## Autodesk<sup>®</sup> Inventor<sup>®</sup> 2015 A Tutorial Introduction



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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.



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.



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



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



- **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.

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#### 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.

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

Figure 11

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- **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					Le	eft Click He	ere
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**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	Left Click Here if Part is Not Attached to Cursor
Drawing View	X
Component Model State Display Options File C:\Scott\Scott\Published Inventor 2015\Chapter 3 Graphics\Chapter 3 Part.ipt Representation T_a- View Master	♥ Orientation          Front         Current         Top         Bottom         Left         Right         Back         Iso Top Right         Iso Top Left         Iso Bottom Right         Iso Bottom Left
Scale 1:1 View Identifier VIEW1	
Raster View Only	OK Cancel
	Part Attached to Cursor

**18.** 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.

Figure 14	/ Left Click Here
Drawing View	X
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View / Scale Laber	
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**19.** 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.



- **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	Left Click Here	
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Create	Modify Sketch Sheets	

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



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



#### 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.



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.



**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	Left Click Here
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Crop I Projected View Undo C	
Auxiliary View	

**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	Left Click Here
Drawing View	×
Component       Model State       Display Options         File       C:\Scott\Scott\Published Inventor 2015\Chapter 3 Graphics\Chapter 3         Representation       Fig View         Master       Master	Part.ipt
View / Scale Label Scale 4 : 1 View Identifier Raster View Only	VIEW4

**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.





#### Chapter Problems

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

Problem 3-1



Problem 3-2



#### Problem 3-3



#### Problem 3-5











