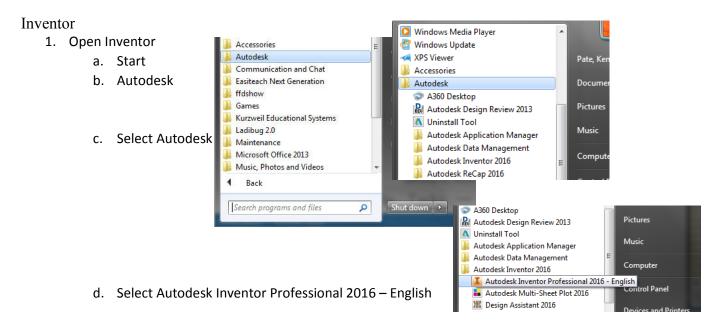


When you complete this assignment you will:

- 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.

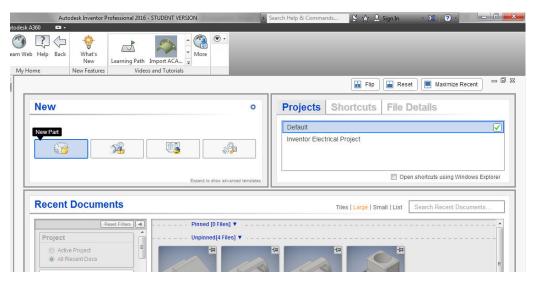


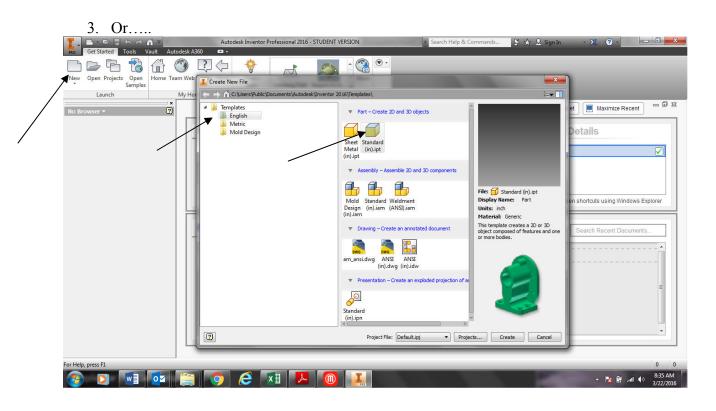
As mentioned in the introduction to the module, this is not your father's method of designing with CAD. The approach for creating a design is completely different from the methods of using two-dimensional AutoCAD. You will draw a 2-d sketch without concerning yourself to the exact size for the model.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



2. Start a new part

a. Select new, English, .ipt part





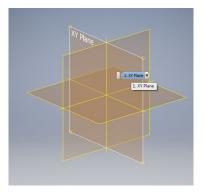
4. You will see this screen.

	🗁 🔚 🕤 n ladel Sketch		Tools Mar			Default	\checkmark \bigcirc \bigcirc \bigcirc $f_x =$ Get Started Vault	Autodesk A360	1	Search Help	o & Comn	nands	\$ ★	Q _. Sign In	- 20	? ·	-0	×
Start 2D Sketch	Extrude Revolve	Goil	Emboss	Decal	Hole Fille	Chamfer	r 📳 Thread 🚰 Combine 🥪 Thicken/ Offse	Split	Plane	✓ Axis • ♦ Point • ↓ UCS	22 00 010	Box	✤ Face⋧ Convert	🚳 Sculpt	≫ Trim 🕼	Stress Analysis	Convert to Sheet Meta	● •
Sketch		Cre					Modify *		Wor	k Features	Pattern	Crea	te Freeform	S	urface	Simulation	Convert	
Model - Parti - Dram - Origin - Origin - Origin			ate				initiany -		wor	r reatures	Pattern	Crea	(c rreetorm	3	unace	Simulation		
				► ×	Home Part	1 20												
Ready	-							_	-	_	-	-	_	-	_		1	
1		0		Q	<u>e</u>	×1 🧕								-	-	🍓 🕼 .all	() 8:40 3/22/	2016

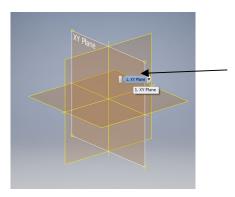
5. Select Start 2D Sketch....

	🖻 🗖 🖶 🔿 👘 Iodel Sketch Inspec		and the second se	Finvironmen	Default ts BIM	\checkmark \bigcirc \bigcirc f_x $+$ Get Started Vault	Part1 Autodesk A360	63 -	Searc
Start 2D Sketch Sketch	Extrude Revolve	eep Emboss t 🔂 Derive		Hole Fillet		er 📳 Thread ightarrow Combine ightarrow Thicken/ Offset Modify 👻	Split		Pa ◆ Pa ↓ U Featur
Model - T A Parti the Tay Verwer the Other Send of		× 2							

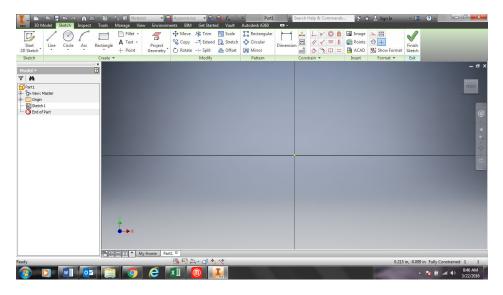
6. You will see this.....



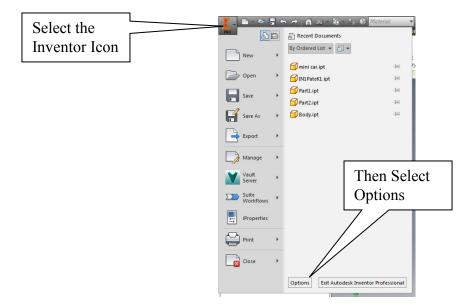
7. Click on the x,y plane.



8. Now the screen will look like this....

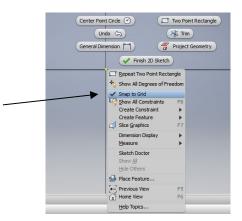


9. To turn view the grid, Select the Inventor Icon and select Options.



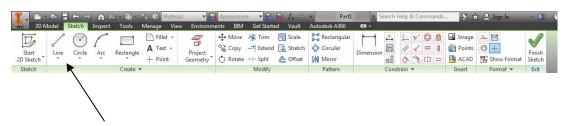
- L Application Options X General Save File Colors Display Hardware Prompts Drawing Check Grid Notebook Sketch Part iFeature Assembly Content Center 2D Sketch Lines raint Settings Display Grid lines Select the Settings... Minor grid lines Sketch Tab Spline fit method Axes Standard Coordinate system indicator AutoCAD Heads-Up Display Minimum Energy - Default Tension Enable Heads-Up Display (HUD) Settings... 0 100 🔽 Snap to grid Autoproject edges during curve creation Autoproject edges for sketch creation and edit V Look at sketch plane on sketch creation and edit Autoproject part origin on sketch create V Point alignment
- 10. Click on the Sketch tab and then check Grid, Apply and close.

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.



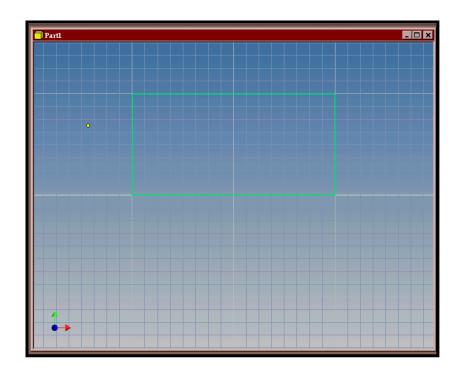
Using the Sketch Toolbar

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 leftclick 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.

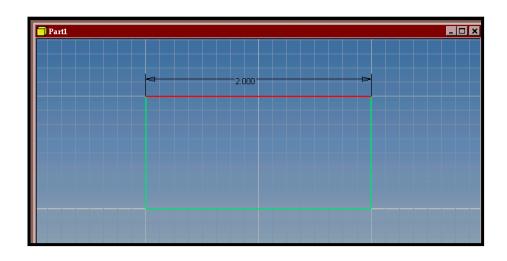
3. Continue drawing until you complete a rectangle that is two large squares by one large square in size.



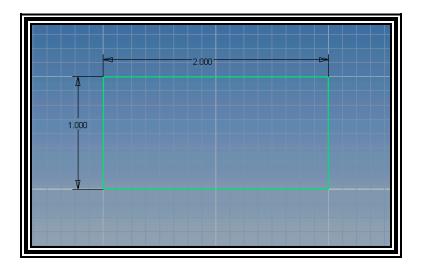
- 4. Left-click to close the rectangle, then right-click and a dialogue box will appear. Select done to complete the rectangle.
- 5. In the next step, you will begin the process of adding dimensions to the drawing. Dimensions on the sketch can be altered at any time.
- 6. Select the general dimension tool from the Sketch toolbar.



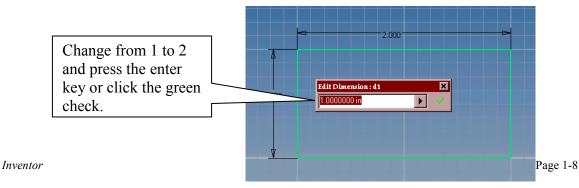
7. After selecting the dimension icon move the mouse to the top horizontal line of the rectangle that you have drawn. Left-click the line it will appear red in color. Move the mouse above the line and left-click again. The dimension indicates that the line is 2" long.



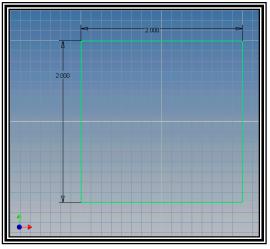
8. Continue by dimensioning the left vertical line.



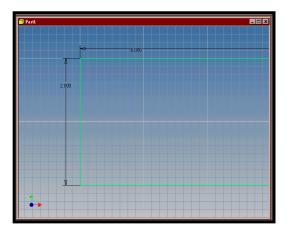
9. You can change the dimension size of the rectangle by clicking onto the number of the dimension, an edit dimension dialogue box appears. You will change the sizes in this box.



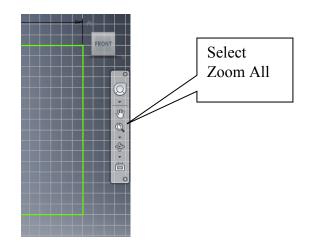
10. You will notice that the rectangle is now a 2 by 2 square.



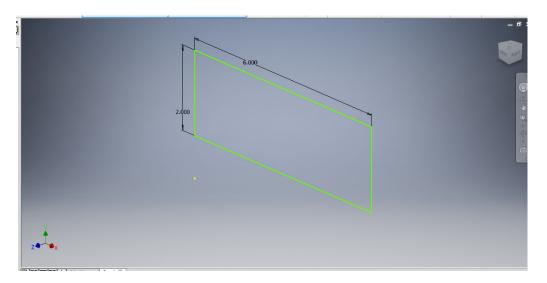
11. Change the horizontal to 6 from 2. You will notice that the complete image of the rectangle does not fit the screen.



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.



- 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 threedimensional 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.

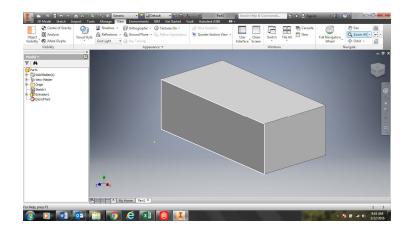
Start 2D Sketch	Extrude Revolve	😽 Loft	The second secon		100	Split Direct	Plane	Axis +	**	Box Co			Convert to Sheet Metal	• ک	Finis Sketc
Sketch	R	Crea	te		Modify -		Wo	rk Features	Pattern	Create Freefo	rm Surface	Simulation	Convert		Exit

2. When you click the extrusion tool icon a dialogue box appears. You will need to make changes in box.

📘 🖿 🗁 🖥 🖘 🖻 💁 🔞 - 🧐			Default	- 🚭 🚭			Search
	anage View	Environmen	ts BIM	Get Started	Vault	Autodesk A360	63 -
The sweep Sweep Sweep Sweep	s 🛃 Decal		Chamfe	r 🗐 Thread		Split Split	Axis
Start Extrude Revolve Loft Derive	E Import	Hole Fillet	D Shell	🕼 Combi	ne	Direct	Plane 🔶 Poir
2D Sketch		THORE THREE	Draft	Thicker	n/ Offset	${\mathscr D}_{\rm K} {\rm Delete} {\rm Face}$	• L UCS
Sketch Create				Modify 🕶			Work Features
Model -							
Y H				-	No.		
Part1						_	
Part1			F	\leq			
							~
- Ketch 1						-	6.000
End of Part				2.000			
- G End of Part							
		C1					
		Chang	ge to			-	
	x	3" -				A	
Extrude	~)		and the second se			
Shape More		-			v In	D	
Extents						The second se	
Profile Distance				1.15	Profile 🔻	and the second division of the second divisio	
Solids 📋 1 in 🔸					• 😚 •		Concession of the local division of the loca
				-	×		The second s
Output 📃 📄 🔀 🕅	۲.			<u> </u>			
Match shape	_						
Match shape							
Cancel OK Cancel	el						
	My	Home Part1					
		F	H C */-	- 1 * *	ŀ		

Extrude	
Shape More Image: Profile Extent Image: Solids Image: Solids Output Image: Solids Image: Solids Image: Solids <t< td=""><td>Choose Direction 2.</td></t<>	Choose Direction 2.
	Cancel My Home Parti II III III IIII IIII IIIIIIIIIIIIIII

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.





- 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.



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.



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