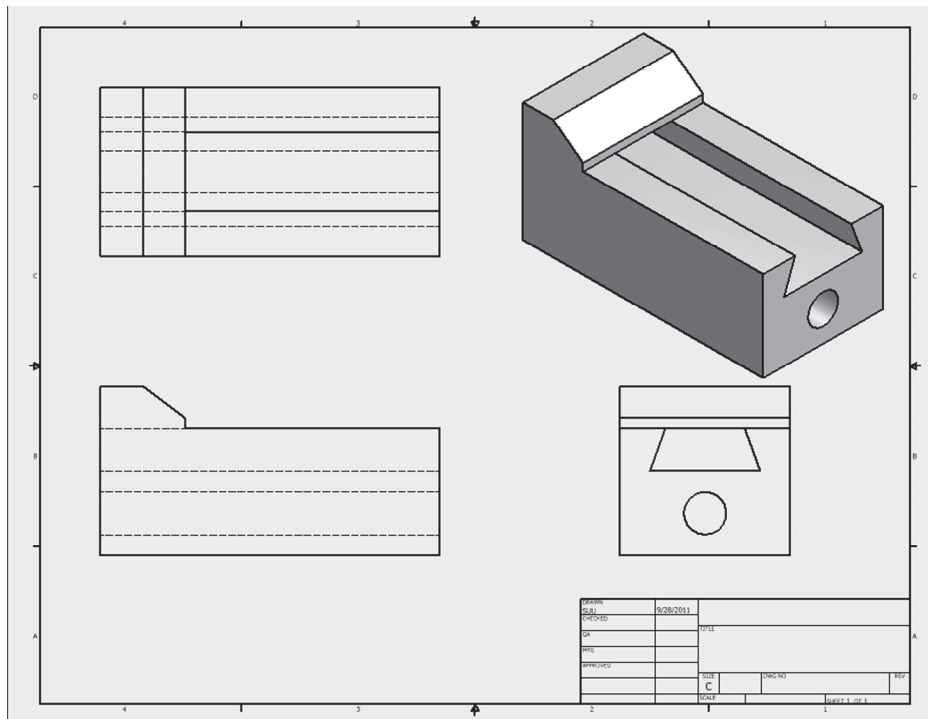
32. The isometric view will become a miniature solid model as shown in Figure 27.

Figure 27



33. Your screen should look similar to Figure 28.

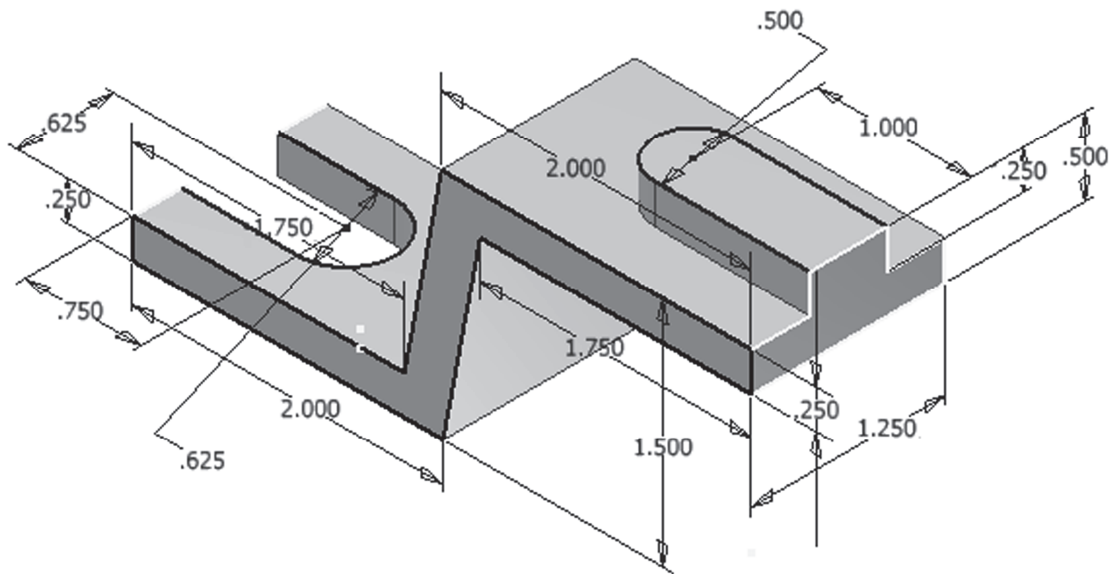
Figure 28



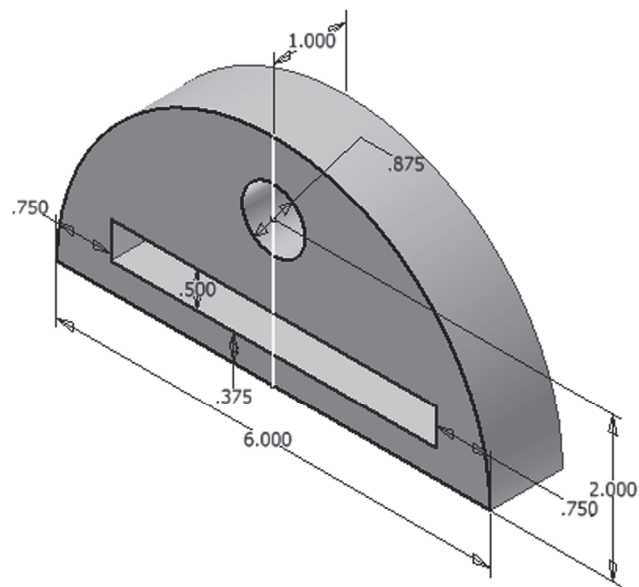
Chapter Problems

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

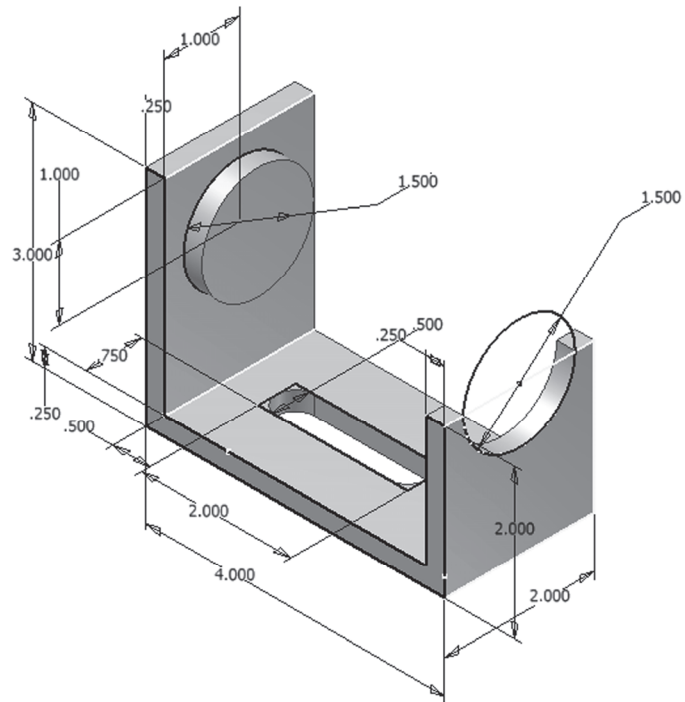
Problem 3-1



Problem 3-2

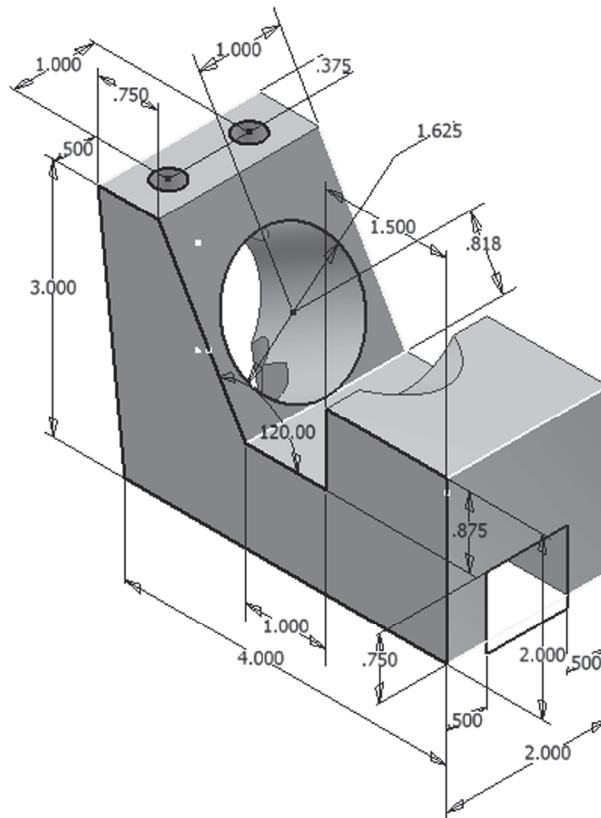


Problem 3-3

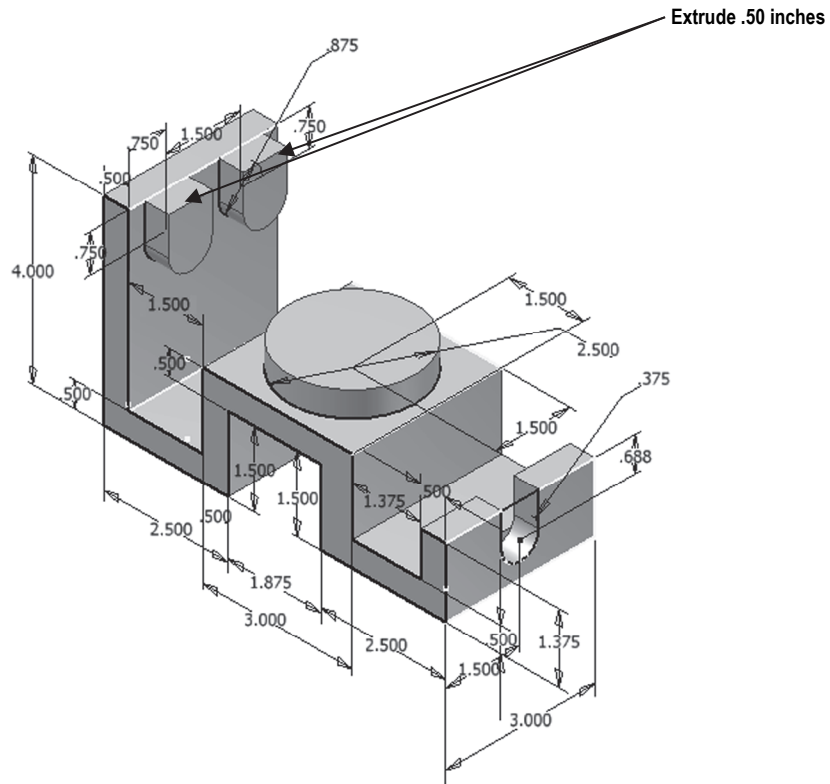


Problem 3-4

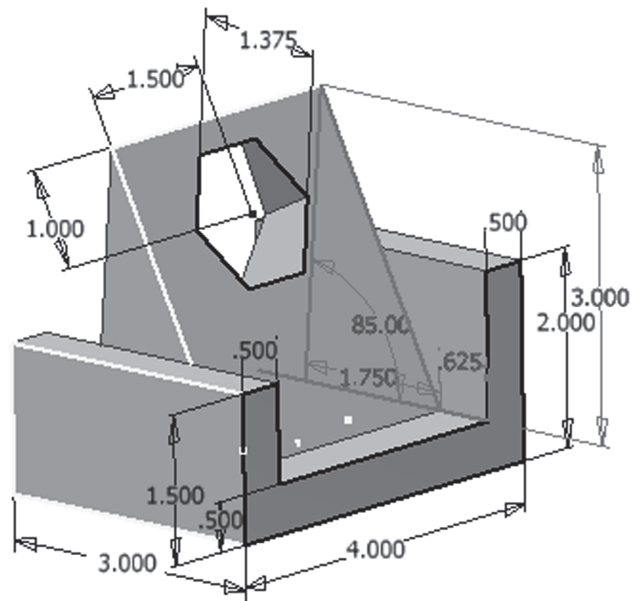
Hint: Use the 2 directional
Extrude-Cut to create
The 1.625 Diameter Hole



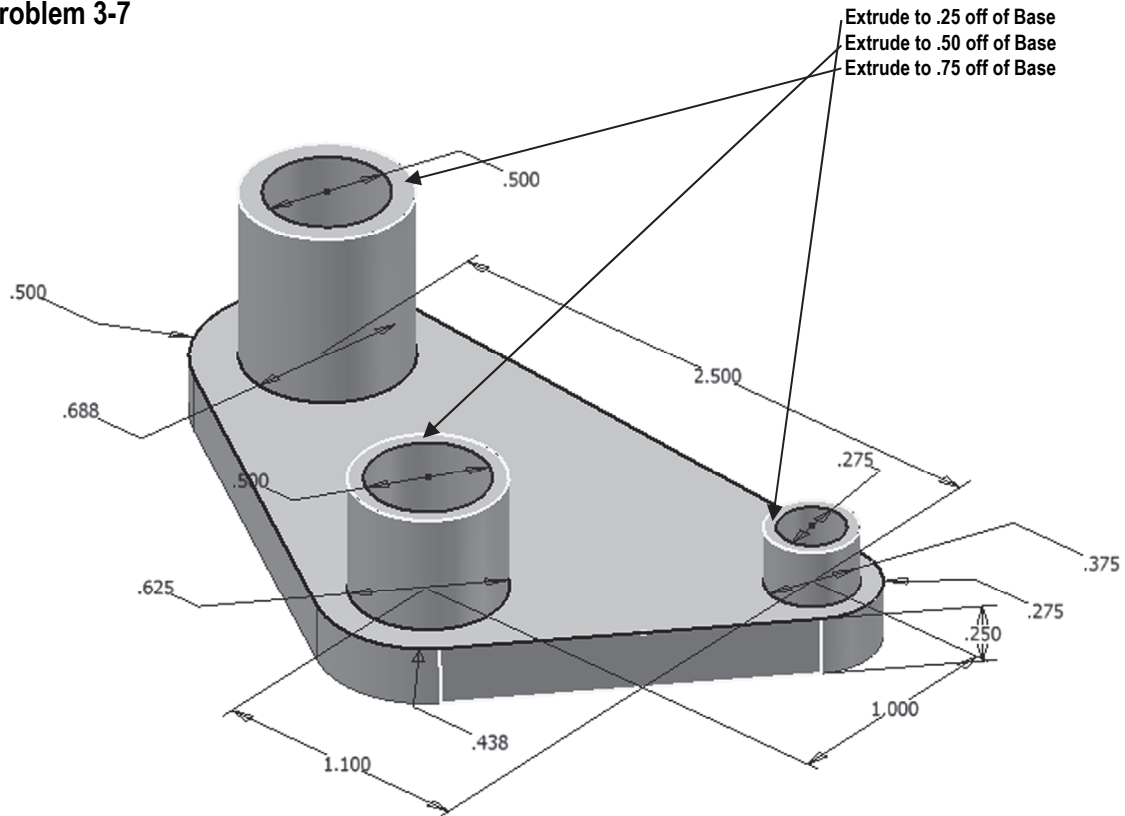
Problem 3-5



Problem 3-6



Problem 3-7



Problem 3-8

