



Assignment 5

Inventor

Engineering Drafting

Objectives

When you complete this assignment you will:

1. create an engineering drawing file using the management file menu.
2. dimension the engineering file using the drawing annotation menu.
3. produce engineering drawings from Inventor model files.

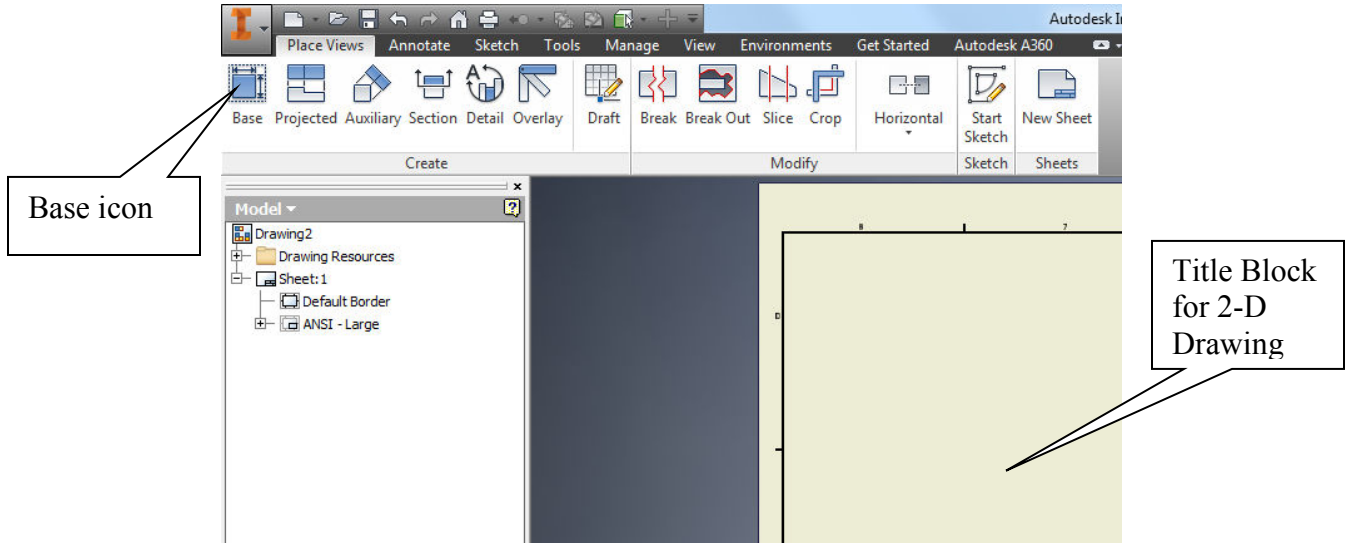
General Instructions

In this activity you will open an existing model file and create an engineering drawing file from that file. Industry primarily uses two-dimensional drawings for the production of products. This method for production is rapidly changing. Many design programs work and originate drawings in a three dimensional environment without ever working in the two dimensional mode. Industry still uses two-dimensional drawings. In this activity you learn to take a three-dimensional model back into a two-dimensional setting.

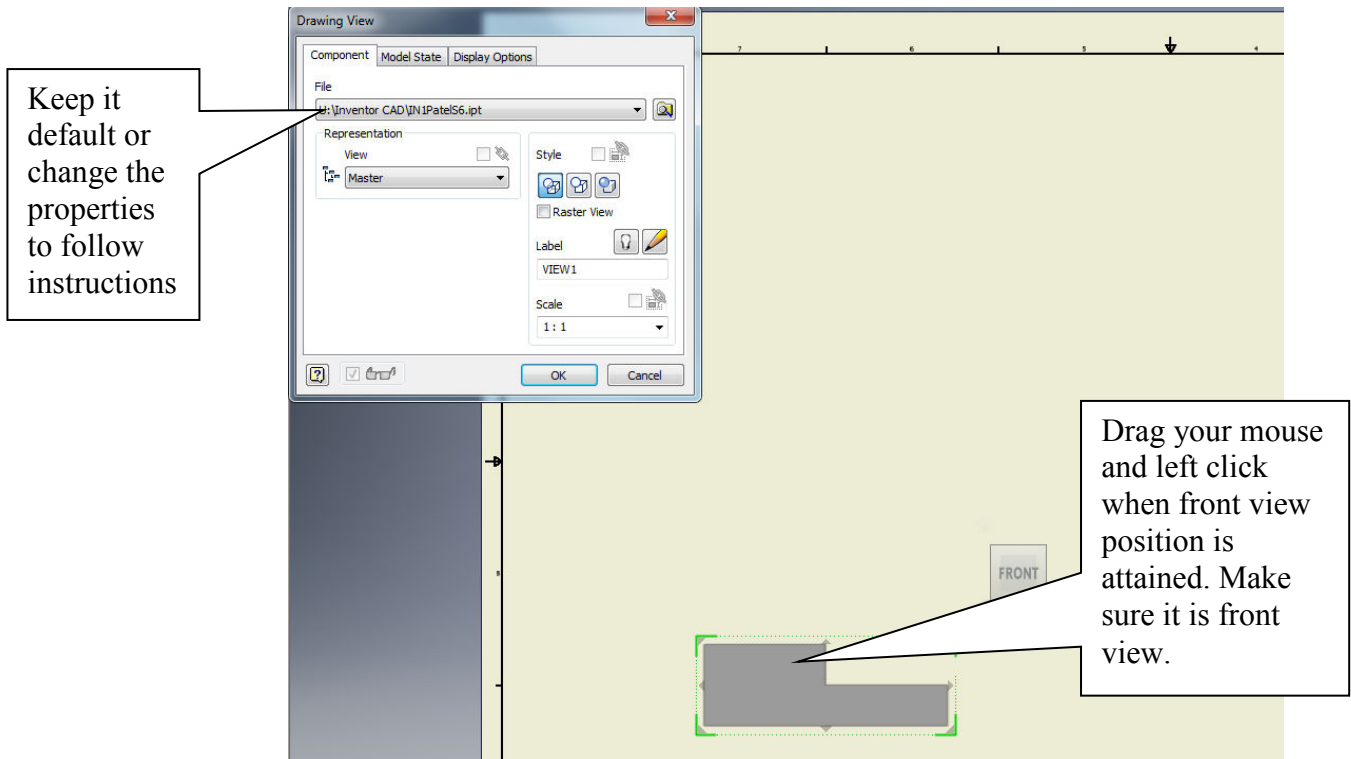
Getting Started

1. Launch Inventor 2016 and open the first Inventor file that you created, IN-1, End Lap Member.
2. After the file opens, select file and new. You will now select **Standard.idw** template file. This template file is used to create **engineering drawings** from model files.

3. A page with a title block will appear after the Standard.idw file is loaded. You will now select the **Base** icon from the Place Views menu.

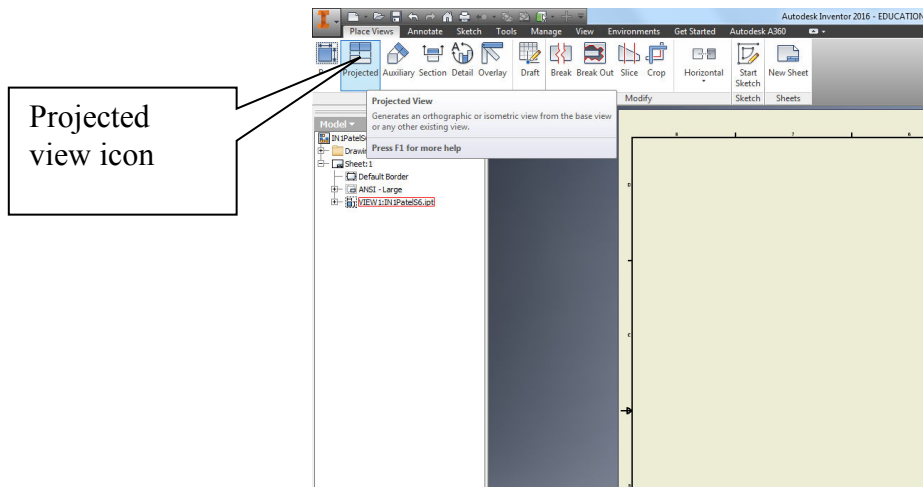


4. After selecting the Base view icon the dialog box opens:

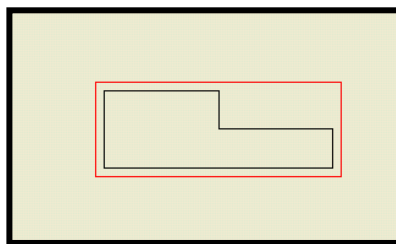




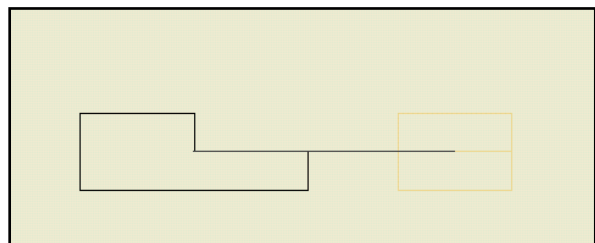
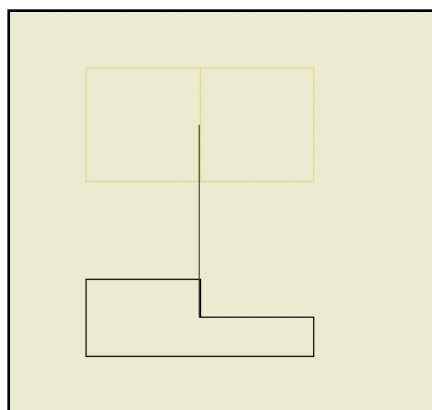
5. The **front view** should be **positioned** in the **lower left corner** of the page.
6. Select the **projected view** icon from the Drawing Management menu.



7. Position the mouse over the front view:

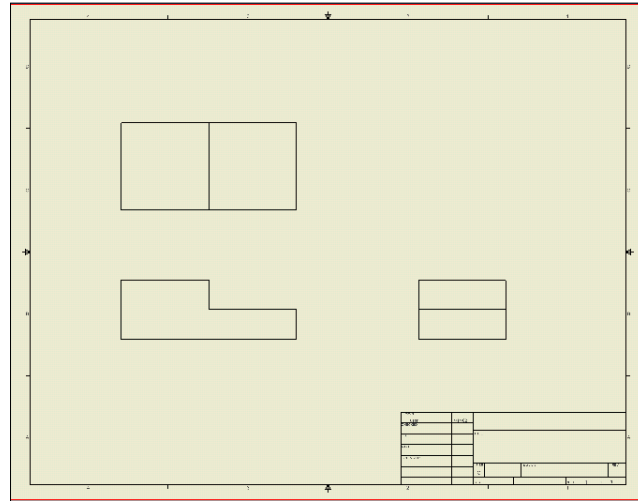


8. **Left-click** the mouse and move the mouse above the front view. Notice the top view appears. Left click one time. Then move the mouse to the right of the front view and notice the right side view appears. Left click again.



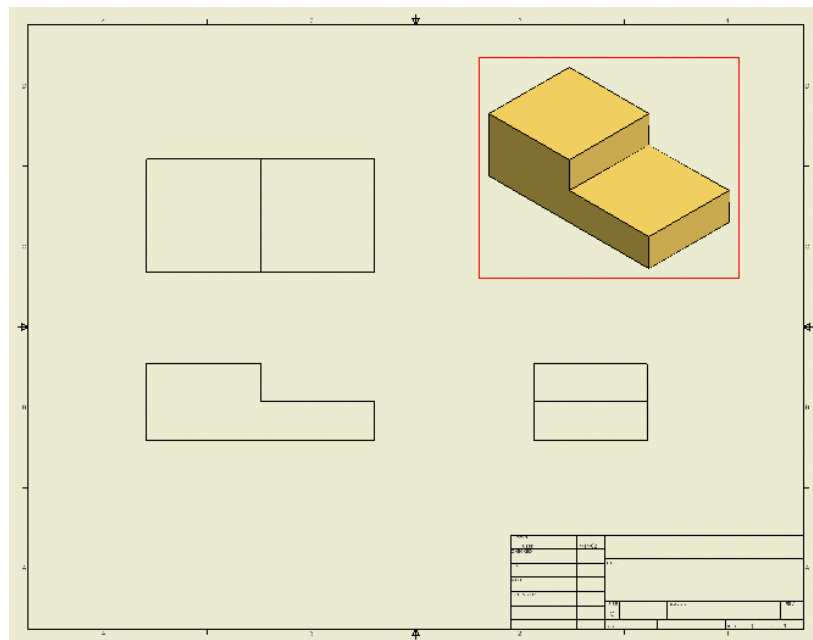


9. Your drawing should look similar to the one below.



10. Continue with the projected view. This time you are going to insert a three-dimensional **shaded model** into the upper right corner of the title block.

11. Position the model and left click. The drawing should appear as illustrated:



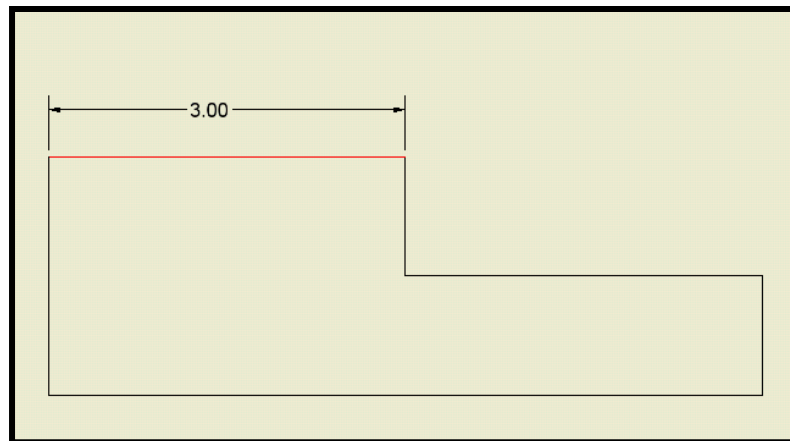
Dimensioning the drawing

1. To dimension the drawing the same rules should be followed as you did in the Orthographic Projection Module. Remember place dimensions between views

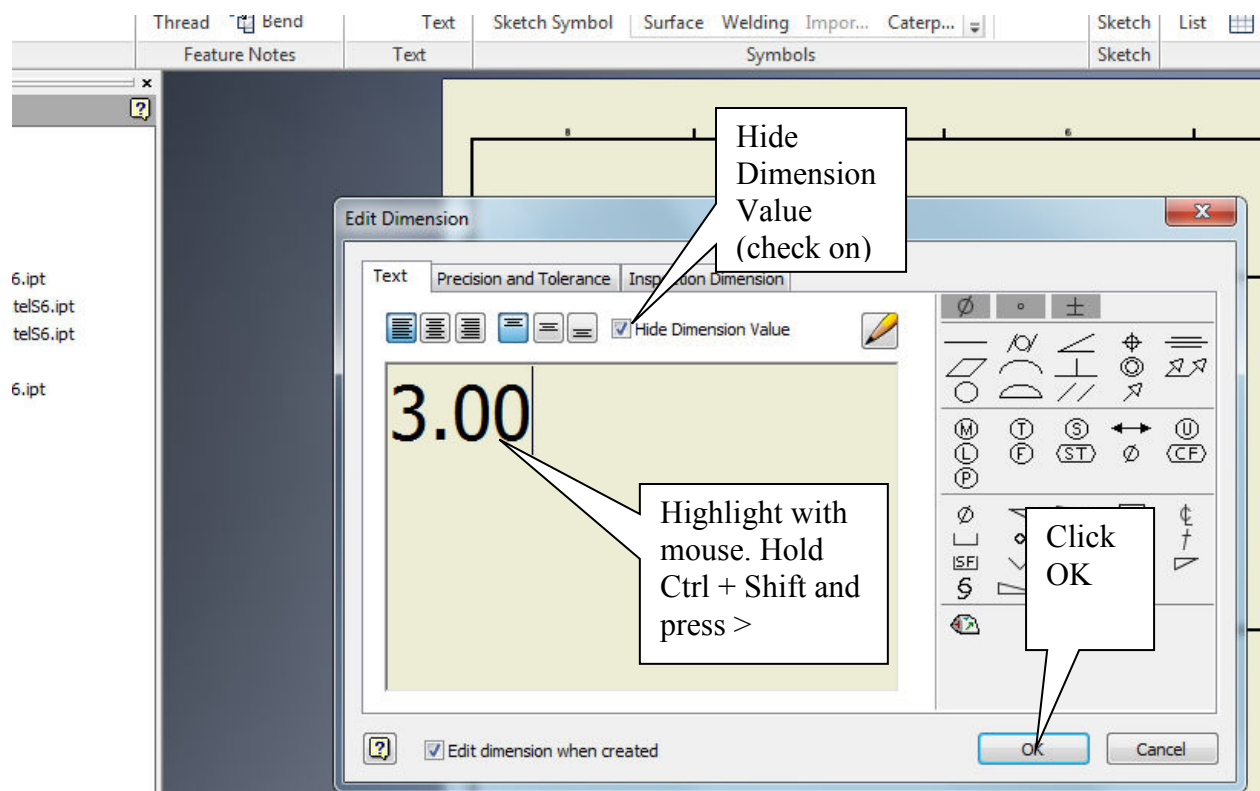


when possible. Shorter dimensions should be placed inside of over-all dimensions. Dimensions should only be placed one time on a drawing.

2. **Zoom** into the front view. Select the dimension tool icon from the **Drawing Annotation** tool bar (if this toolbar is not visible, choose it from the **View** drop down menu and the **toolbars** option). Touch the top view with the pointer the line turns red. Left-click the line and move upward to position the dimension as illustrated:

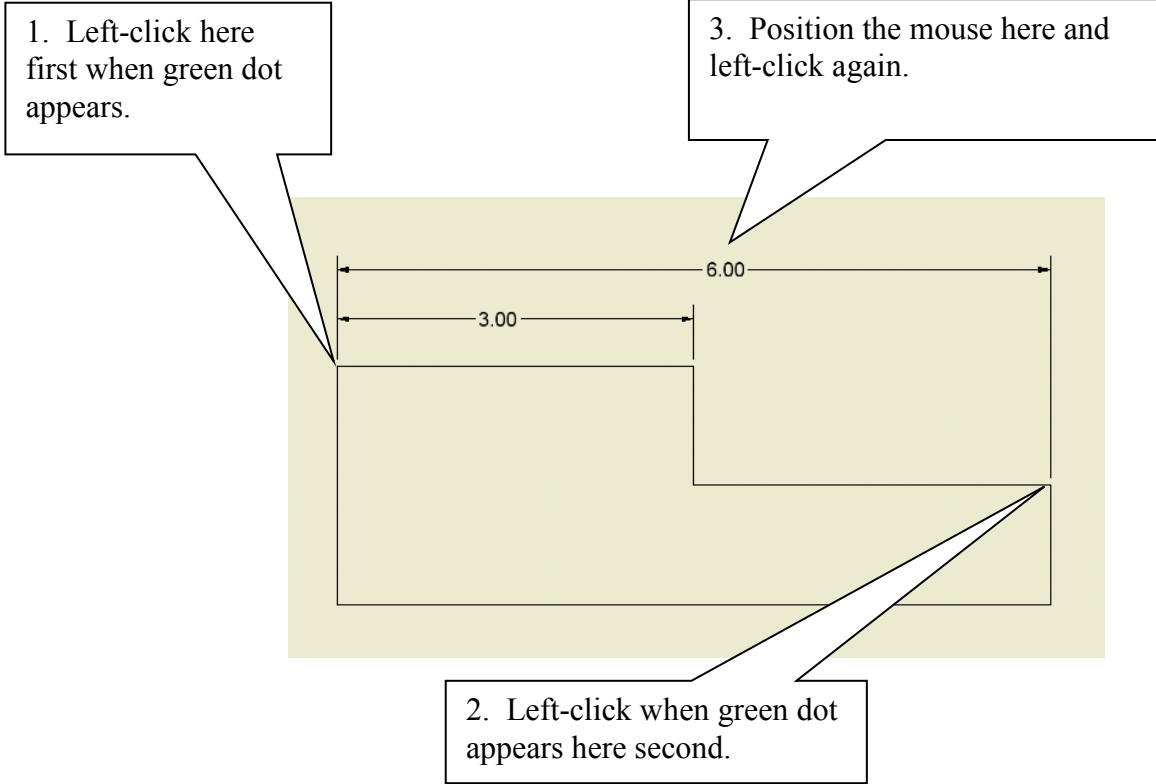


(To make the dimension number larger, click Hide Dimension Value and enter in the measurement of the line, zoom in to see the number, and highlight it using your mouse. Once you highlighted, hold Ctrl+Shift and press > until it is large enough to read without zooming in.)

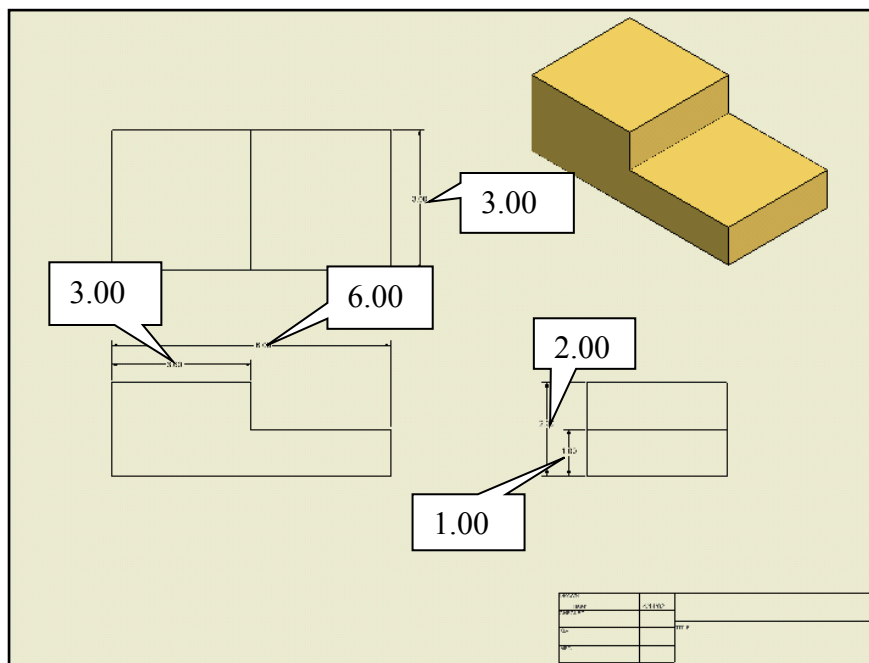




- To place the over-all dimension of six inches touch the upper left corner of the drawing. A green dot appears left-click the mouse then touch the second position and left-click again. Move the mouse above the three inch position and left click again. Use the previous steps to change the dimension size.

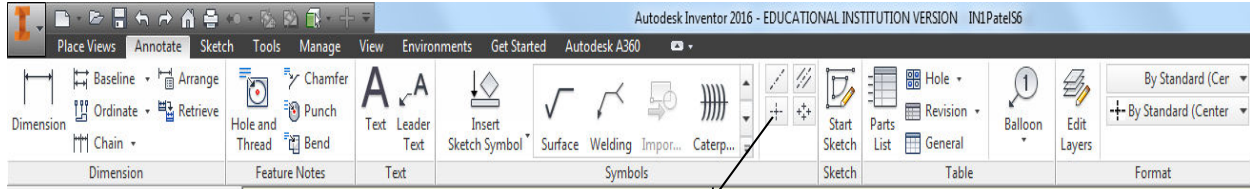


- Complete all of the dimensions as illustrated:

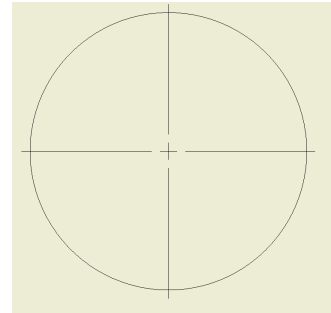




5. This activity did not include holes or circles, but later you will need to know how to dimension those. When **dimensioning a circle**, you must include **center lines**, **the radius or diameter dimension**, and the **location of the center**. The center line tools are found below for later reference.



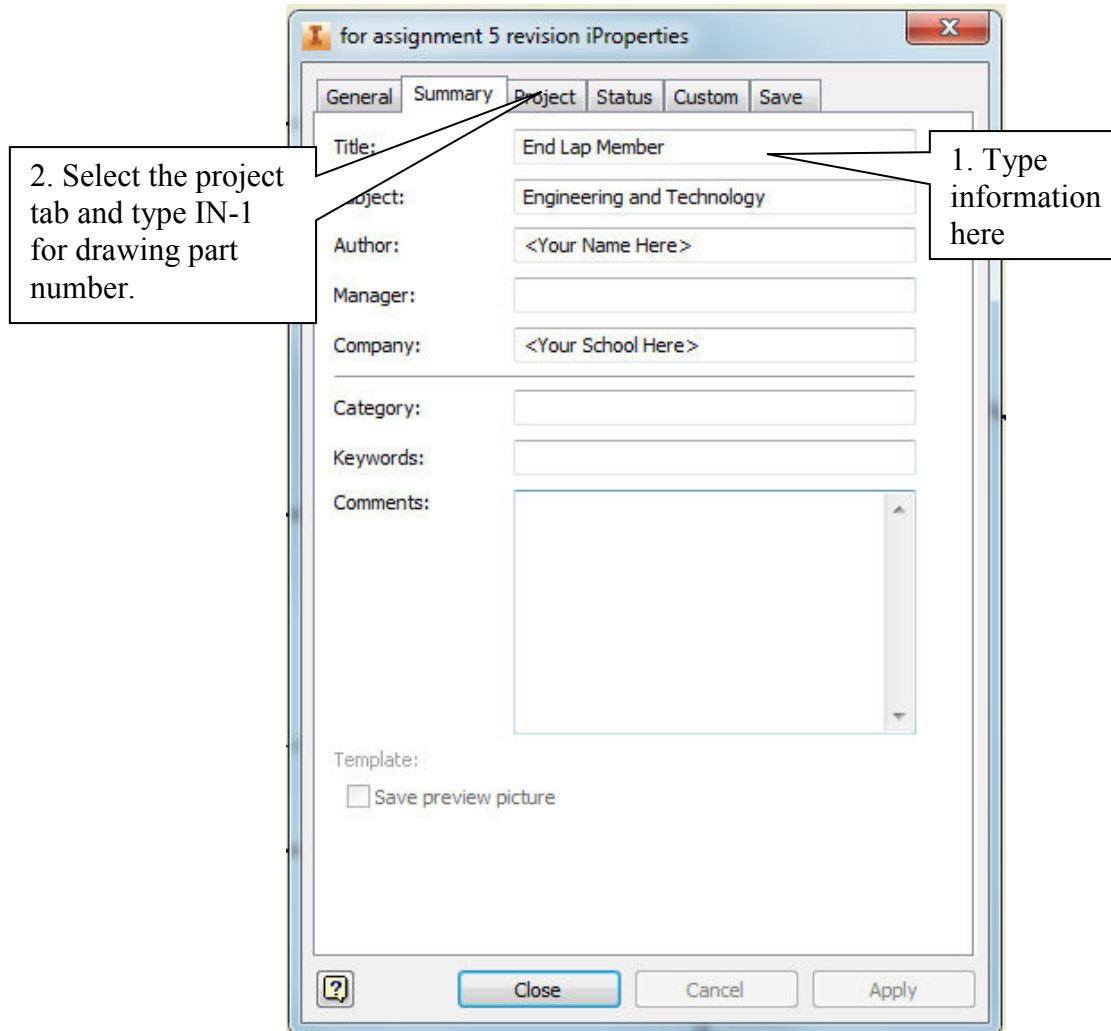
Click the center and then the circle.



To dimension the circle click the dimension button and find the green dot on the top of the circle then click it and find the center green dot and click it and bring out the dimension (for radius). To do a diameter dimension click any part on the circle and click again to confirm and apply the dimension.



6. The title block information needs to be completed before the drawing is considered finished. From the **File** menu select the **Properties** command. Fill in your information in the fields completed below. Before selecting OK, select the **Project tab** and number the file IN-1.



To Change an Inventor Drawing Title Block Information

- a. Click on the Inventor Icon in the upper left hand corner of the screen
Select iProperties
- b. Select each of the following Tabs and make changes
 - Summary Tab
 - Title.....Changes **TITLE**
 - Author.....Changes **DRAWN**
 - Company.....Changes space above **TITLE**
 - Project Tab
 - Part NumberChanges **DWG NO**
 - Revision Number.....Changes **REV**
 - Status Tab
 - Eng. Approved ByChanges **APPROVED**
 - Checked By..... Changes **CHECKED**



8. When the title block is completed, it should appear as illustrated:

DRAWN <Your Name Here>	6/11/2002	<Your School Here>		
CHECKED		TITLE		
QA		End Lap Member		
MFG				
APPROVED				
		SIZE C	DWG NO IN-1	REV
		SCALE		SHEET 1 OF 1

9. Now go back and do this for all of the other 17 drawings also. This was only IN-1 but you have to do this for the other 17 drawing as well. Follow these steps for those 17 as well.

What Do You Know?

Load Assignment 5 of module 81.40 and answer the following questions:

- What is the file name for creating an engineering drawing?
 - Standard.ipt
 - Standard.idw
 - Engineering Template
 - IN-7
- What is the name of the icon used to import the two-dimensional drawing into the engineering drawing?
 - Create View
 - New View
 - Projected View
 - Insert View
- When the view is created, what icon is do you select to create the top view and the right side view?
 - Create View
 - New View
 - Projected View
 - Insert View
- What was the last view you created called?
 - Left Side
 - Top
 - Iso Top Right
 - Right Side
- What toolbar is used for dimensioning the drawing?
 - Dimension
 - Standard
 - Object
 - Drawing Annotation



6. Which of the following is **NOT** one of the three rules to follow when dimensioning?
 - a. Place dimension between views
 - b. Shorter dimensions are placed inside over-all dimensions
 - c. Dimensions should only be placed one time
 - d. Dimensions should be at least 10 pt. font

7. Where can you go to edit the title block in a drawing?
 - a. File menu – Properties
 - b. Format – Title Block
 - c. Insert – Name
 - d. Edit – Title Block

8. What option is used to scale the drawing and title block to print on 8 1/2 x 11 paper?
 - a. Model 1:1
 - b. Best Fit
 - c. Custom
 - d. Current Window