

# Drafting Tutorial

---

Part one

Basics

# Objectives

## Understand:

- ❑ The basics of drafting using Autodesk Inventor

## Demonstrate:

- ❑ The ability to create a basic shape with features of some kind using the extrusion tool

# If you're familiar with the program

- ❑ If you have taken the time to learn the program on your own, or have taken a class, like DDP, and already know Inventor, feel free to skip over the tutorial.
- ❑ You will still have to show a mentor or the current drafting lead your completed cube, and the plate
- ❑ The cube should be 1x1x1, have a .25in channel on the front(centered), and a .25in hole in the center

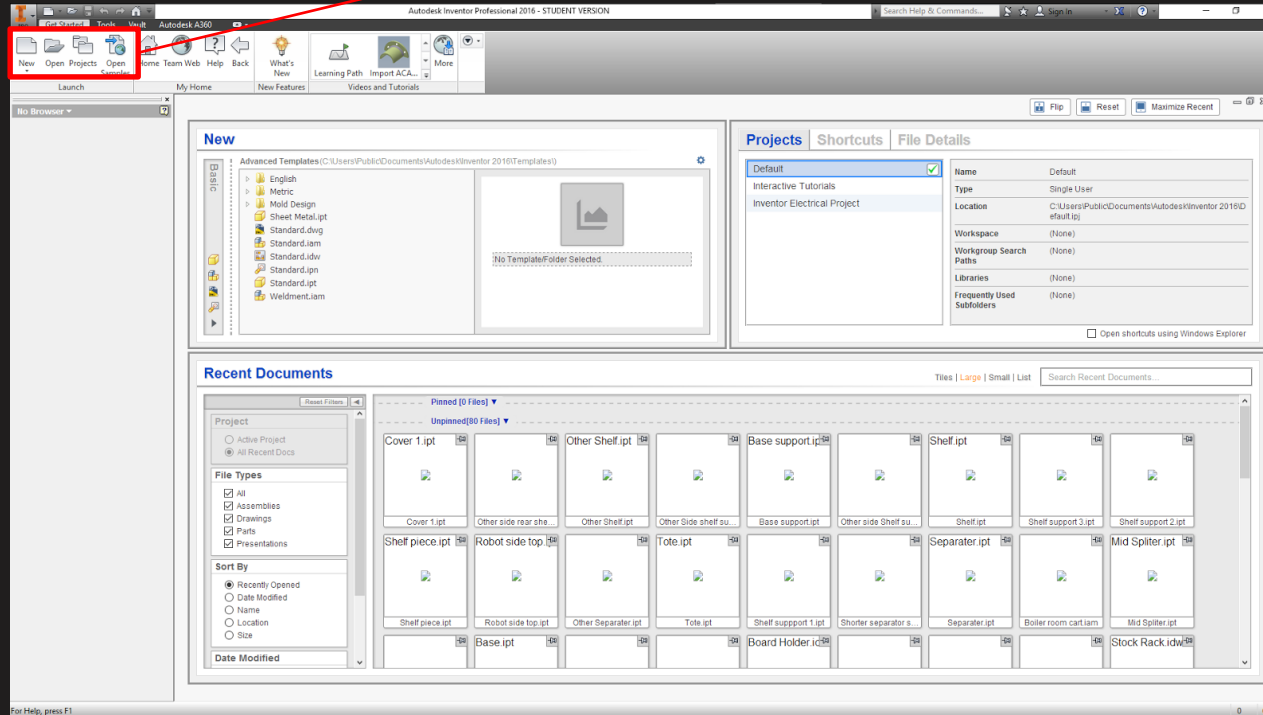
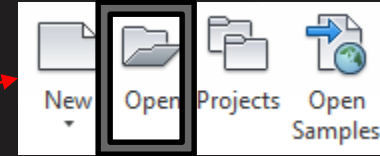
# Getting started

- ❑ To begin, open the program
- ❑ The shortcut will look like this.



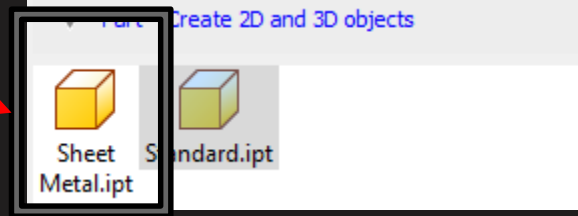
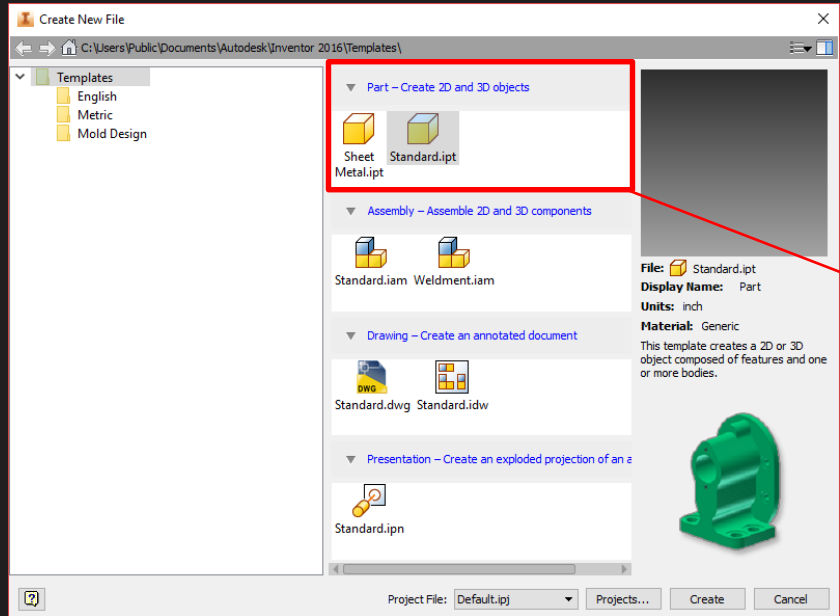
# Creating a new part

- ❑ Click on the icon under the launch tab with a blank piece of paper labeled "New".

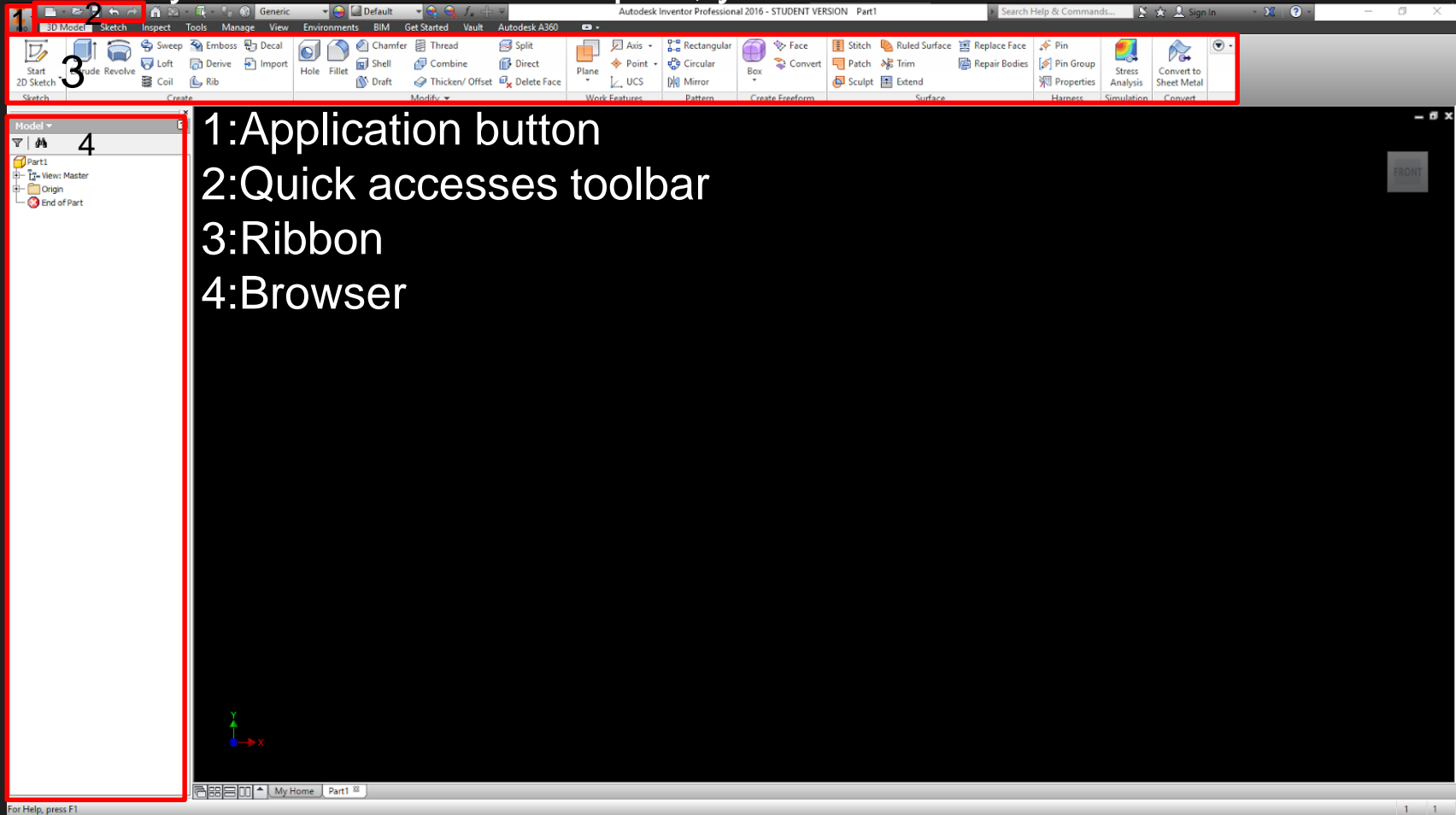


# File creation prompt

- ❑ This is the window where you can select the type to open.
- ❑ In the tutorial, we will be creating a new ipt, or a standard part

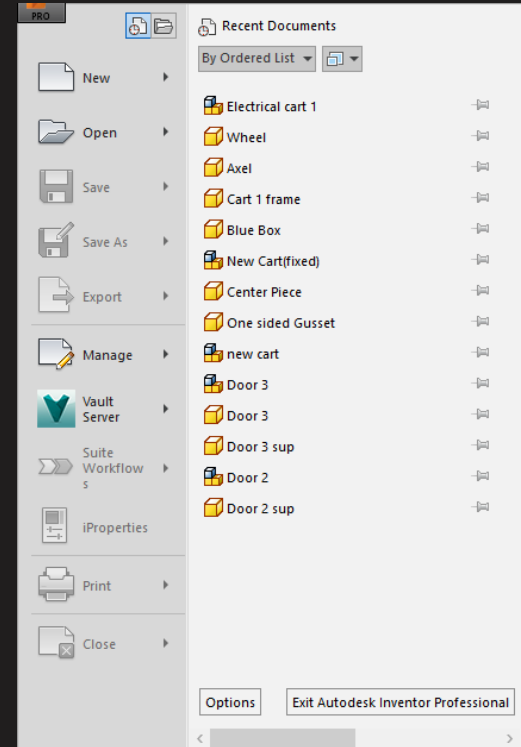


# When you first create a new part, your screen will look like this



# Application Button

- ❑ The Application Button is one of the many ways you can open, save, and create new parts and assemblies, and print
- ❑ It also allows you to access the options menu, but we will not cover that now but feel free to play around in it



# Quick Access Toolbar

Hotkeys:



New file

Ctrl + n



Open

Ctrl + o



Save

Ctrl + s



Undo

Ctrl + u

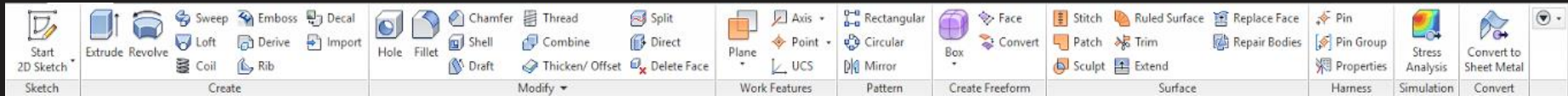


Redo

Ctrl + y

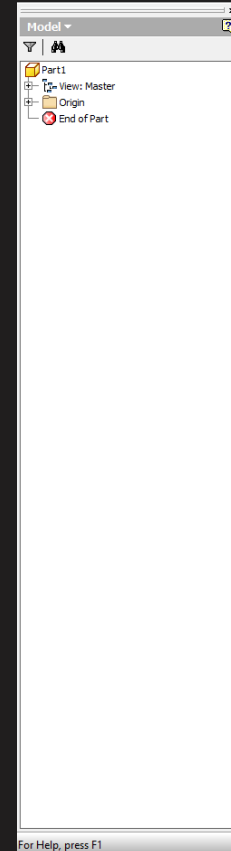
# Ribbon

- ❑ The Ribbon is where all of your of your modeling tools are

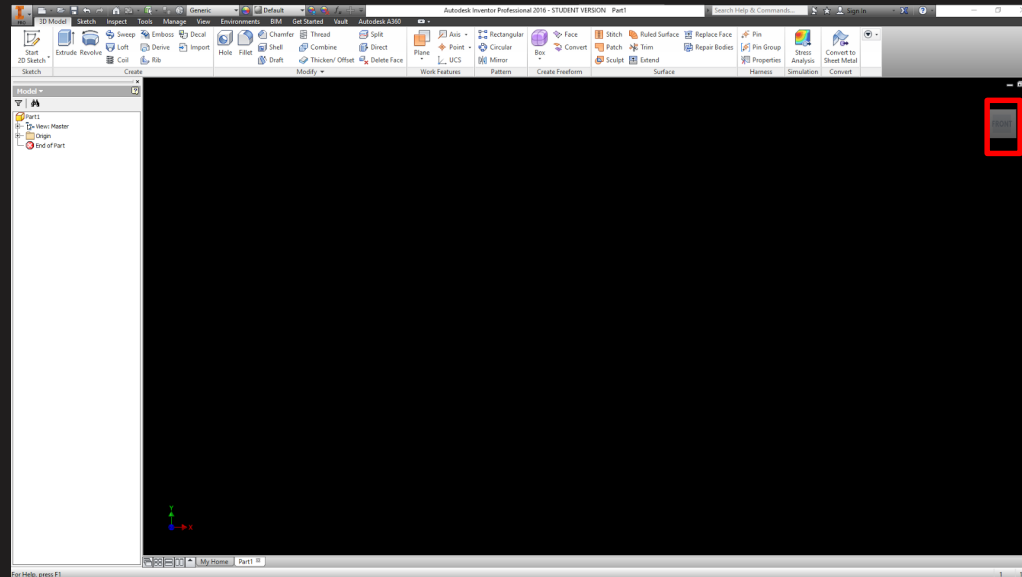


# Browser

- ❑ The browser shows a list of features( extrusions and revolutions)
- ❑ You can edit some dimensions from here

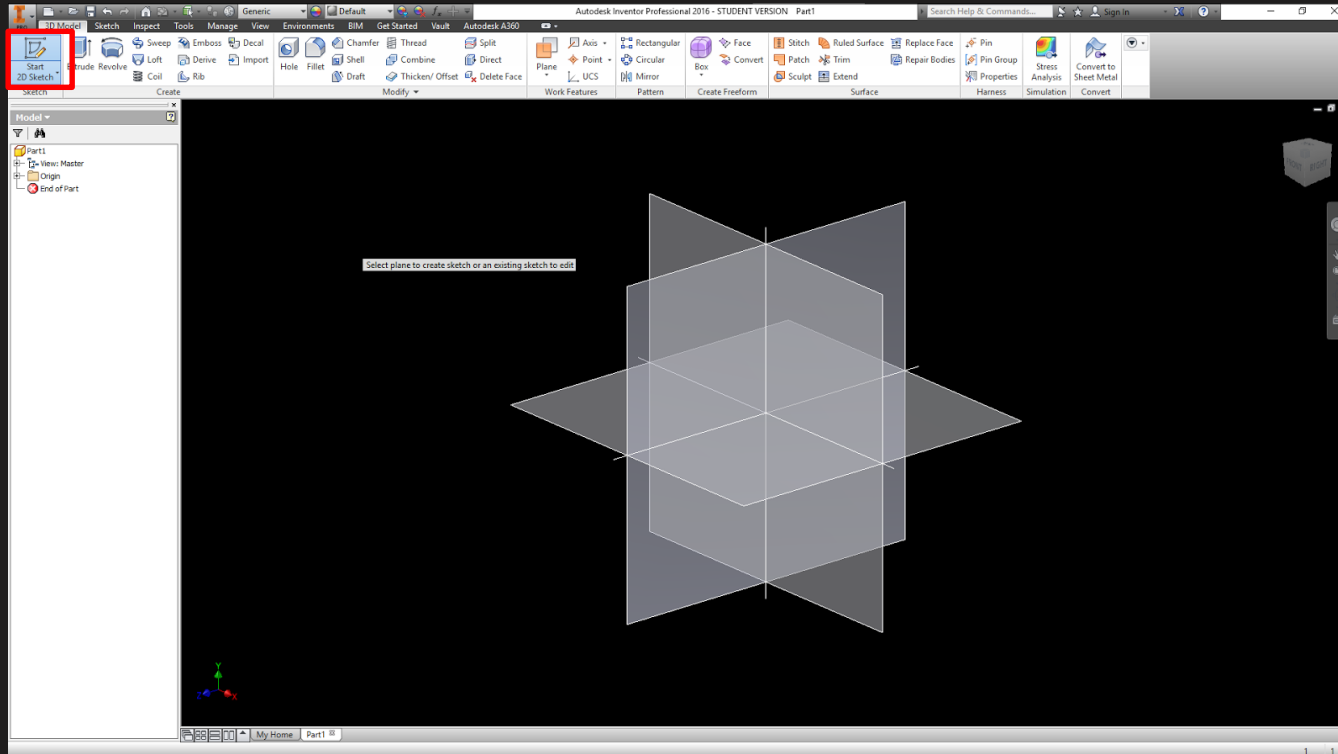


- ❑ This cube will allow you to rotate your part
- ❑ To change views, simply click on a face, corner, or edge
- ❑ Two arrows will appear, they will rotate your part clockwise or counterclockwise
- ❑ You can use the scroll wheel to zoom in and out
- ❑ You can click on the little house icon to return to the default view

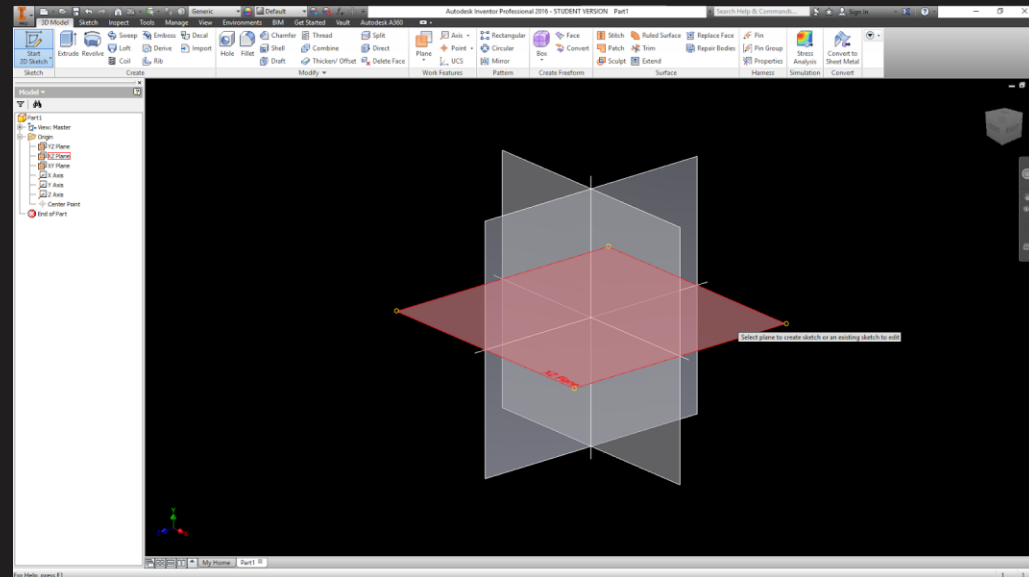


# Making a new part

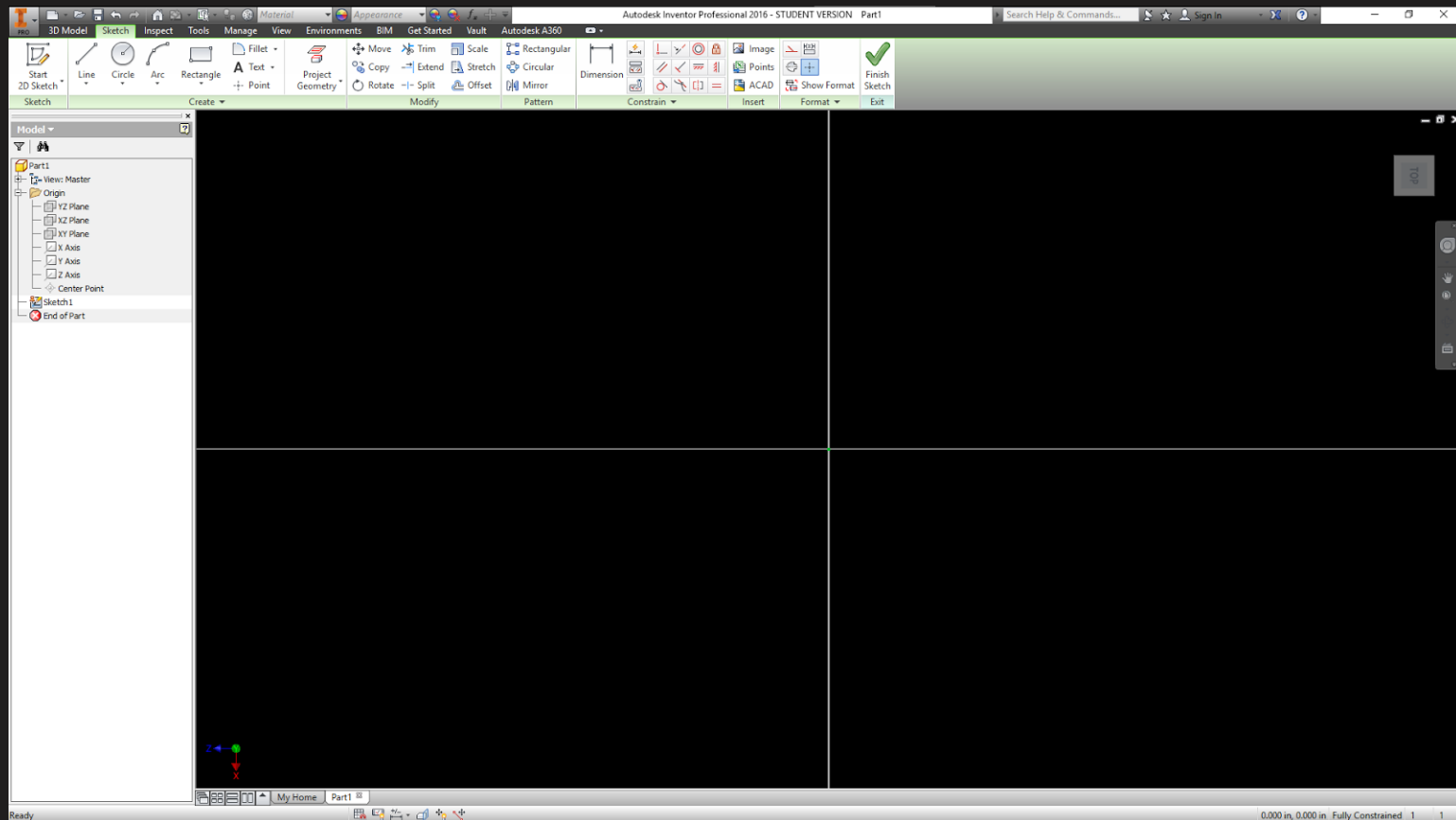
- ❑ To begin creating your part, click “Start 2D Sketch”



- ❑ Then you need to select a plane, of which there are three of
- ❑ For this tutorial, select XZ plane( shown below)

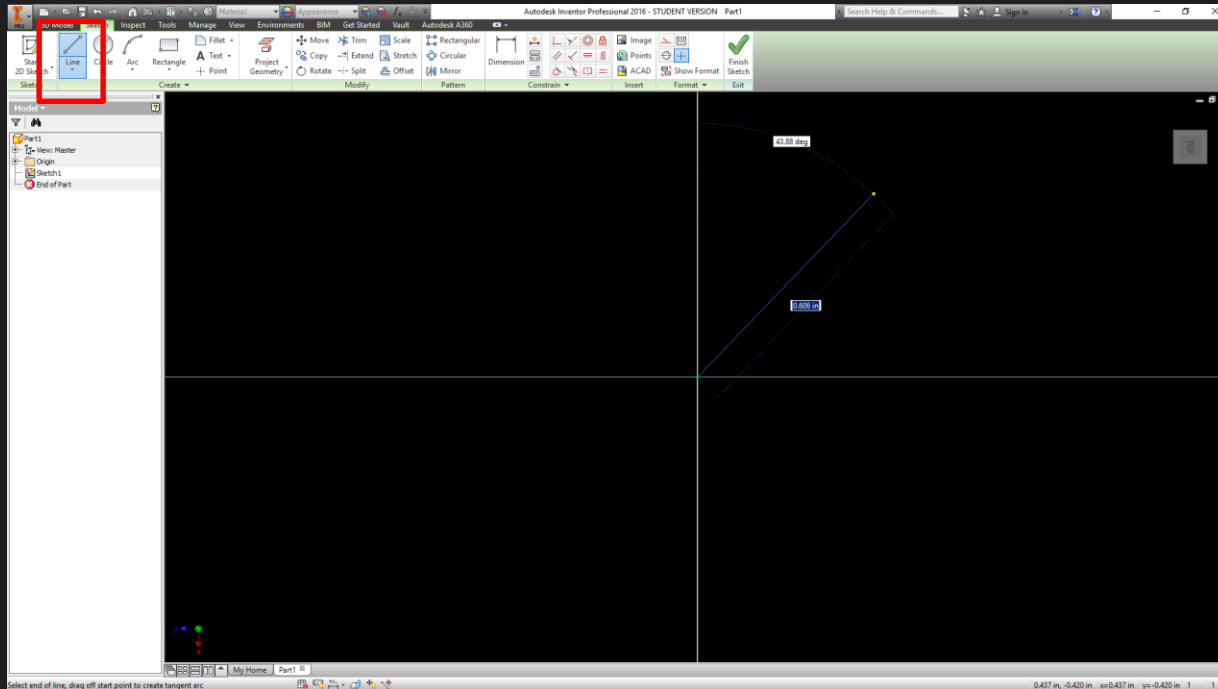


- ❑ From here you can create a sketch of your part
- ❑ We will now quickly go through all the sketching tools you will use most often



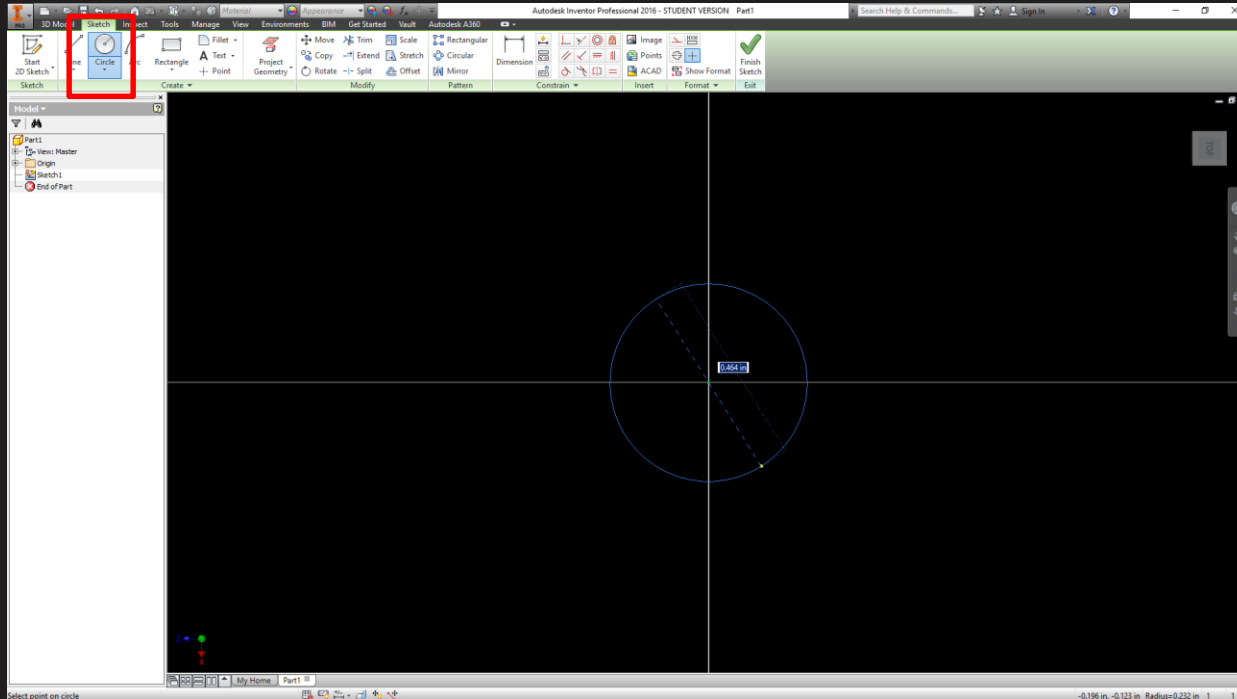
# Line

- The most simple of the tools, simply click once to place one end of the line, then again to place the 2<sup>nd</sup>.



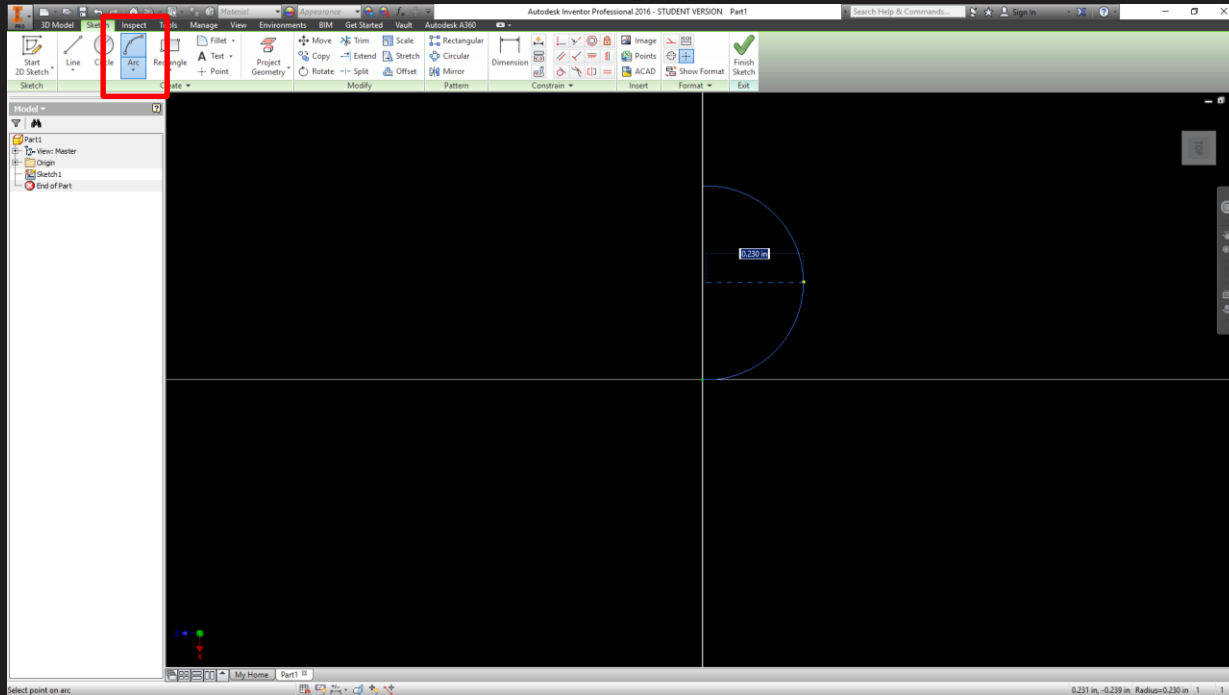
# Circle

- First click for the center point of the circle, then click again to set the diameter of the circle



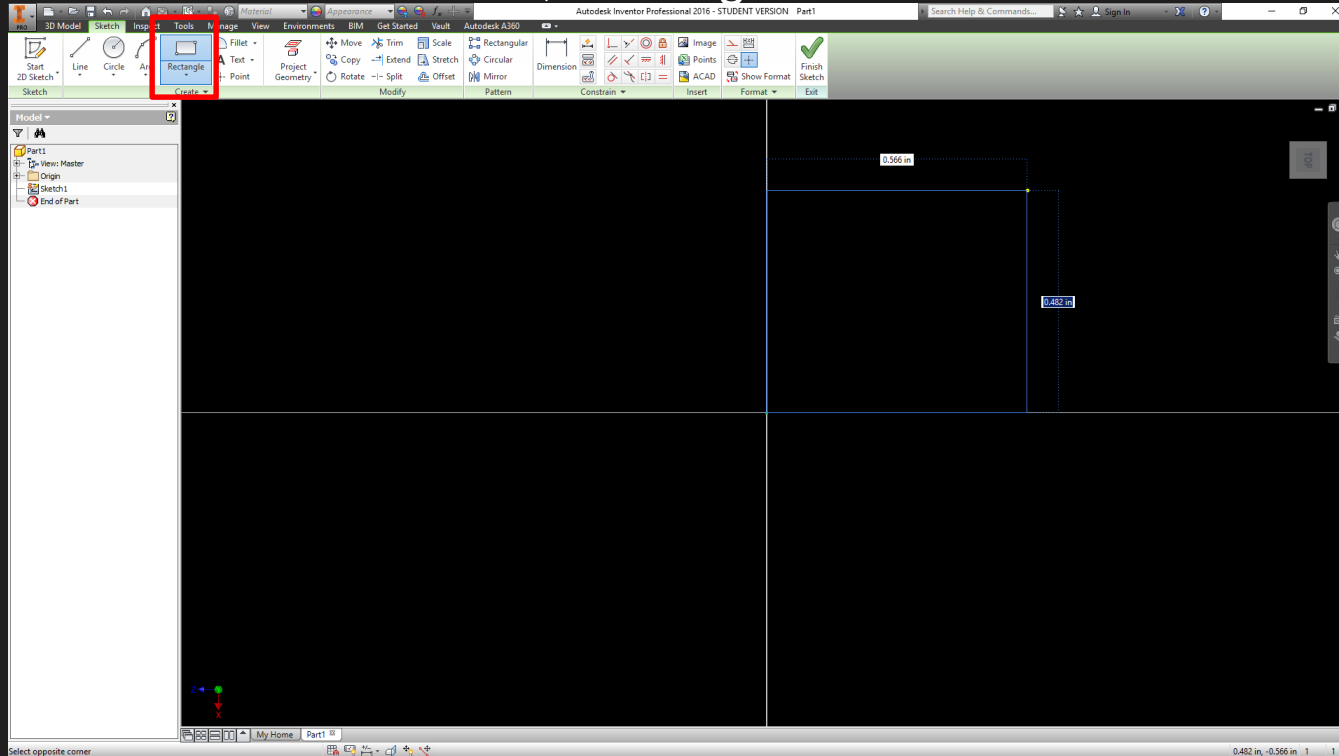
# Arc

- ❑ This tool starts like a line, placing 2 points for the ends, but the third point will choose the size of your arc



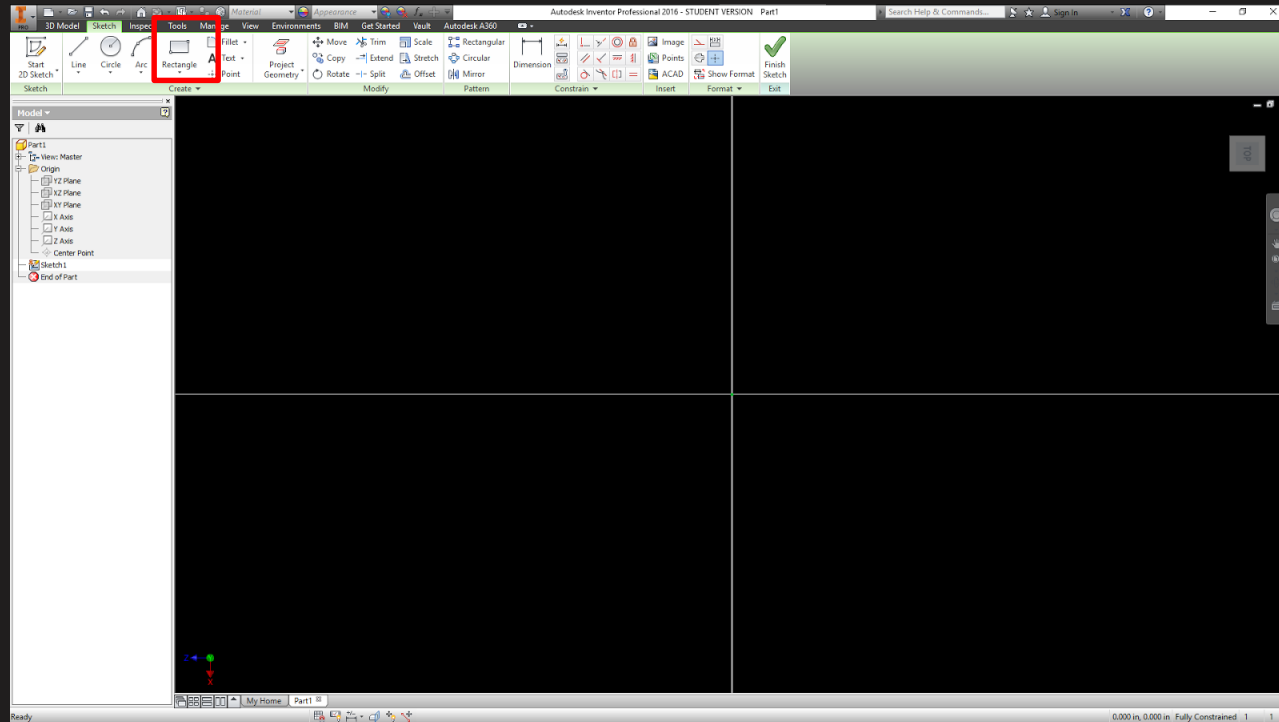
# Rectangle

- Click once for one corner, click again for the other



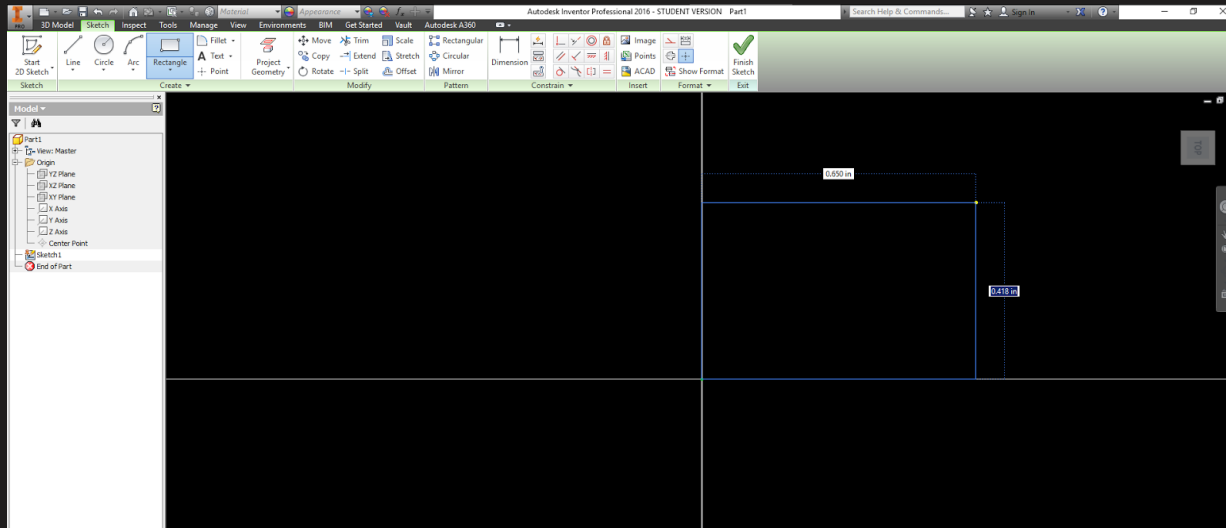
# Creating your square

- ❑ You will now begin creating your cube
- ❑ Start by creating a square



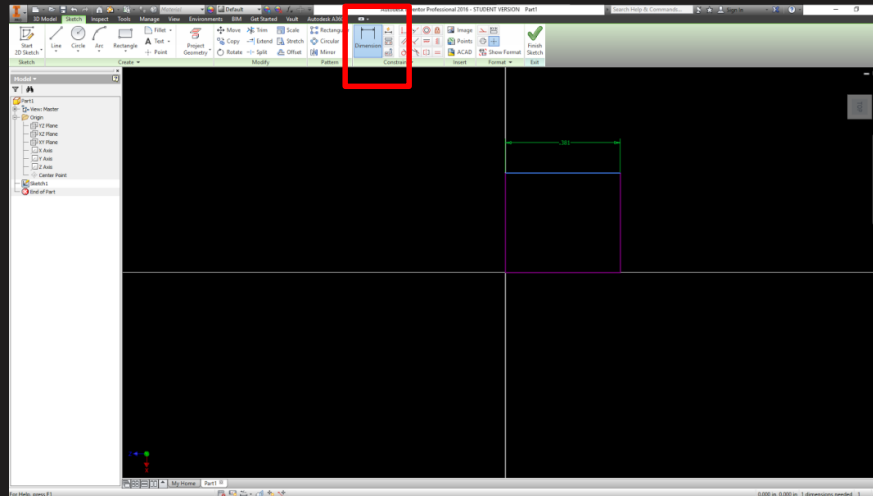
# The origin

- ❑ When sketching anything, try and keep the origin in mind(This is important)
- ❑ If you are simply creating a shape, start it on the origin
- ❑ To create your square, simply click on the origin, and move your mouse anywhere else on the screen and click again.



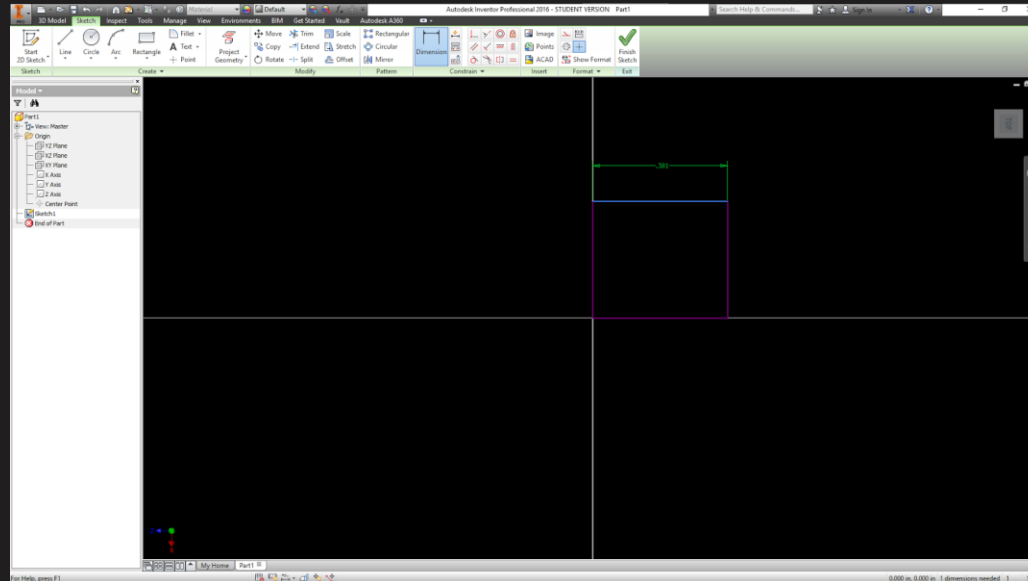
# The dimension tool

- ❑ There are several ways to edit the size of the square
- ❑ You could use the dimension tool; simply click on a line to bring up its current dimension, click again to set it.
- ❑ After doing so you will be prompted to change the dimension
- ❑ For this tutorial, make the square 1 inch by one inch

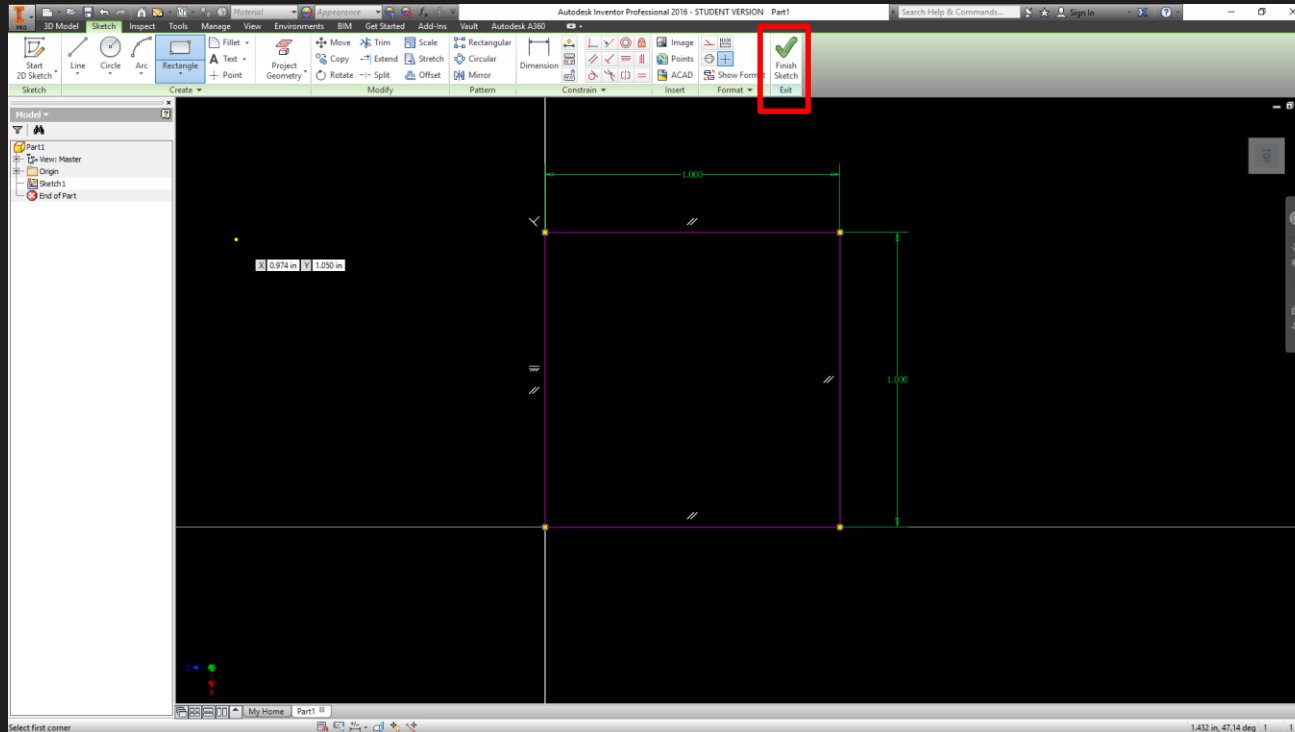


# Another way to dimension

- ❑ Another way to dimension would be to simply move the mouse off in a direction, then enter a value
- ❑ Press tab to switch to a another value
- ❑ Press enter to confirm

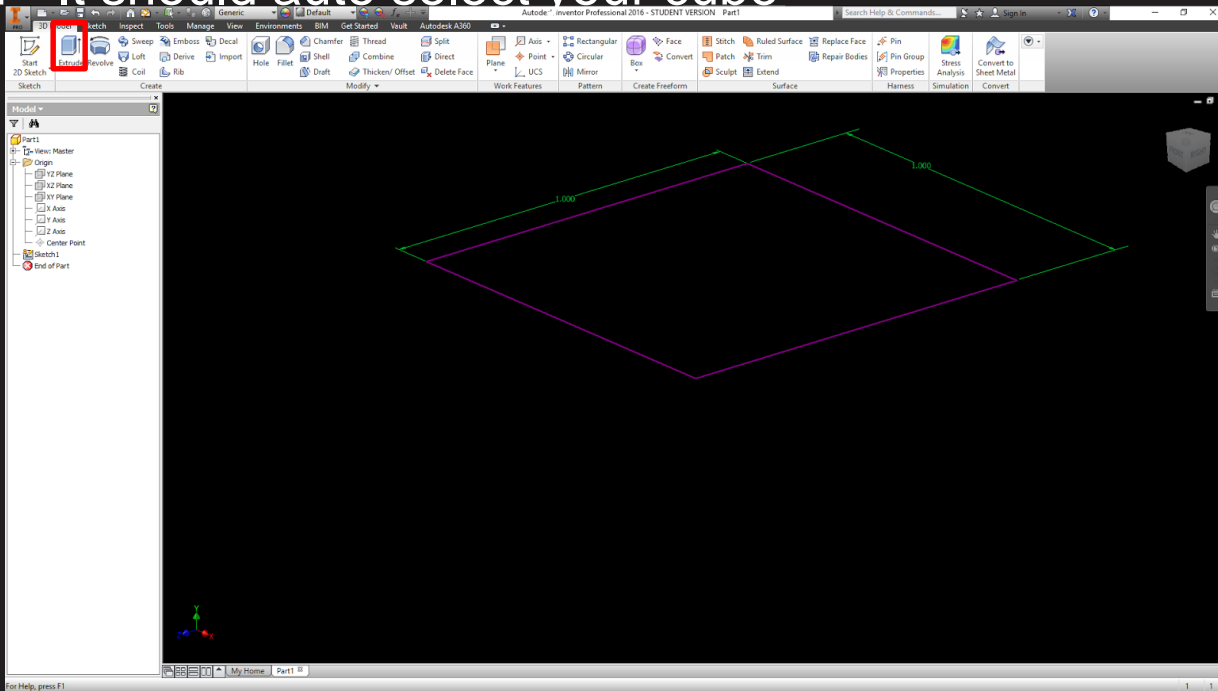


- ❑ So to create your square, click on the origin
- ❑ Then use one of the two methods explained prior to make your square the proper size
- ❑ After you do this, click “Finish Sketch”

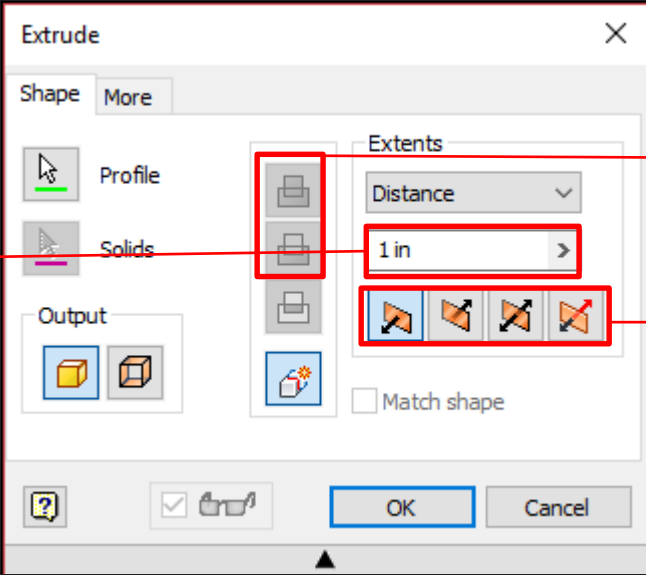


# Extruding your square

- ❑ Next we will extrude your square into a cube
- ❑ To do so, click “Extrude”
- ❑ It should auto select your cube



- ❑ This is the extrusion prompt
- ❑ For this tutorial, just change the distance to match your square and click ok



Distance extruded or cut

Choose between cutting or extruding  
Extrude is on top, below it is cut

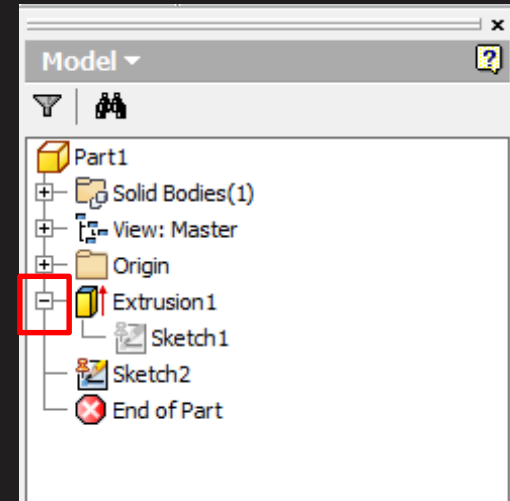
Choose one direction, or both

# A note on extrusions and cuts

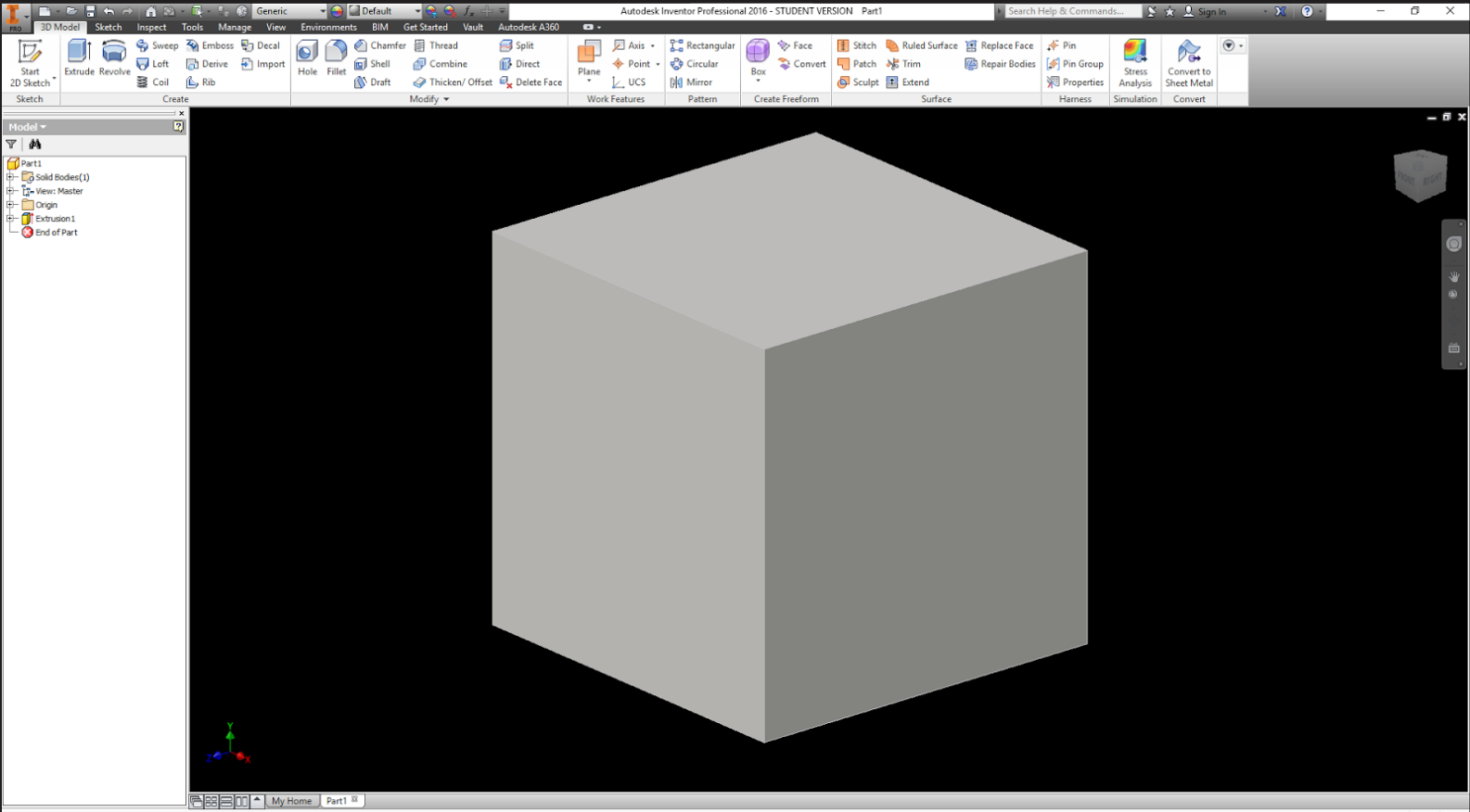
- ❑ When making any extrusions or cuts after the initial, the program will not automatically select the sketch, so make sure the right geometry is selected
- ❑ You can only have geometry from one sketch per extrusion
- ❑ A cut is the opposite of an extrusion, it takes away material instead of adding some

# Editing an already extruded feature

- ❑ To edit something from an already created feature, go to the browser
- ❑ Click on the little plus to bring to bring up the sketch
- ❑ Double click on the sketch to edit it
- ❑ Just be careful, doing this can sometimes mess with other features that may have been constrained to it

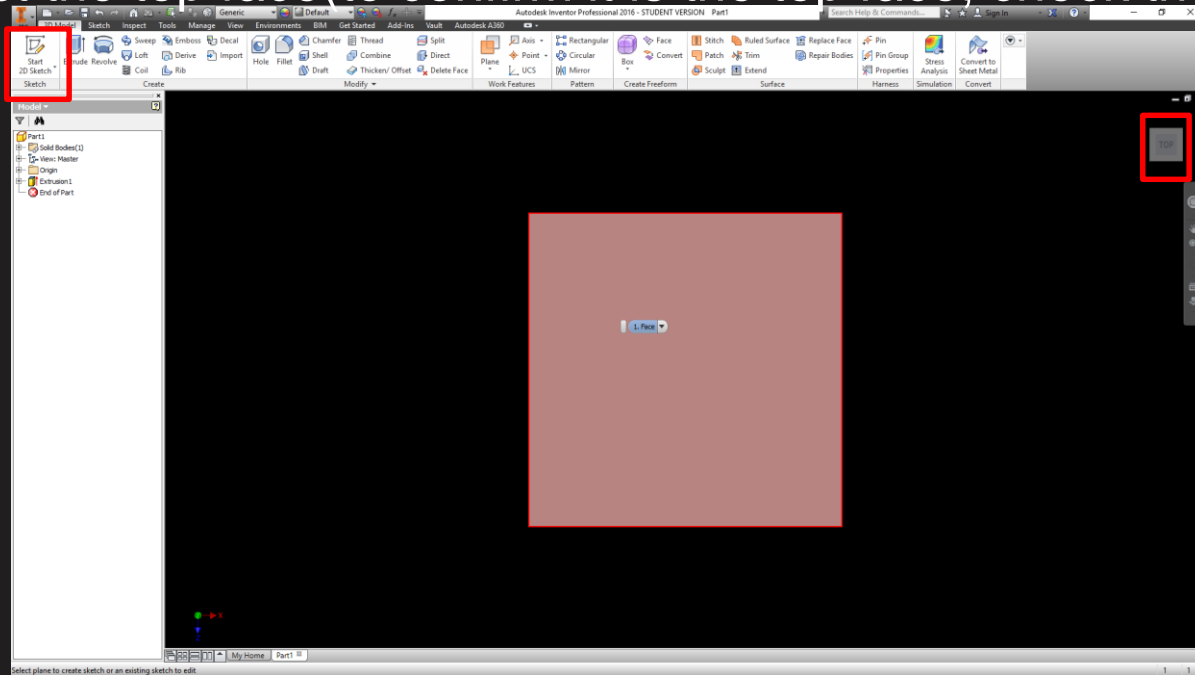


# Save Often



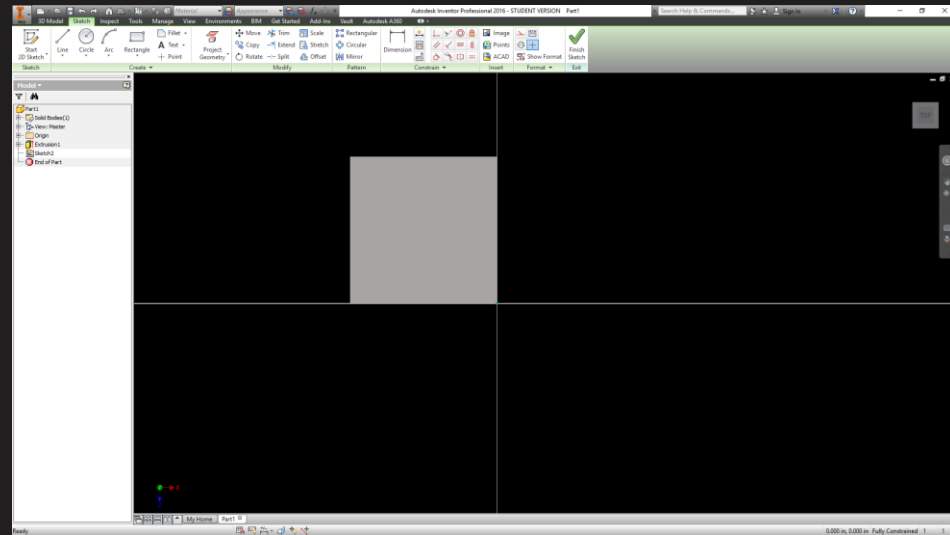
# Adding a channel

- ❑ Click on “Start 2D sketch”
- ❑ Select the top face (to confirm it is the top face, check the ViewCube)



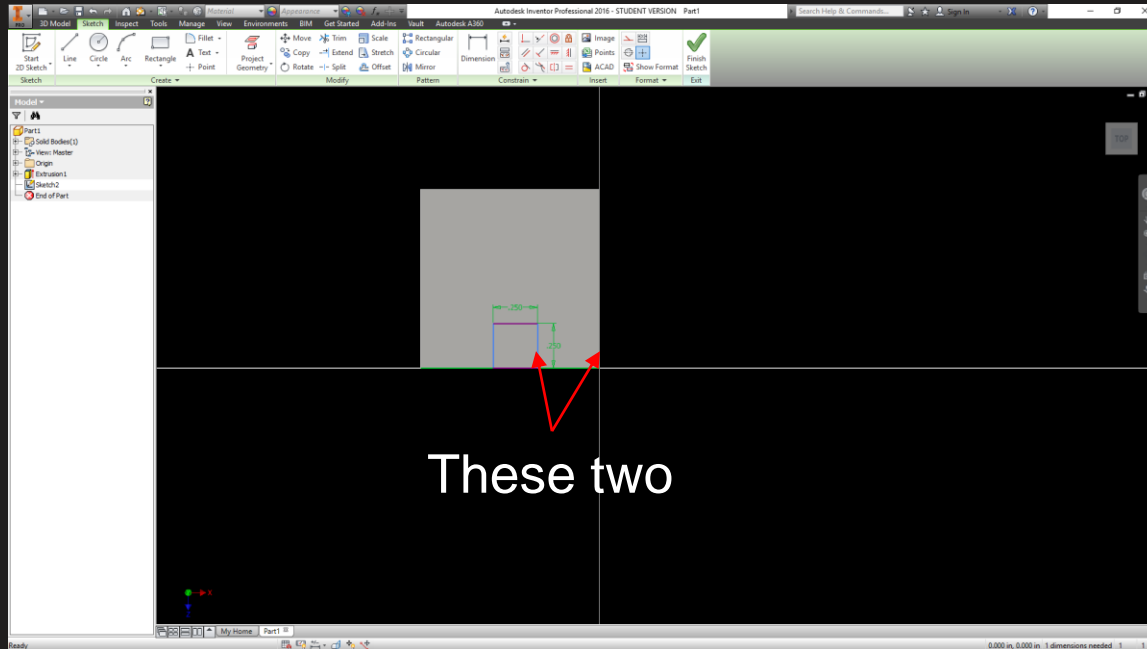
# Sketching the channel

- ❑ To begin, select rectangle
- ❑ Place the first corner on the bottom edge, where exactly does not matter right now
- ❑ Use either of the dimensioning methods explained earlier to make the square .25 inches by .25 inches



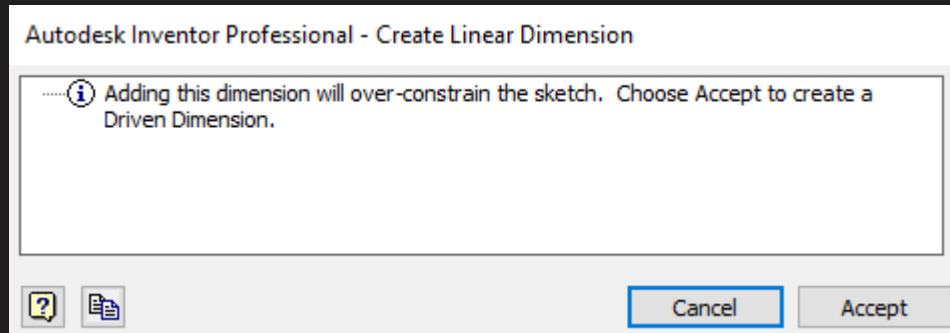
# Centering the channel

- ❑ To center your cube, select one side of your channel to the edge of the cube, and enter the value .375
- ❑ This should center the channel



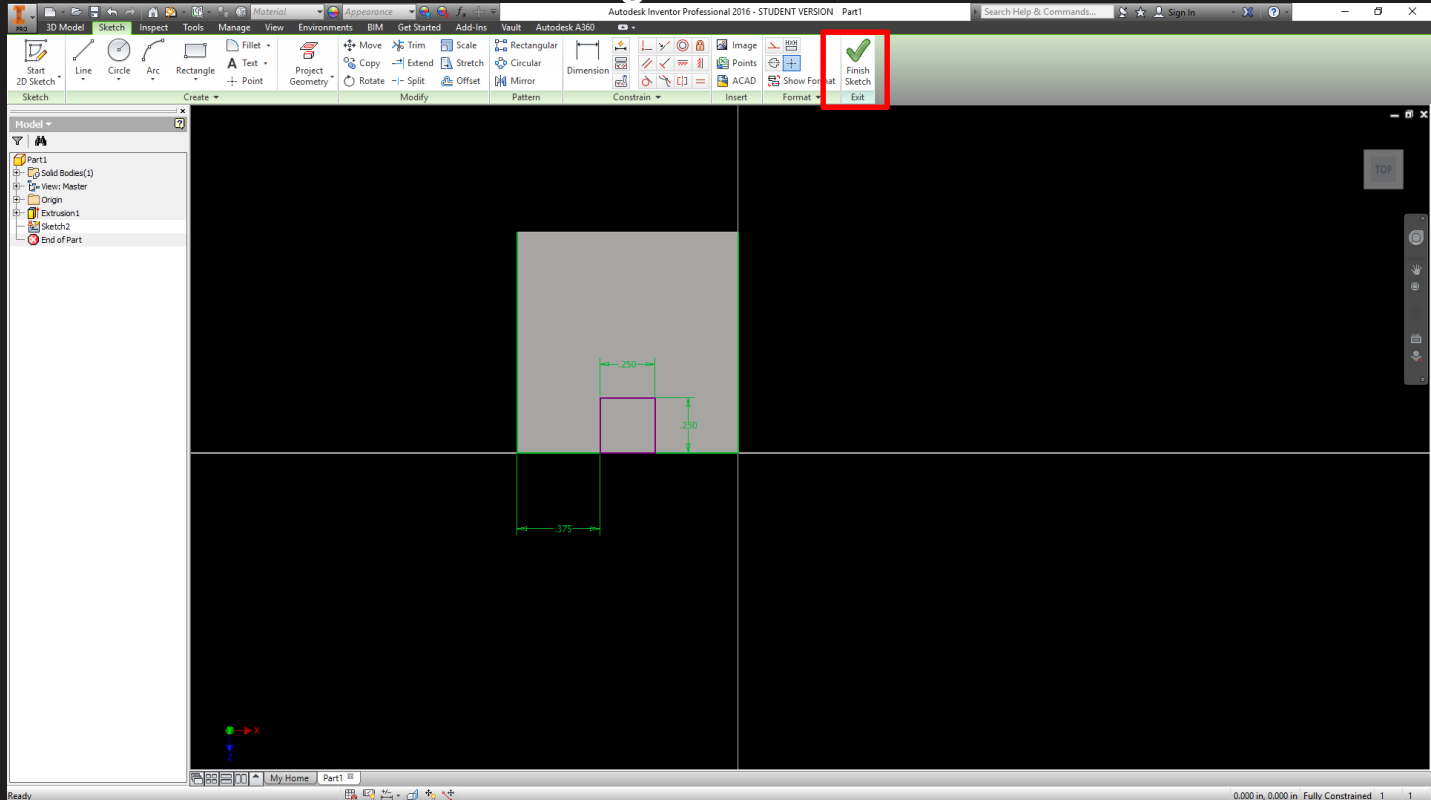
# Driven dimensions

- ❑ If you attempt to add a dimension that will conflict with another if changed, you will get the prompt shown below, and hitting accept will create a driven dimension
- ❑ Driven dimensions cannot be changed, and will appear with brackets



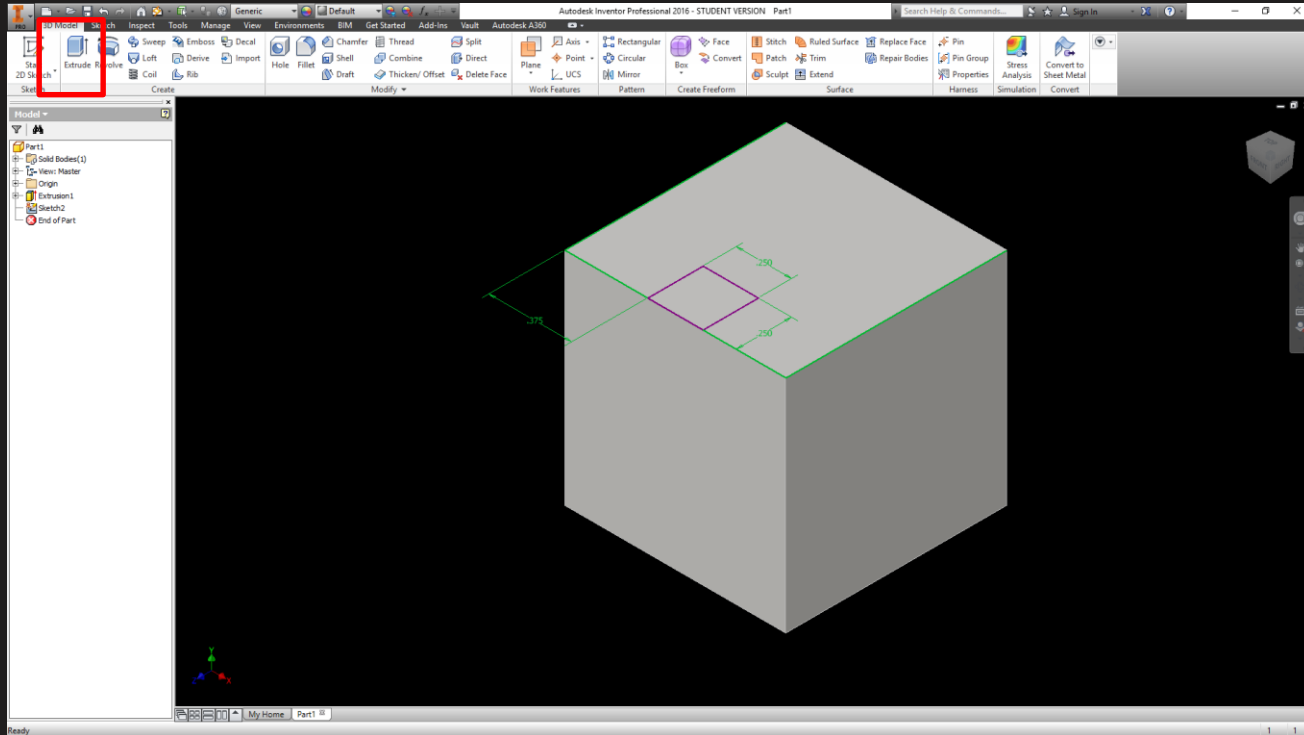
# Your channel should look like this

- ❑ If it looks the same as the image below, click finish sketch



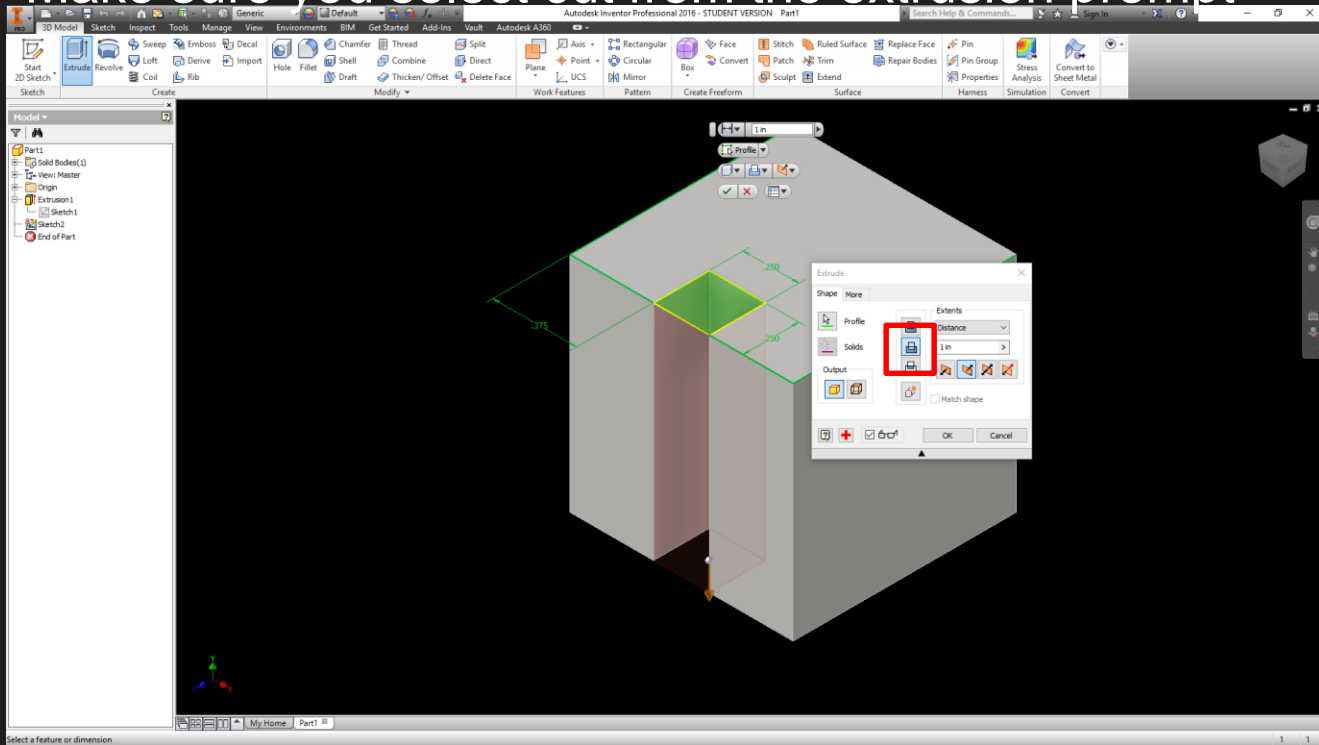
# Cutting out your channel

- ❑ Now we will cut out your channel
- ❑ Click on “Extrusion”

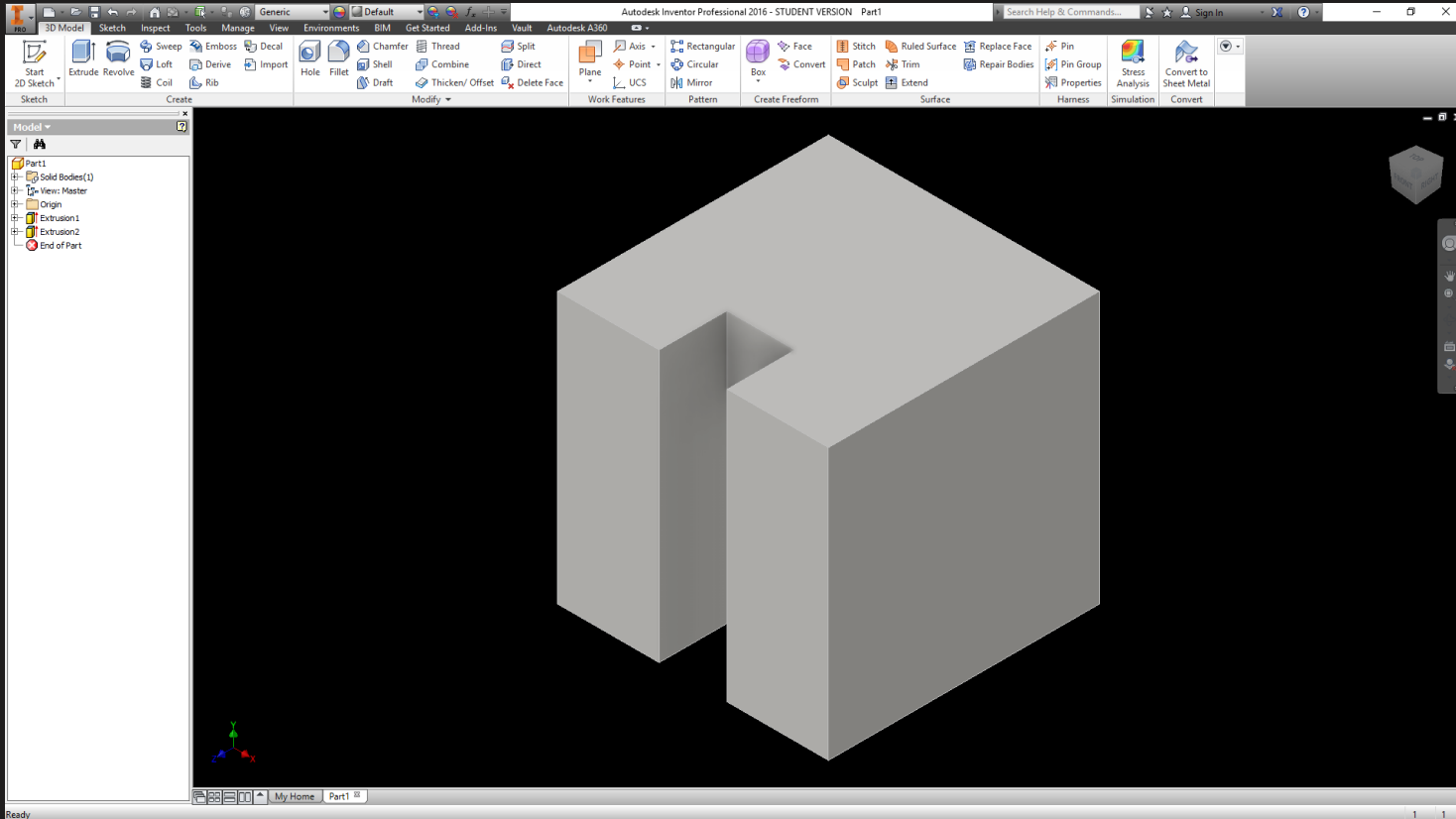


# Cutting out your channel cont.

- ❑ Select you channel
- ❑ Make sure you select cut from the extrusion prompt

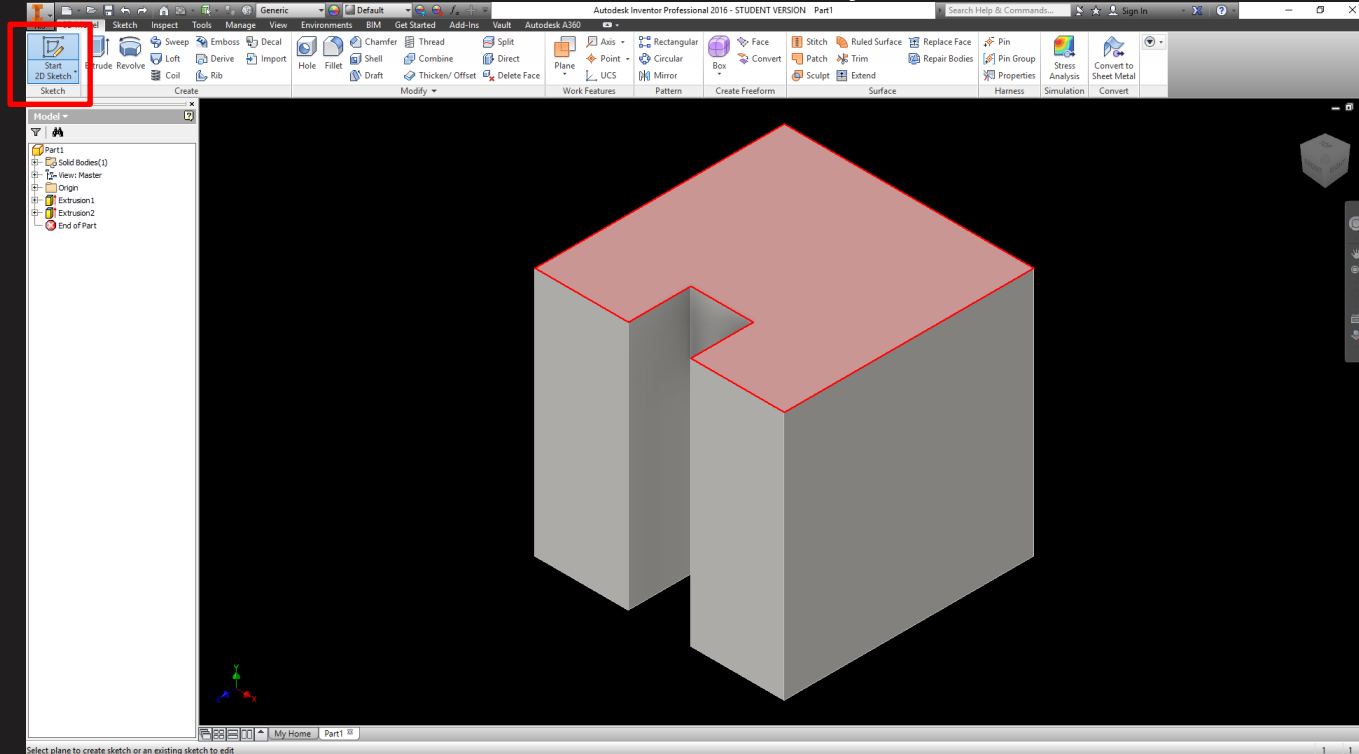


# Your cube should look like this



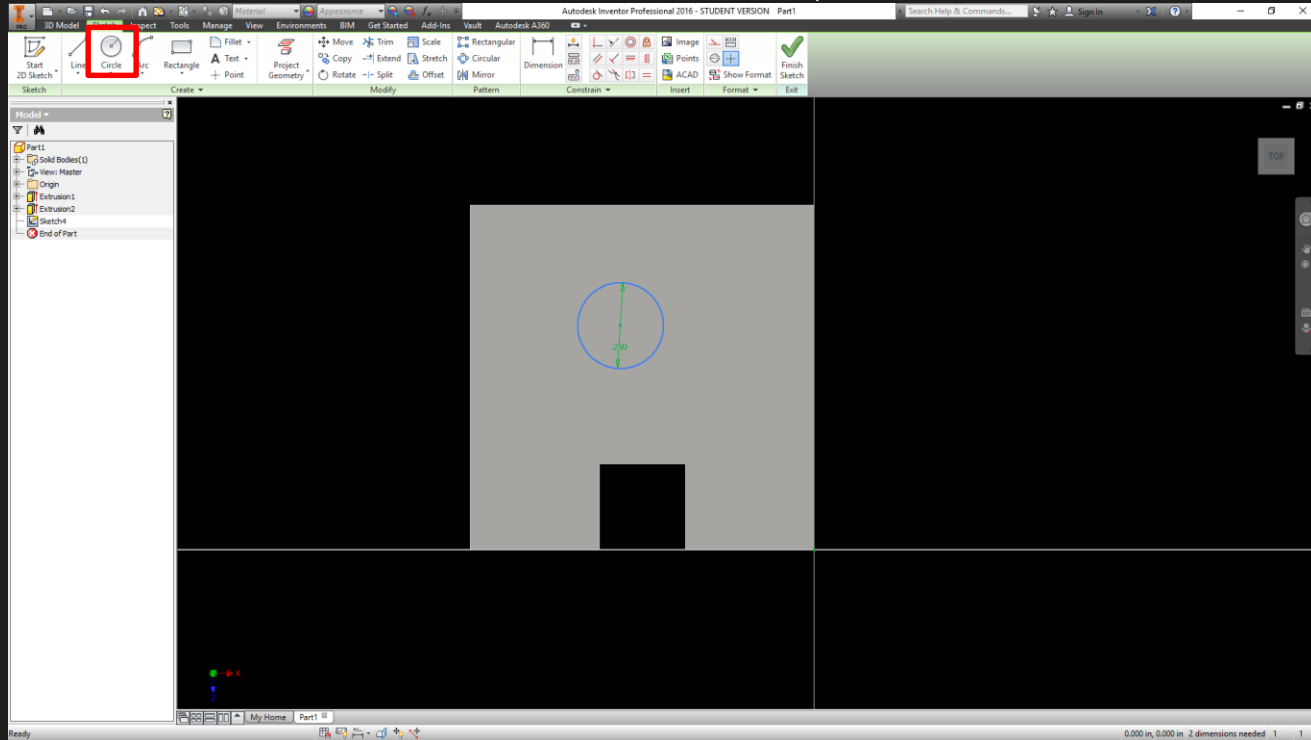
# Adding a hole

- ❑ Create a sketch on the same face as you used for the channel



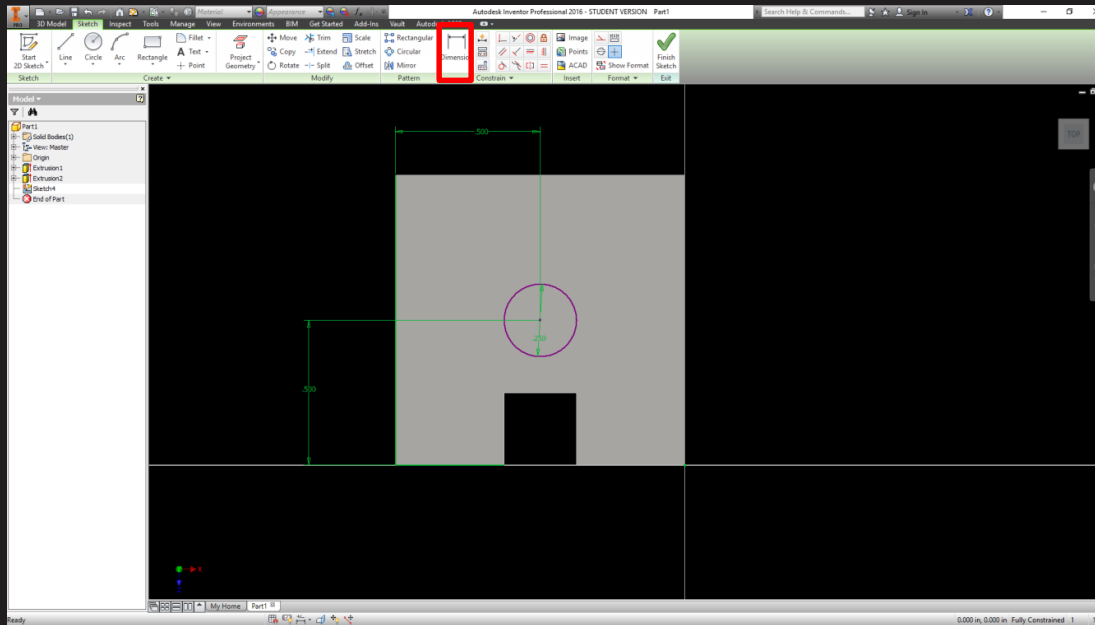
# Sketching the circle

- Create a circle somewhere on the face, make it .25 inches in diameter



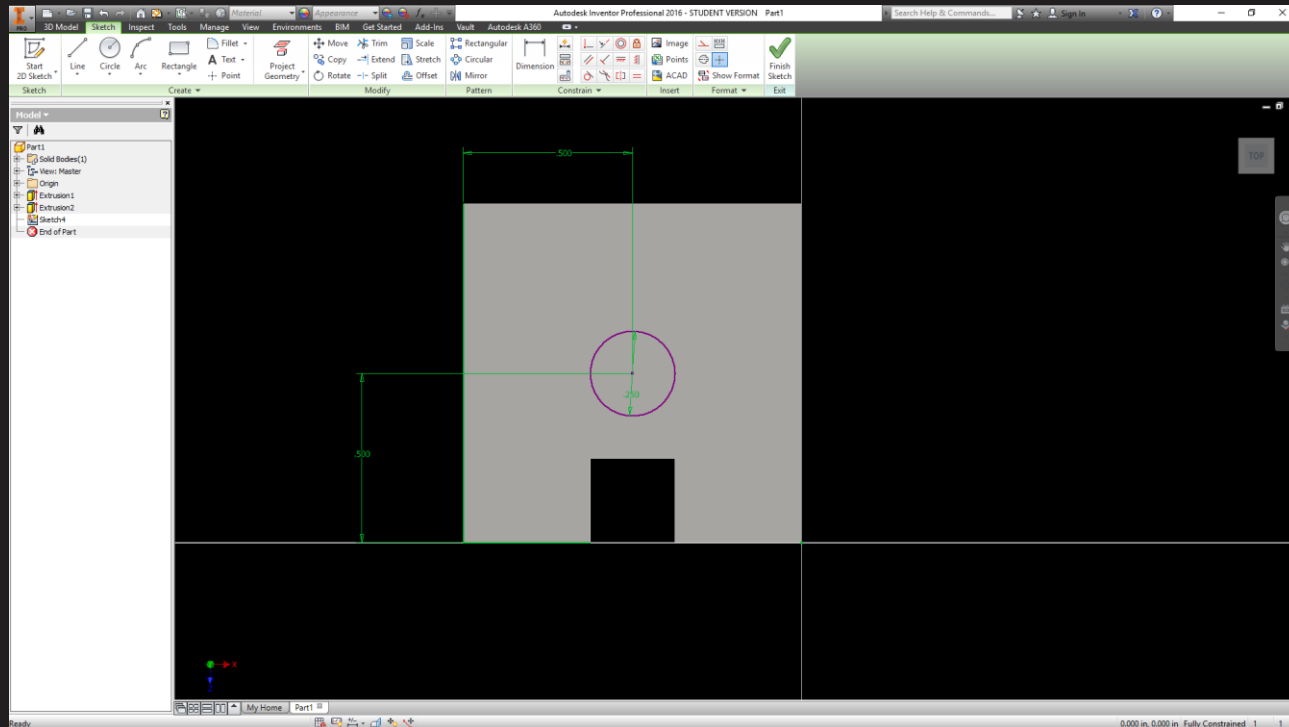
# Centering the circle

- ❑ Using the dimensioning tool, the center point of the circle, then the outside of the cube, set the value to .5
- ❑ Do the same thing for the side of the cube next to it



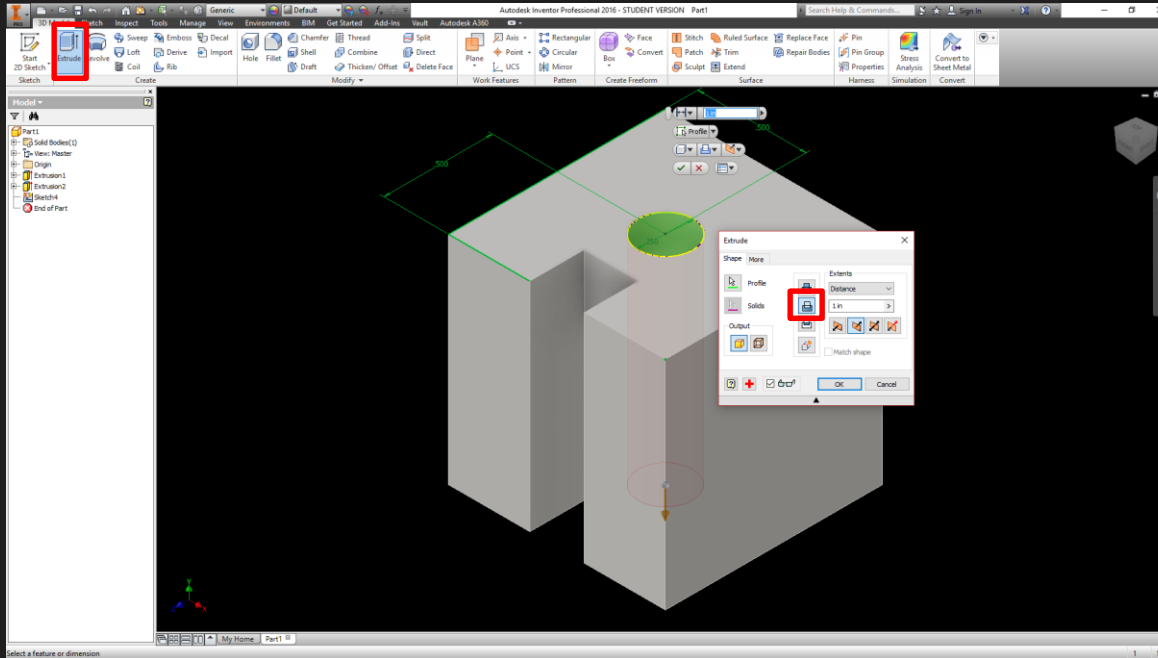
# Your circle should look like this

- If it looks the same as the image below, click finish sketch

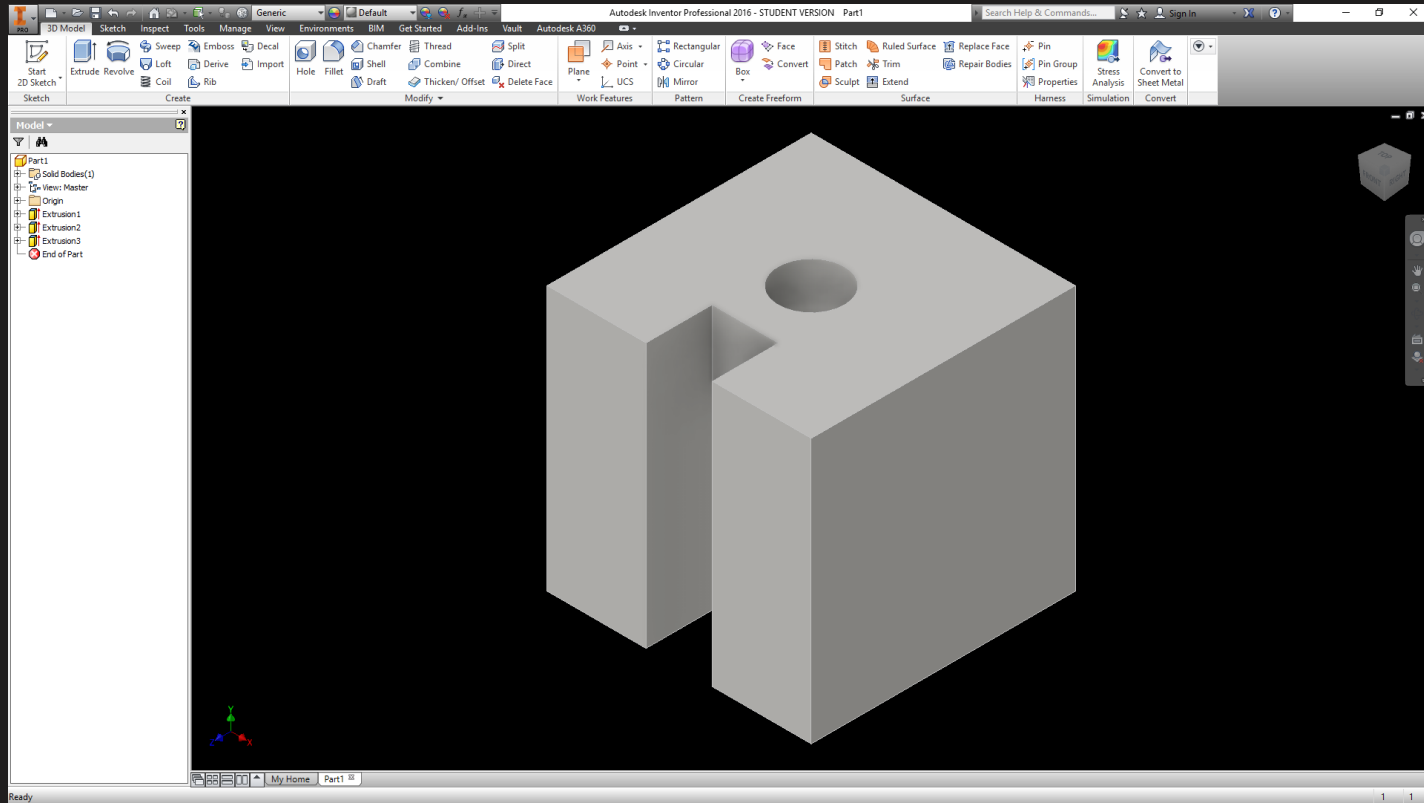


# Cutting your hole

- ❑ Click on “Extrude”
- ❑ Select your circle
- ❑ Click on cut from the extrusion prompt

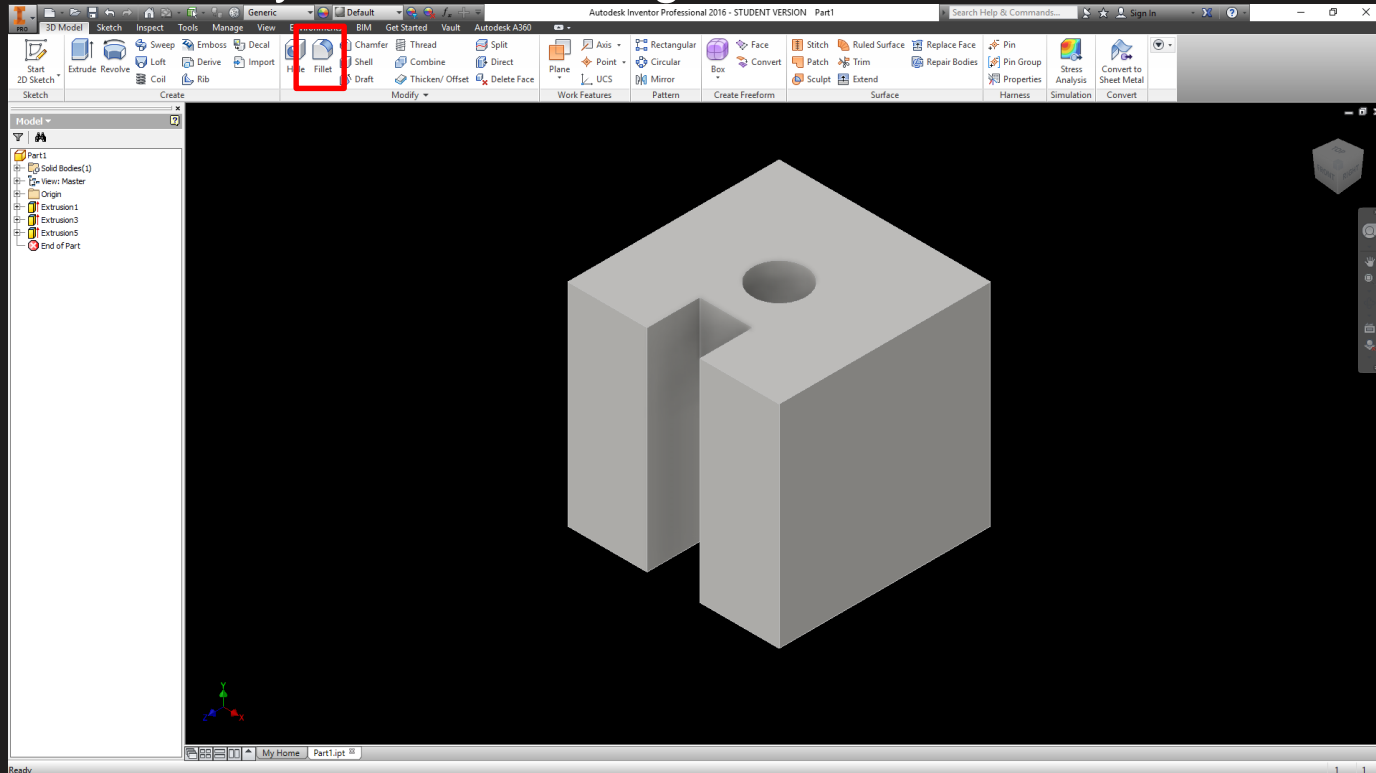


# Your cube should look like this



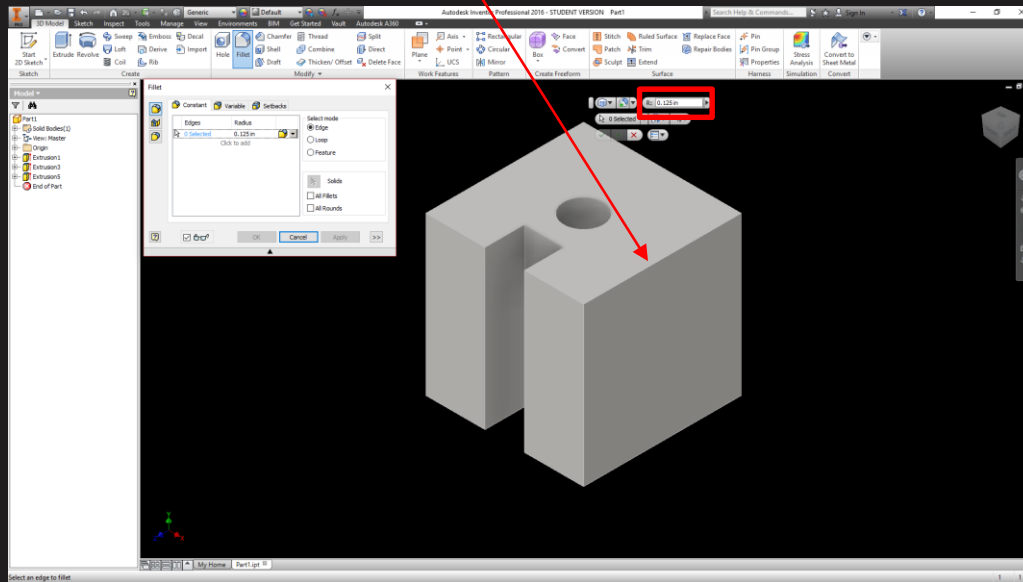
# Fillets

- A fillet is a way to curve an edge

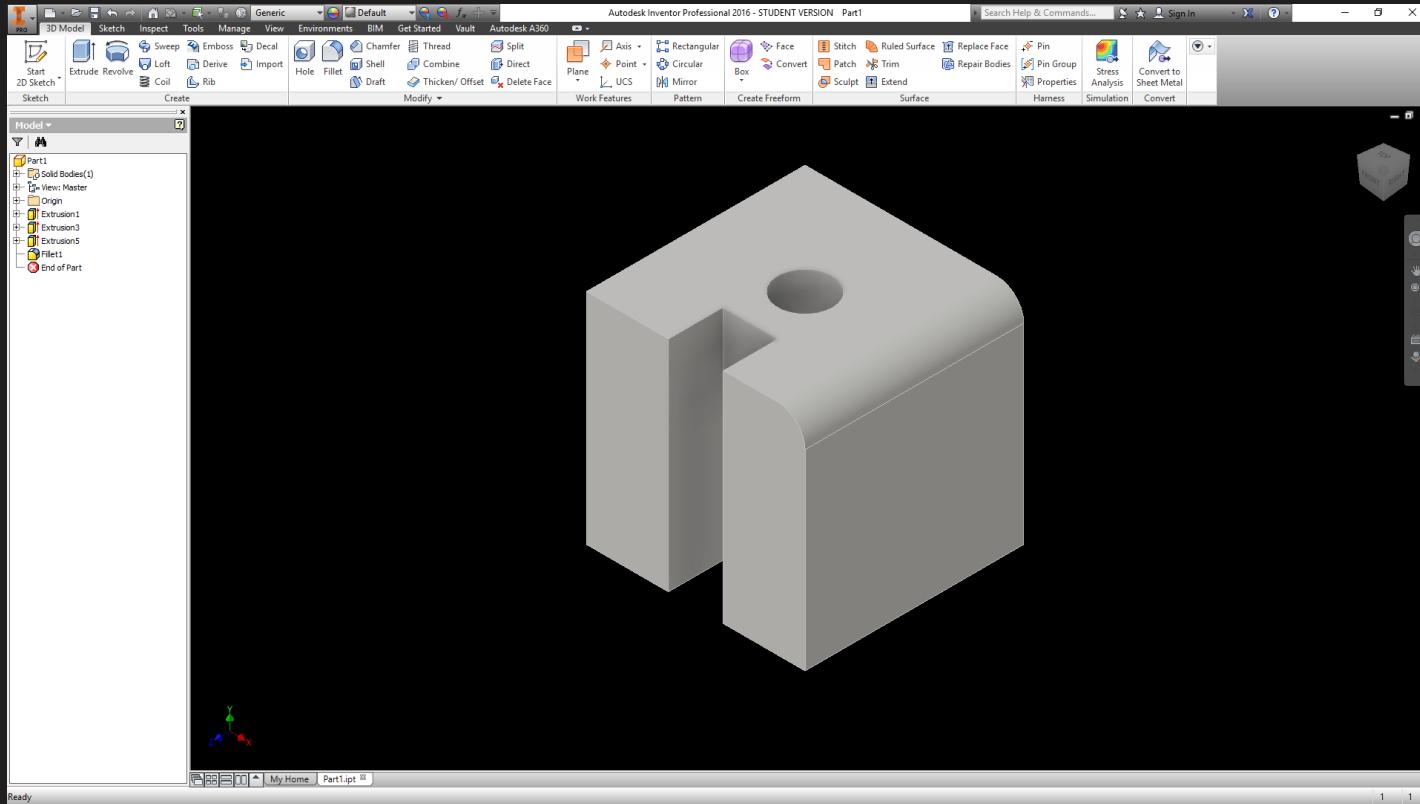


# Fillets continued

- ❑ Don't worry about most of the fillet prompt, just select the edge shown below and hit OK
- ❑ If you want to edit the radius of the fillet, change it in the box shown below

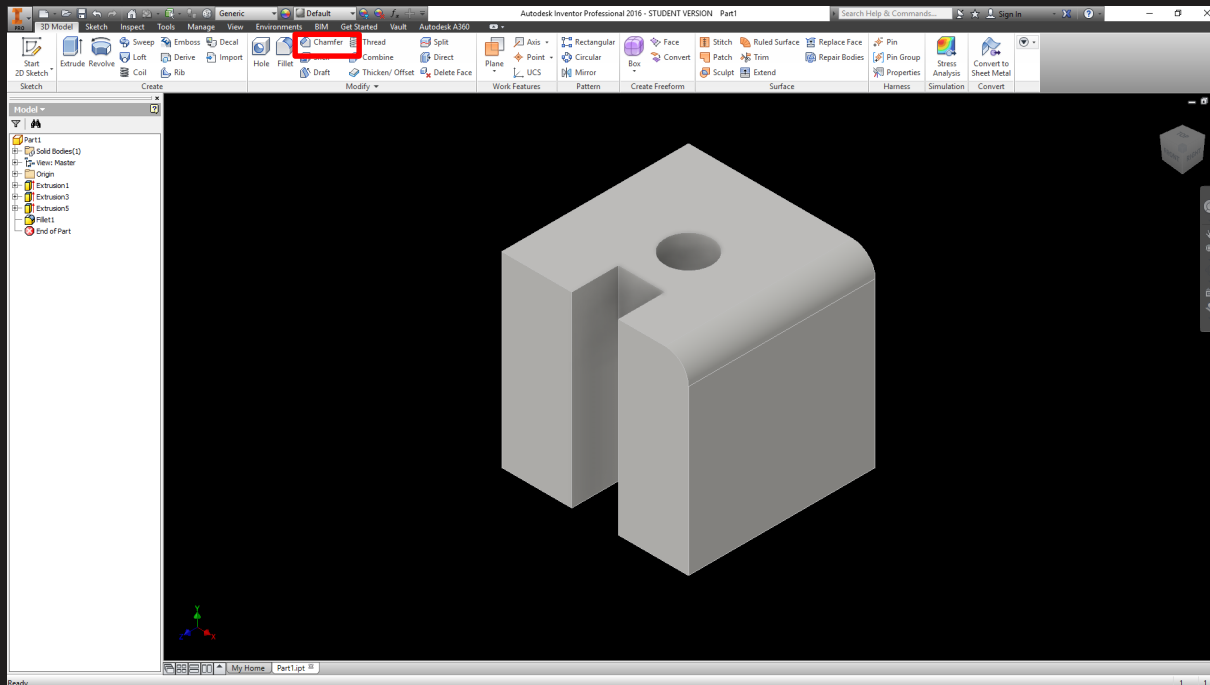


# Your cube should look like this



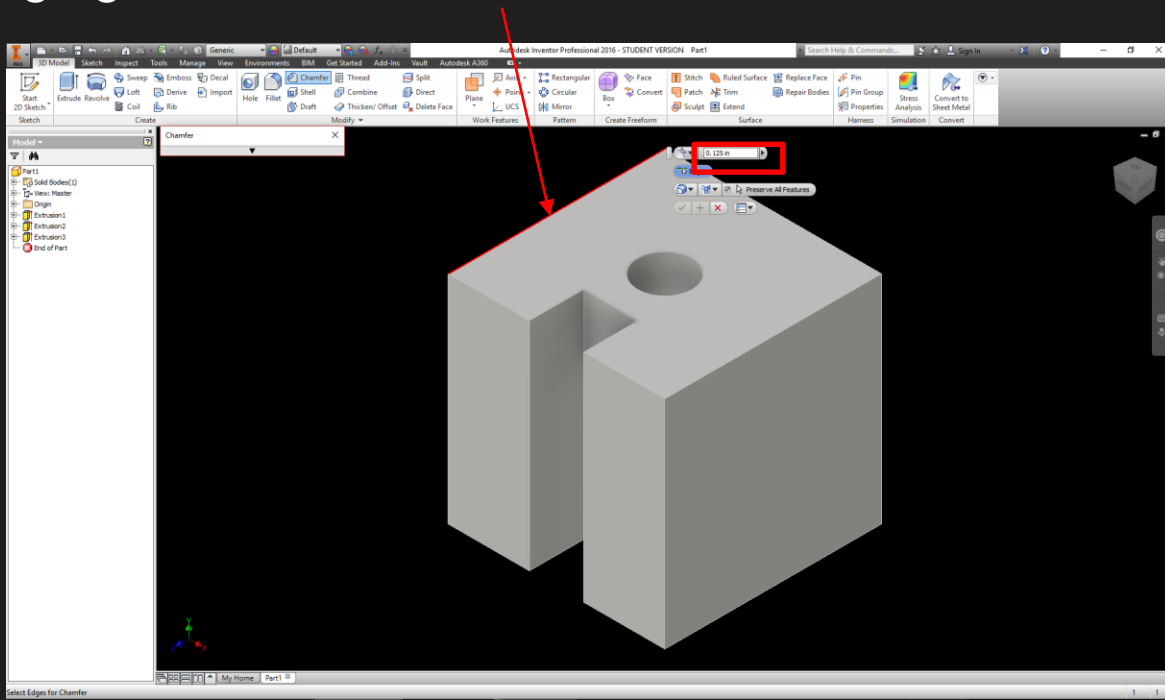
# Chamfers

- A chamfer is similar to a fillet, but instead of being rounded, its angled.

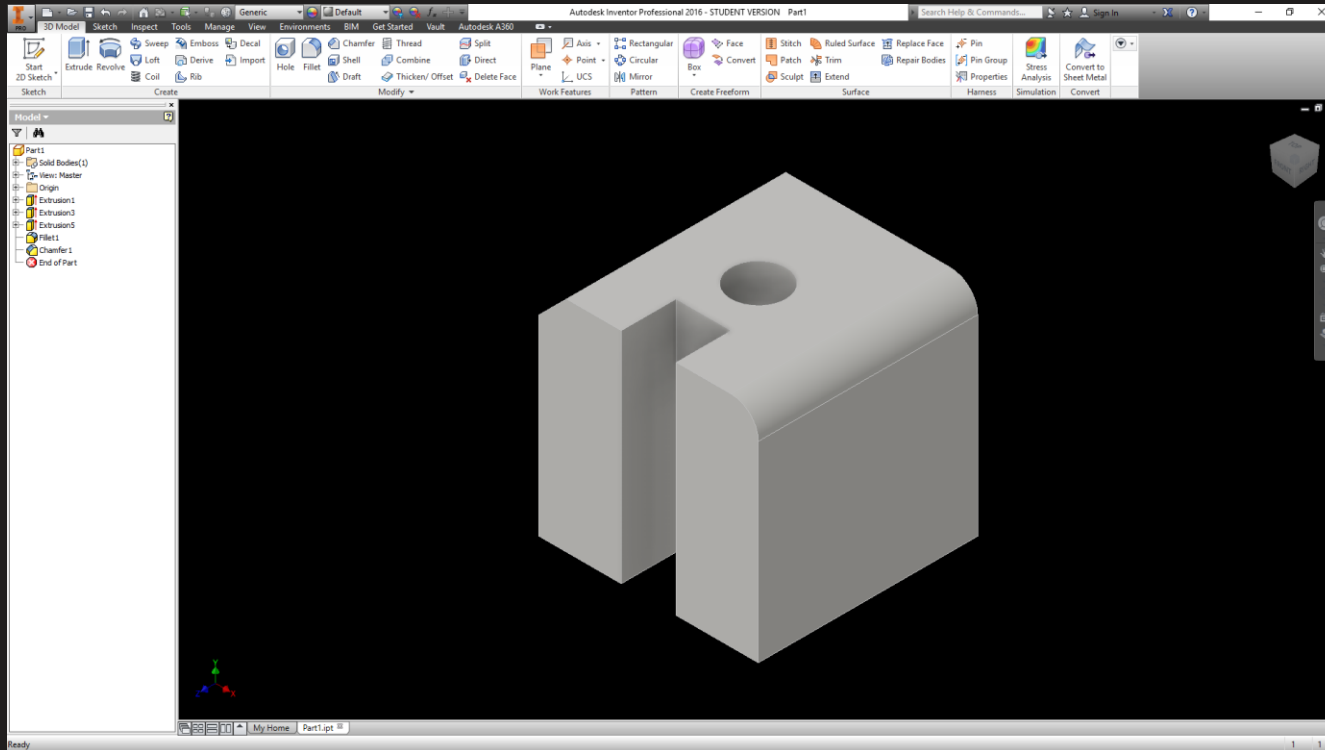


# Chamfers continued

- ❑ It works nearly the same as the fillet command
- ❑ Simply click the “Chamfer” button, and select the edge shown below
- ❑ Changing its size is the same as the fillet

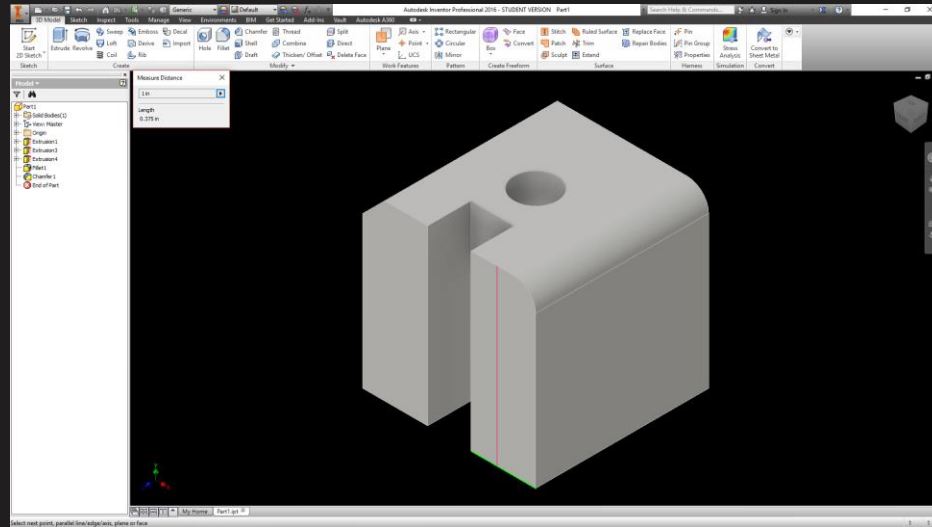


# Your cube should look like this



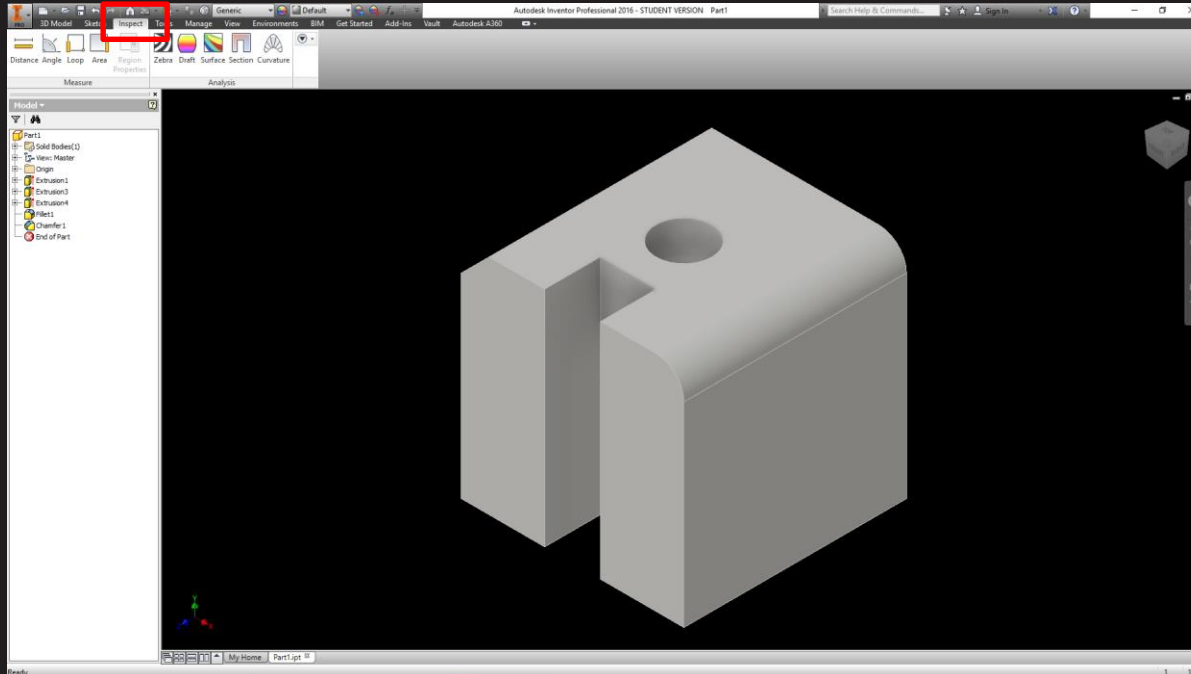
# Distance tool

- ❑ One of the best tools to tell distance between 2 things (planes, lines or points), is the distance tool
- ❑ Pay attention when you use it though, make sure you are measuring what you want, check the line between the two



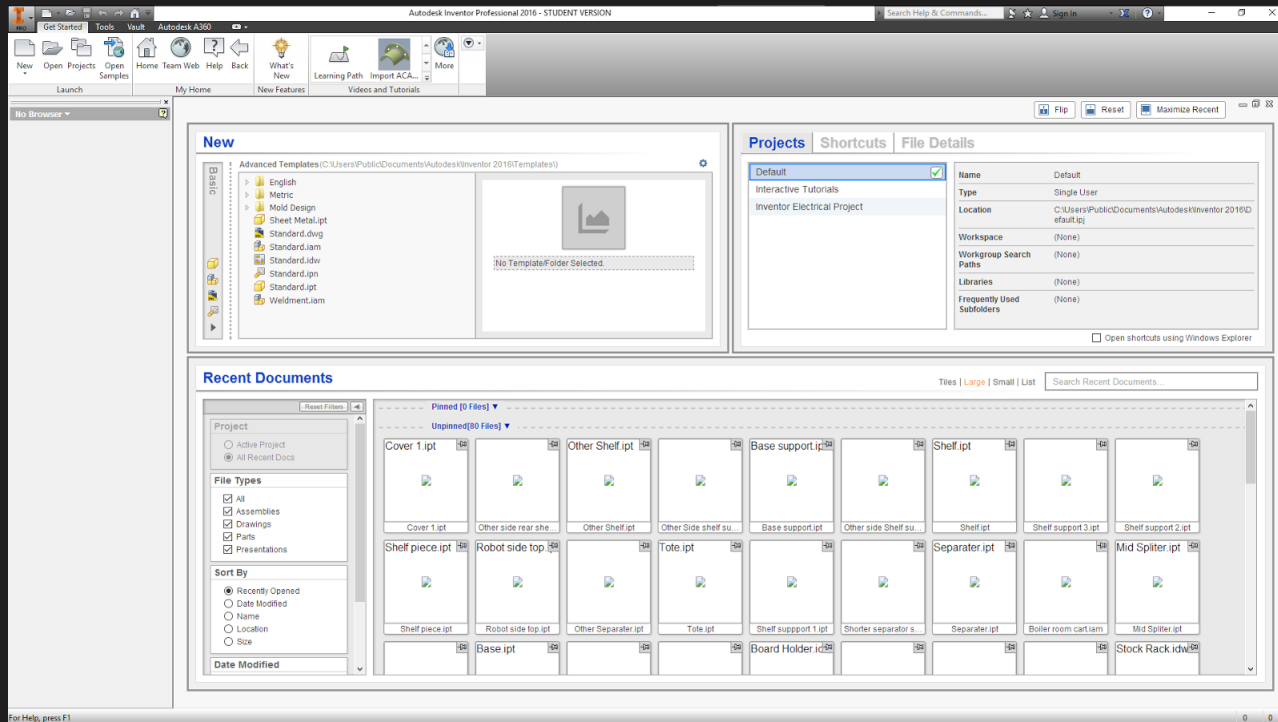
# Distance tool continued

- ❑ To select the “Distance” tool, go the inspect tab and it will be on the left(hotkey is M, it’s faster to do that)

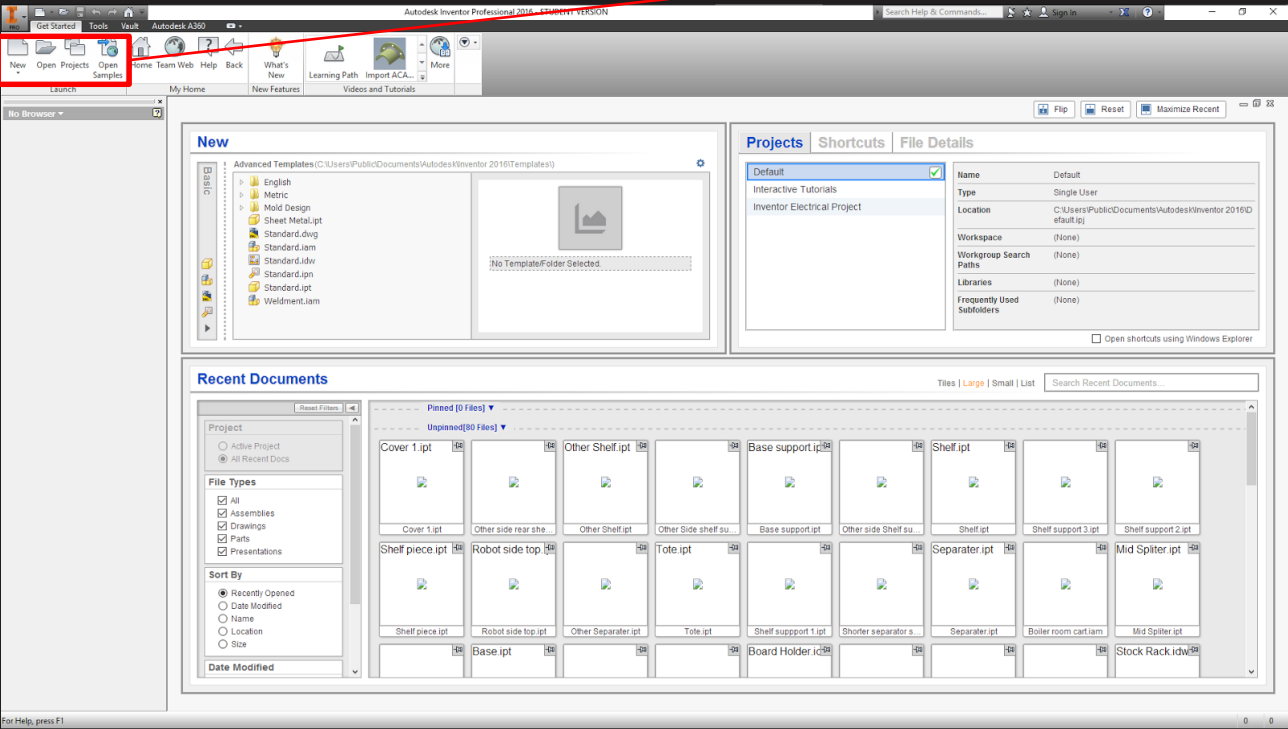
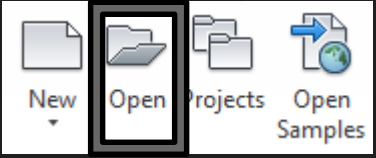


# Opening a part

- When you first open the program, you will see this screen



- To open a part, click on the icon under the launch tab with the open folder icon labeled "open"
- This will open the normal prompt to open a file.



# A note about save location and naming of files

- ❑ Do not save random files in dedicated folders, as it can create a lot of clutter.
- ❑ If a part you create that is not used, delete it.
- ❑ In general keep naming professional.
- ❑ Do not use the Cougartech drive for personal use, only robotics related items.

# Challenge

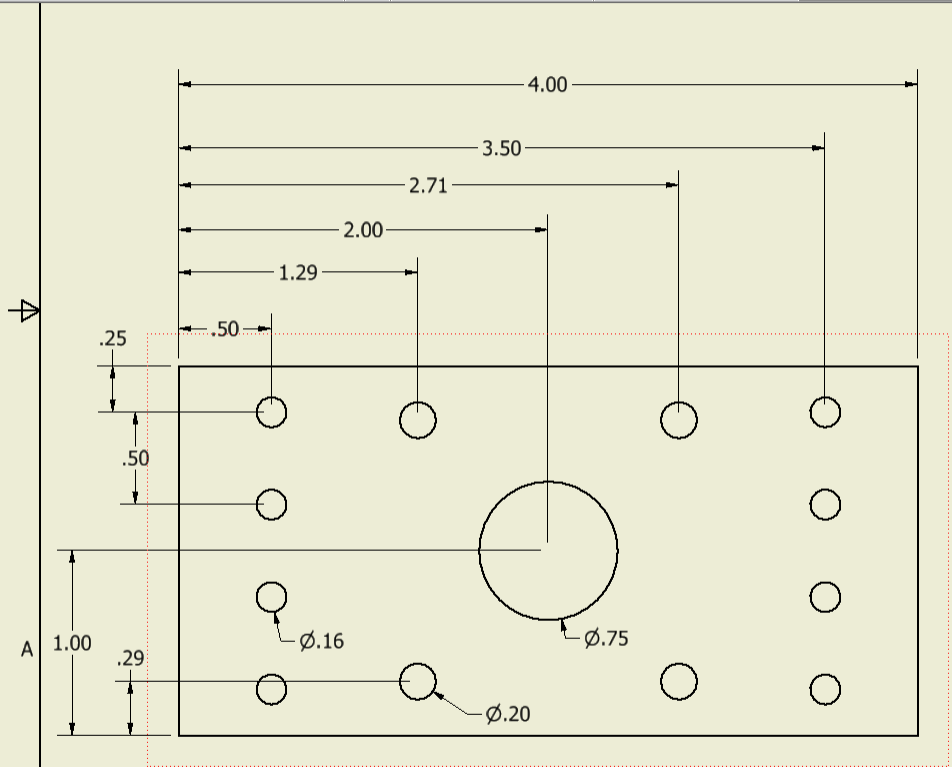
- ❑ In order to compete this tutorial, you must create a motor mounting plate.
- ❑ The layout has been provided for you in the following slide
- ❑ You do not need to make a layout yourself

Place Views Annotate Sketch Tools Manage View Environments Get Started Vault Add-Ins Autodesk A360

Dimension Ordinate Chain Dimension Feature Notes Text Text Insert Sketch Symbol Surface Welding Impor... Caterp... Symbols Start Sketch Parts List Revision General Hole Revision General Balloon Edit Layers Layer Style Format

Model

- part
  - Drawing Resources
  - Sheet: 1
    - Default Border
    - ANSI - Large
    - VIEW1:part.ipt
    - VIEW2:part.ipt
    - VIEW3:part.ipt
    - part
    - VIEW4:part.ipt



DRAWN	
Owner	9/3/2016
CHECKED	
QA	
MFG	
APPROVED	

# Congratulations

Upon showing a mentor or the cad lead your completed cube and plate, you will have completed the first drafting tutorial