Assignment 1

Inventor

Designing With Solid Modeling

Objectives

**When you complete this assignment you will:**

1. Set-up menus and drawing for designing modeling problems.
2. become familiar with the Sketch menu tools and commands.
3. Produce a three-dimensional model using the Features toolbar.

Getting Started

As mentioned in the introduction to the module, this is not your father's method of designing with CAD. The approach for creating a design is completely different from the methods of using two-dimensional AutoCAD. You will draw a 2-d sketch without concerning yourself to the exact size for the model. A dimension tool and constraints will be applied to the model to attain the exact sizes that will be desired for the model.
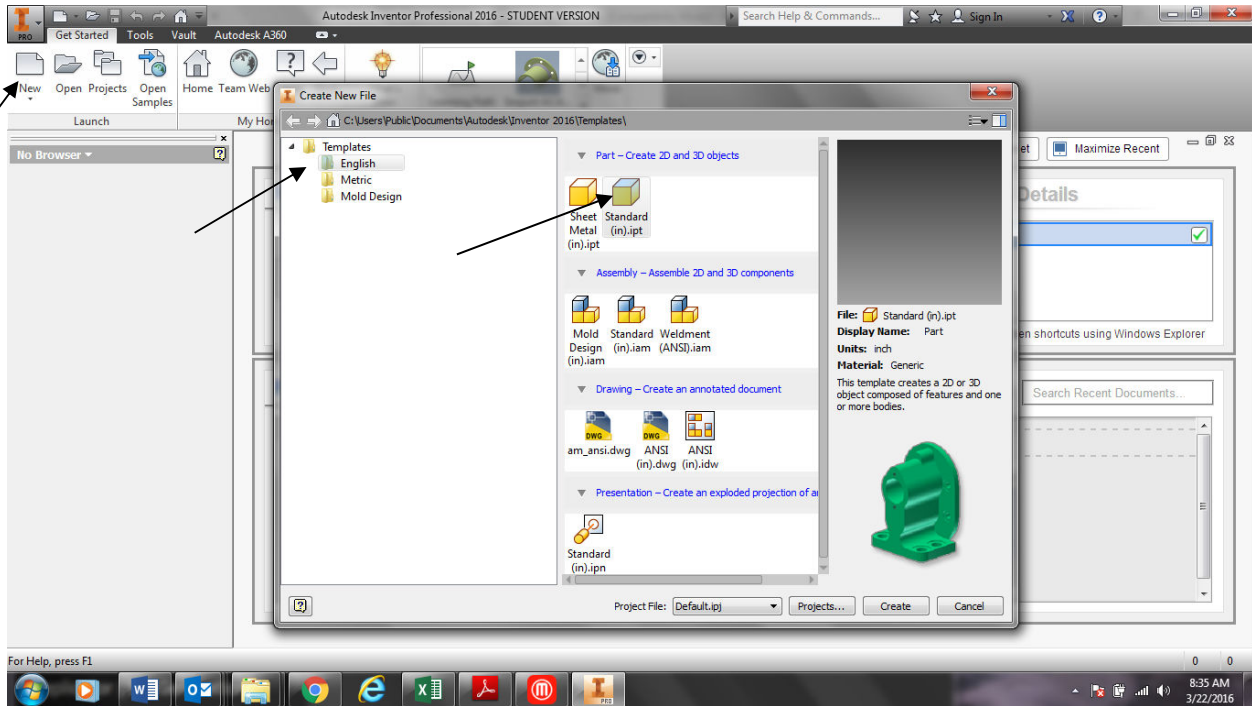
Inventor
1. Open Inventor
   a. Start
   b. Autodesk

   c. Select Autodesk
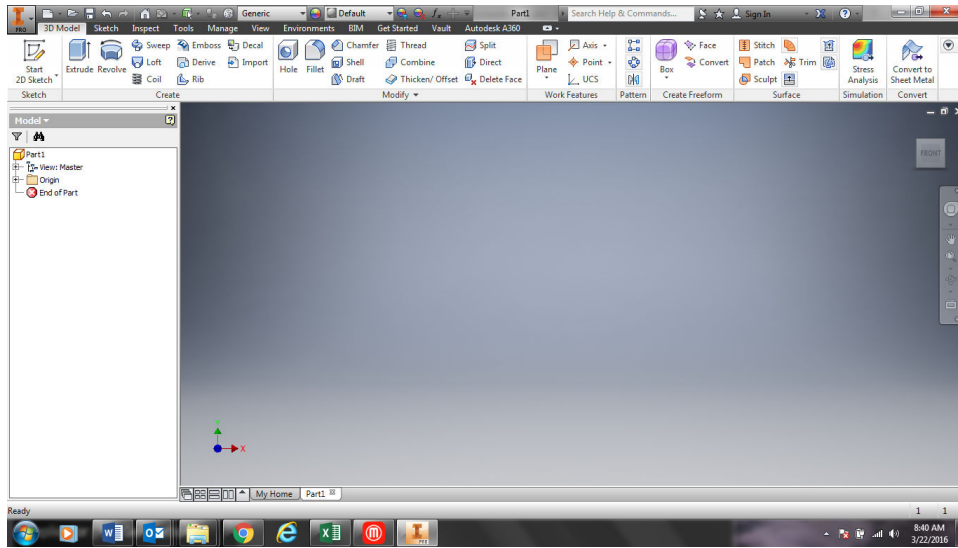
   d. Select Autodesk Inventor Professional 2016 – English
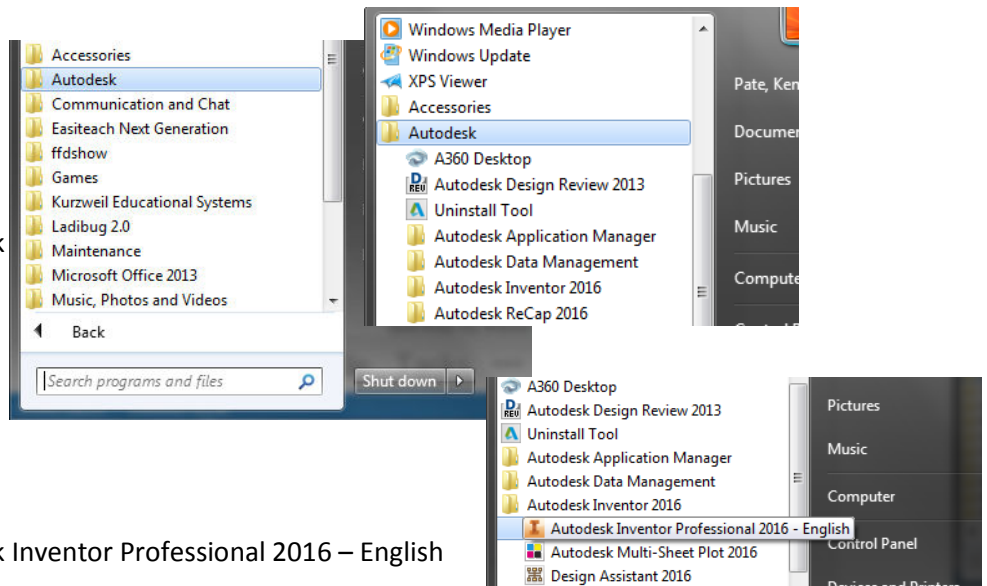
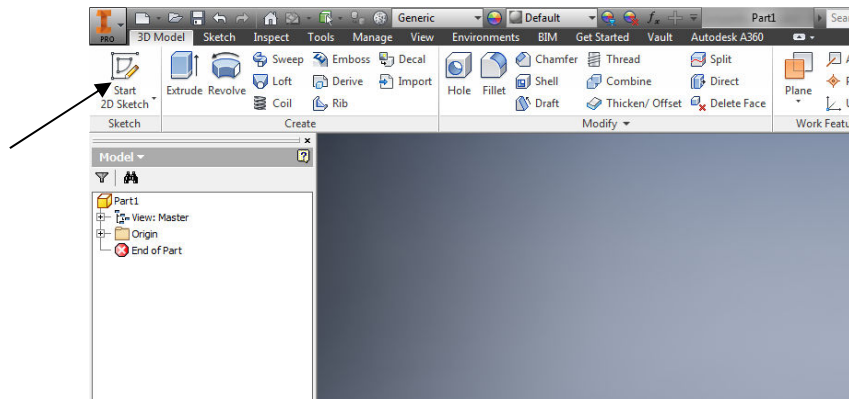2. Start a new part
   a. Select new, English, .ipt part
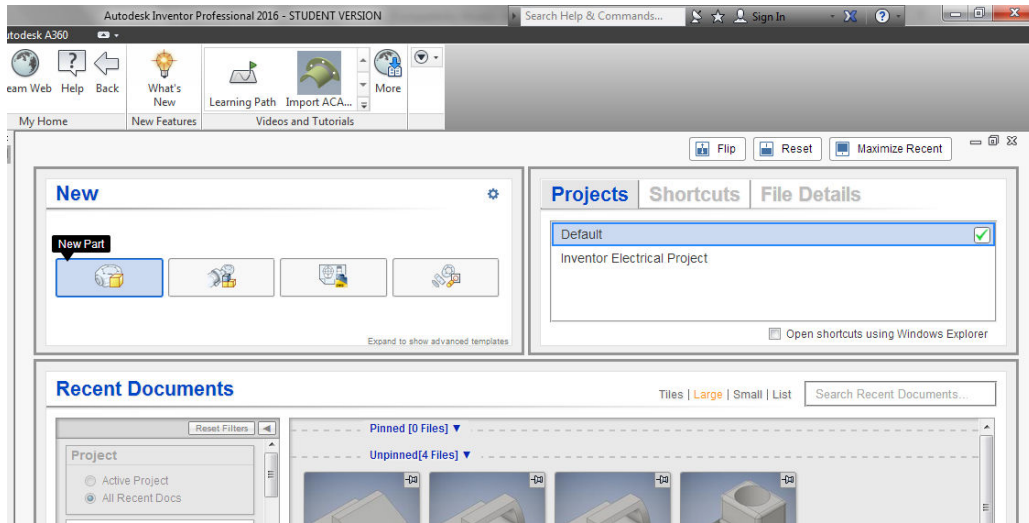
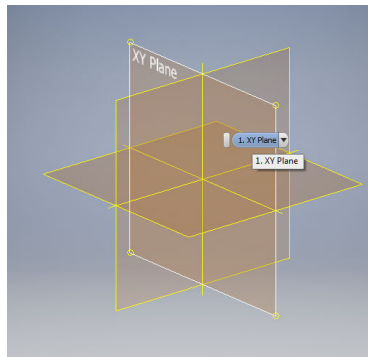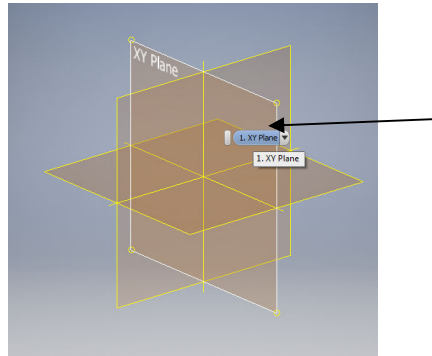3. Or…..

4. You will see this screen.

5. **Select Start 2D Sketch….**
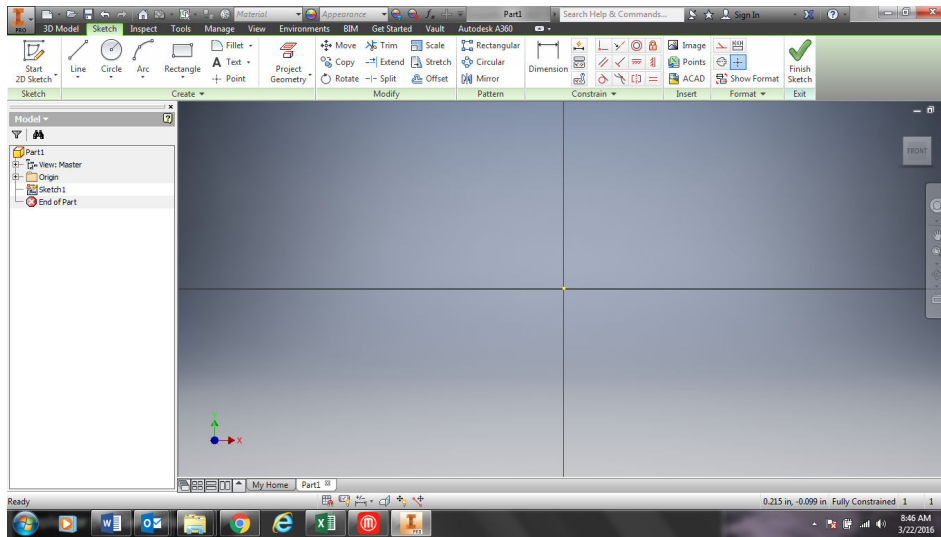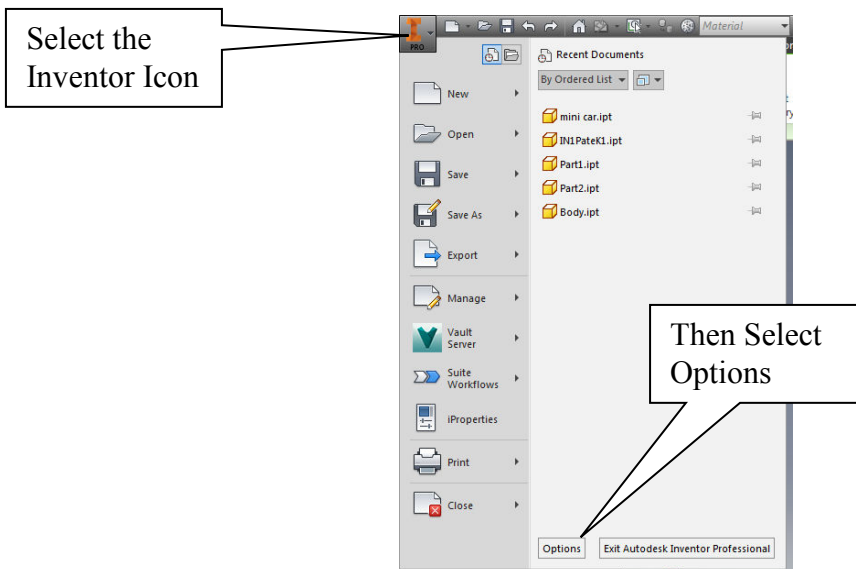

6. You will see this……

7. Click on the x,y plane.

8. Now the screen will look like this….

9. To turn view the grid, Select the Inventor Icon and select Options.

Select the
Inventor Icon

Then Select
Options

10. Click on the Sketch tab and then check Grid, Apply and close.

Select the
Sketch Tab

Check Grid
Lines
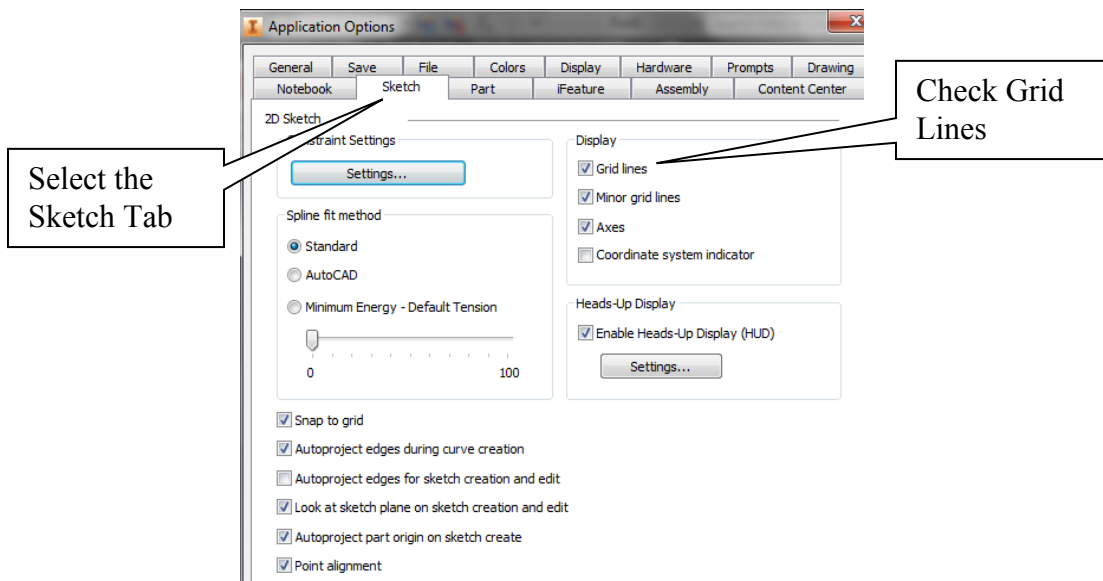
11. **Right click** on the **design window** and choose the **Snap to Grid** option. This
will give you more control as you draw later on. **Remember to perform this
step each time you begin a new drawing if it has been turned off.** A check
mark will be added if snap is on.
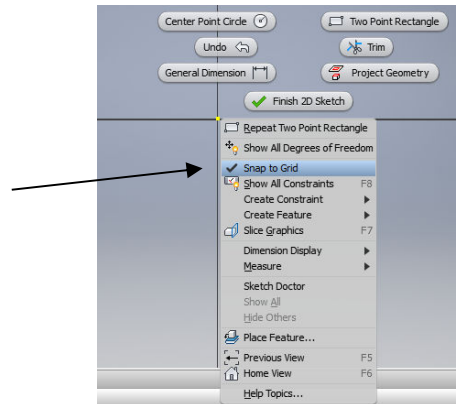
Using the Sketch Toolbar

1. Select the **line tool** from the Sketch menu. When you move the cursor to the drawing screen, you will notice that a small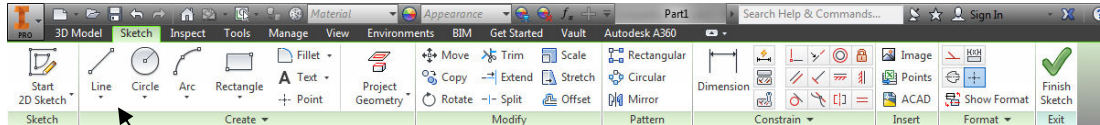 dot represents the first insertion point of a line segment. In AutoCAD you had a crosshair insertion point. When you left-click the mouse, this will begin the first point for a line segment.

2. Move the insertion dot to an area of the drawing grid as illustrated below and left-click the mouse. This will begin the first point of the line segment. Move the mouse horizontally two squares to the right. Left-click the mouse at this point. You have now drawn a line segment. In inventor, you will not be concerned with the exact size until you complete the basic shape.
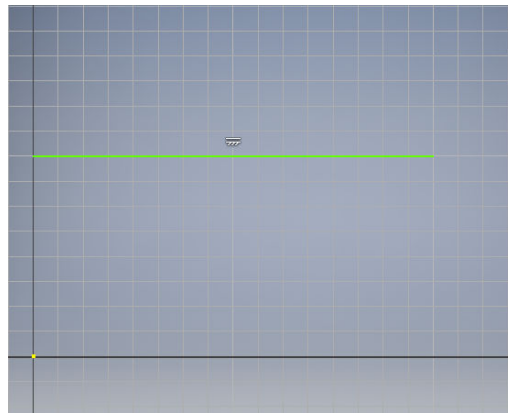
3. Continue drawing until you complete a rectangle that is two large squares by one large square in size.
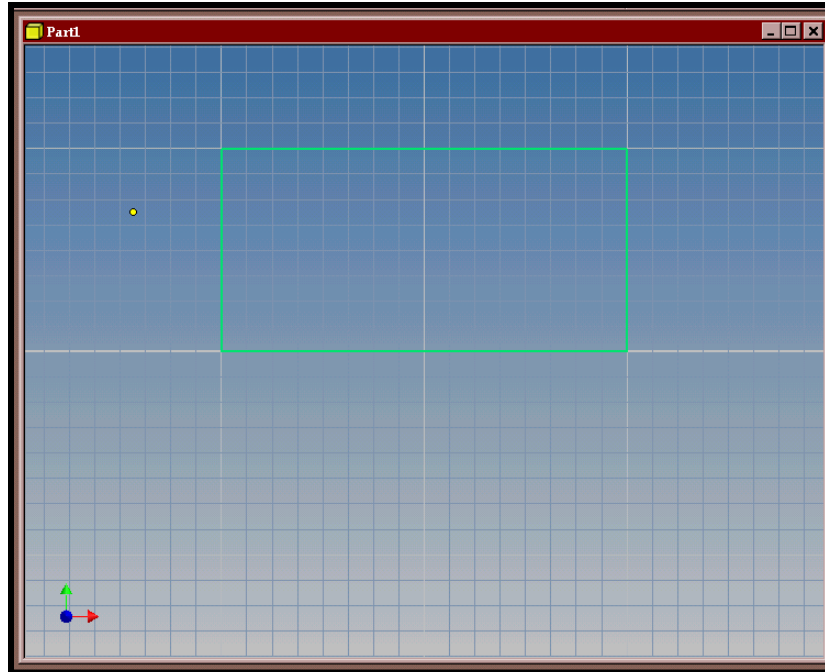

4. Left-click to close the rectangle, then right-click and a dialogue box will appear. Select done to complete the rectangle.

5. In the next step, you will begin the process of adding dimensions to the drawing. Dimensions on the sketch can be altered at any time.

6. Select the **general dimension** tool from the **Sketch toolbar**.
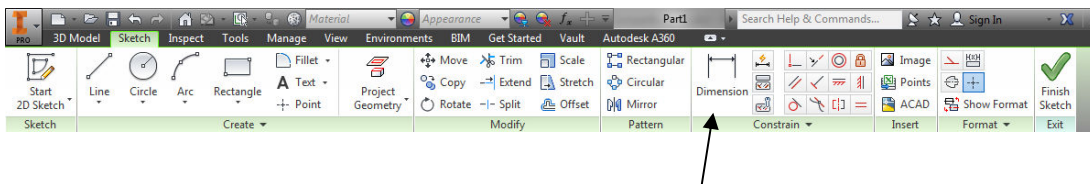
7. After selecting the dimension icon move the mouse to the top horizontal line of the rectangle that you have drawn. Left-click the line it will appear red in color. Move the mouse above the line and left-click again. The dimension indicates that the line is 2" long.
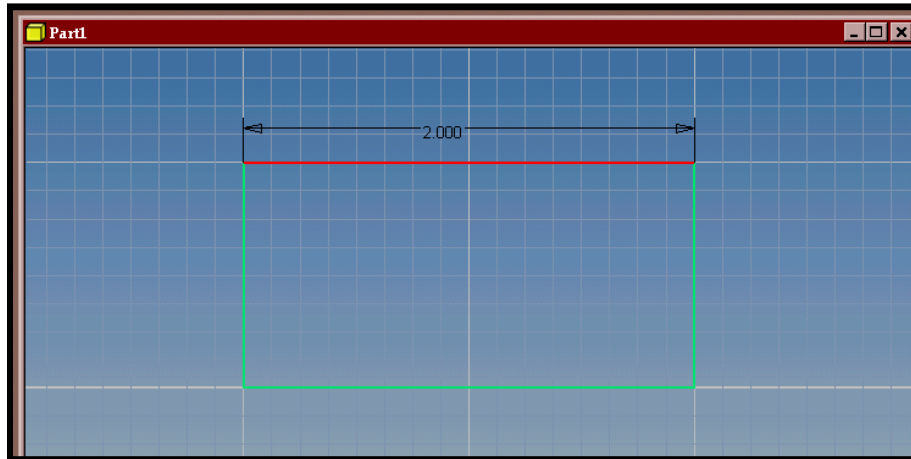
8. Continue by dimensioning the left vertical line.
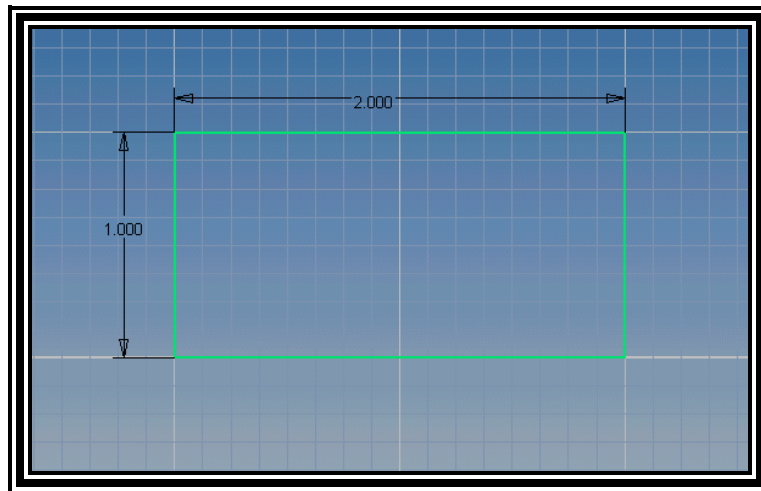
9. You can change the dimension size of the rectangle by clicking onto the number of the dimension, an edit dimension dialogue box appears. You will change the sizes in this box.

Change from 1 to 2 and press the enter key or click the green check.
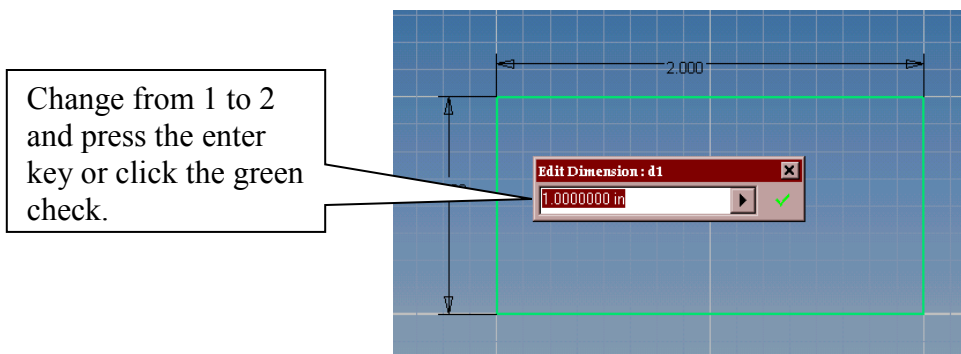
10. You will notice that the rectangle is now a 2 by 2 square.
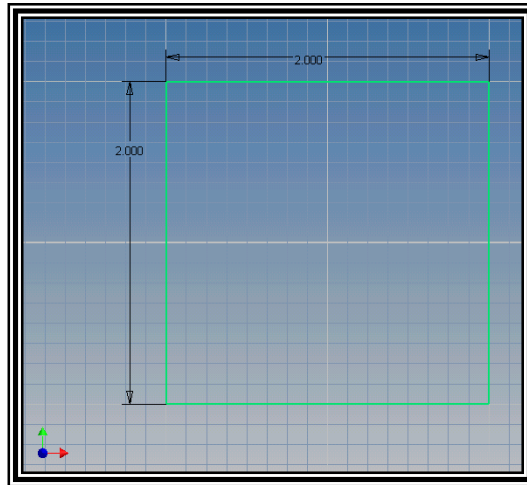
11. Change the horizontal to 6 from 2. You will notice that the complete image of the rectangle does not fit the screen.
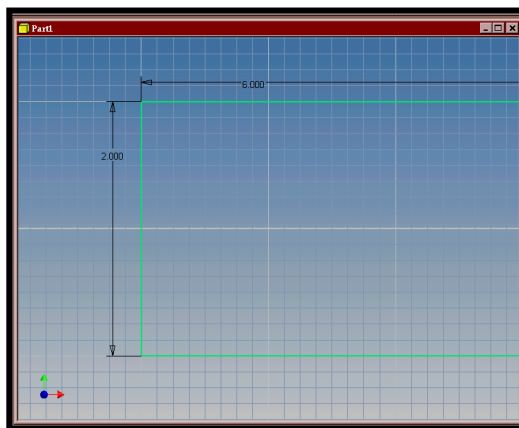
12. You will now use the **zoom all** icon from the Standard toolbar. When you click the icon, the drawing image will then fit into the screen.

Select
Zoom All

12. The image now fits the screen. **Right-click** in the middle of the drawing and select **Finish 2D Sketch.**

13. **Right-click** again and select **Finish Sketch**. Finally, **select view all**. The image will then be slanted into an isometric view as illustrated:
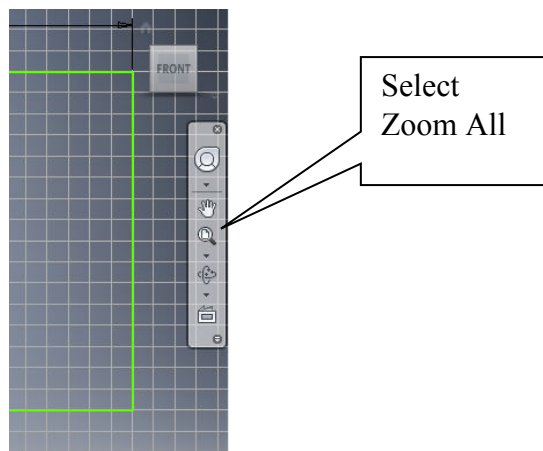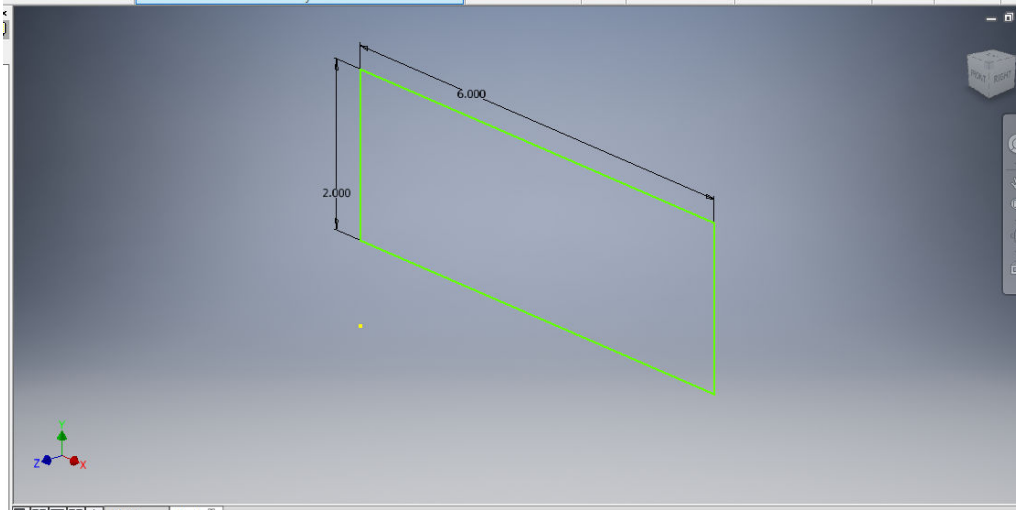

Using the Features Toolbar

1. The **extrusion tool** is used to change two-dimensional shapes into three-dimensional models. Currently the model is merely a rectangle that is 2" by 6". We are going to add depth to the model in the next step.
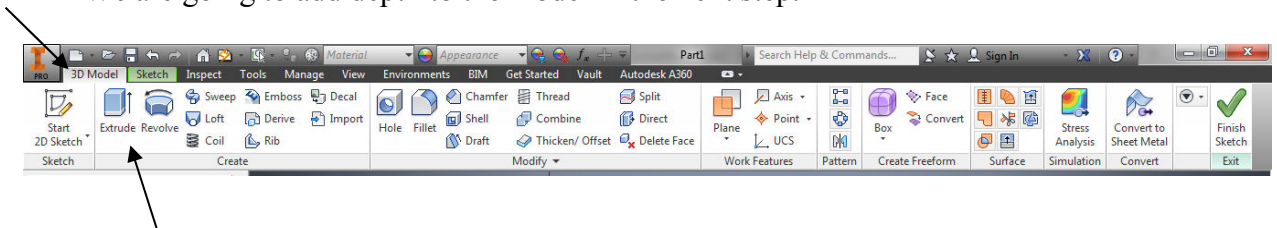
2. When you click the extrusion tool icon a dialogue box appears. You will need to make changes in box.

Change to 3"

Choose Direction 2.
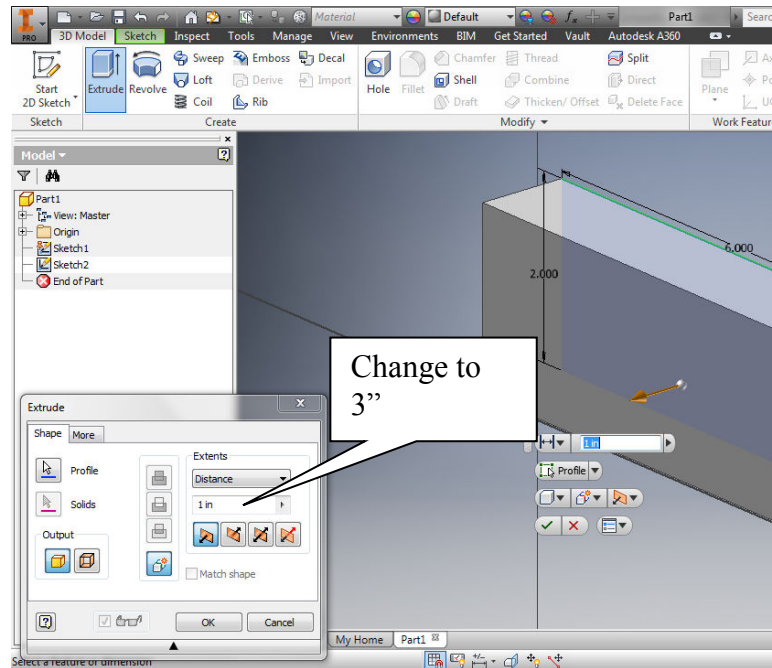
3. When you click OK you will now have a three-dimensional model created. It will not fit into the view entirely. You will need the **zoom all** icon for the model to fit.
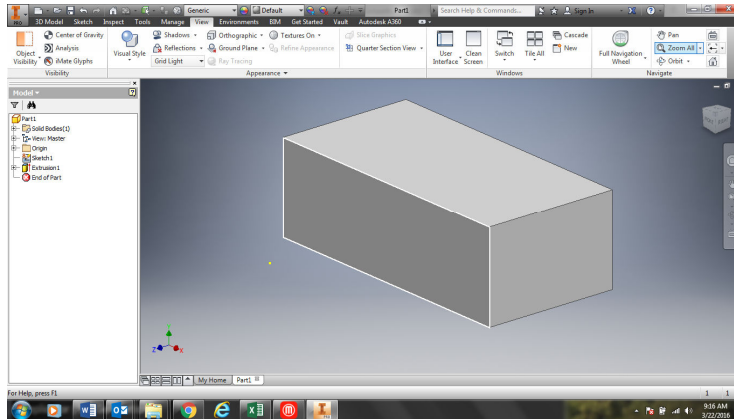
Saving the Model

1. From the file menu, select the save command. Navigate to your server folder and **save the file as IN-1(Last Name, First Initial) (Pd)**.

2. The steps taken to create the simple block that is 2" x 6" x 3" will be done in many of the assignments in this module. All of the problems will begin using these same dimensions. Activity 1 can be used as a guide for the beginning of all assignments until mastery is attained.

Conclusion

You have now completed Assignment 1. In this assignment you learned the basic tools and commands that are needed to create a simple three-dimensional model. The steps that were used in this activity will be repeated in many of the upcoming problems and assignments. The problems that were drawn in the Orthographic Module will be used in the Inventor Module. Repetition will breed success.

What Do You Know?

Load Assignment 1 of module 81.40 and answer the following questions:

1. What is the name of the template file used to begin a new modeling file in Inventor?

    a. Standard.ipt
    b. IN-1
    c. Standard.idw
    d. My Template

2. What is the name of the toolbar used to draw lines and create basic shapes?

    a. Features
    b. Sketch
    c. Draw
    d. Model

3. What tool was used to draw the 1" x 2" rectangle?

    a. Center Point Circle
    b. Three-point Arc
    c. Rect. Edit
    d. Line

4. What tool was used to change the size of the rectangle?

    a. Line
    b. Dimension
    c. Rect. Edit
    d. Format

5. What tool was used to make the edited rectangle fit the screen?

    a. Best Fit
    b. Dimension
    c. Zoom All
    d. Isometric View

6. What is the name of the view that changes the drawing from a two-dimensional to a three-dimensional view?

    a. Previous
    b. Look
    c. Standard.idw
    d. Isometric

7. To add a depth of 3" to the model, you used this tool.

    a. Extrude
    b. Model
    c. Cut
    d. Extend

8. What should you have saved your first Inventor file as?

    a. My Template
    b. IN-1
    c. Standard.idw
    d. (Your name)-(period)-1