SOLIDWORKS: Lesson 04 - Basics and Modeling Fundamentals

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Assist Proof

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Engineering Technical College / Najaf

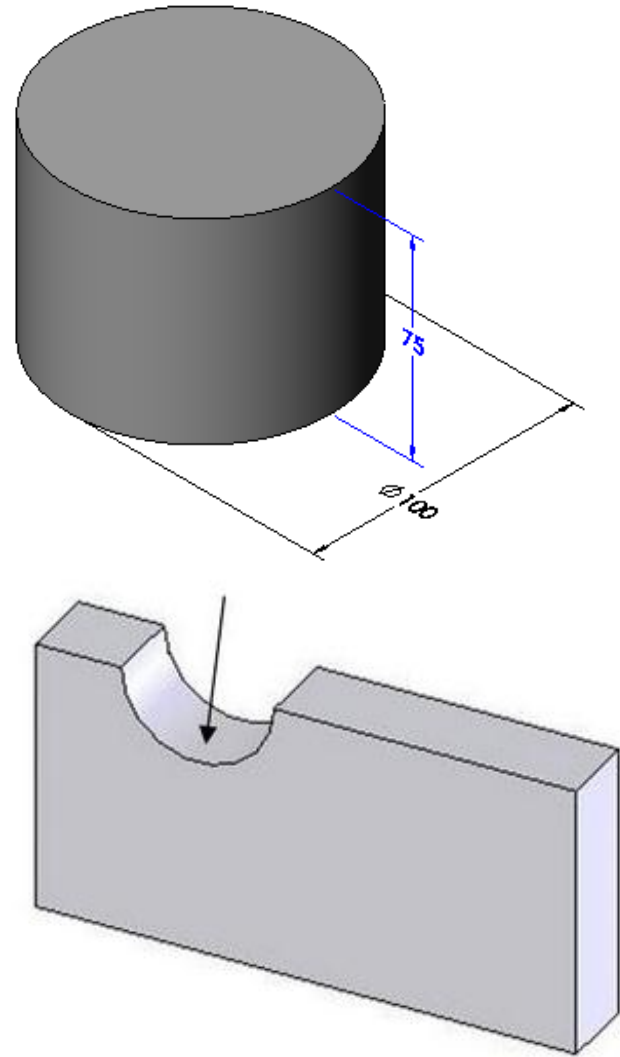**email:coj.dfr@atu.edu.iq**

SolidWorks Exercise

Now that we have explored the interface of SolidWorks, lets create a simple part step-by-step.

For now, we are only going to concern ourselves with one type of feature: Extruded Boss/Base.

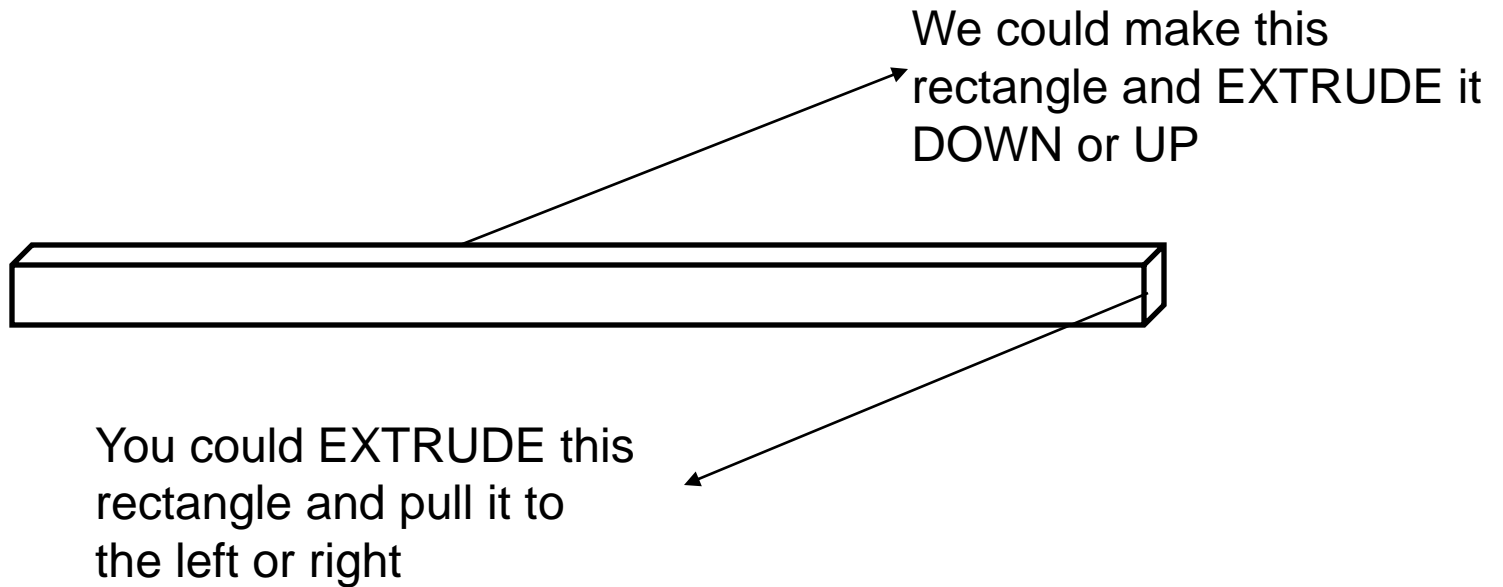# SolidWorks Exercise – What is Extrude?

Extrude – When you take a 2D area and push the design out into another dimension. A 2D area, for example, can be made into a 3D volume by extruding it out a specific distance, d.

You can extrude to make a SOLID or you can extrude to make a CUT
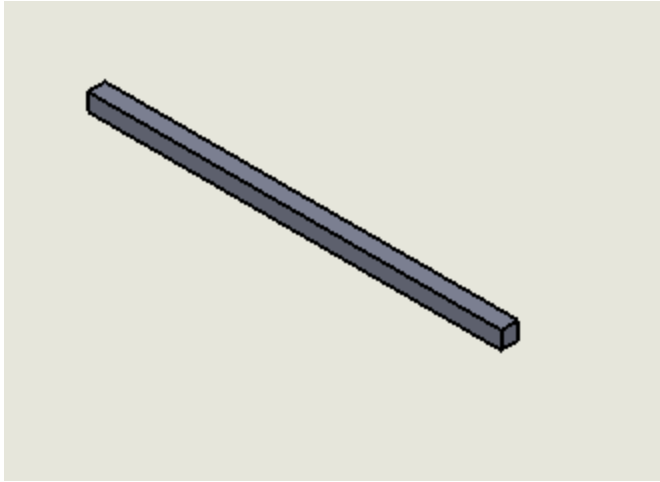
SolidWorks Exercise

There are MANY ways to EXTRUDE a surface

We could make this rectangle and EXTRUDE it DOWN or UP

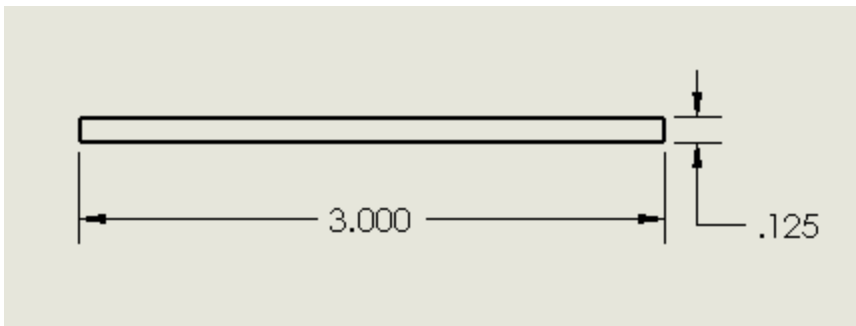You could EXTRUDE this rectangle and pull it to the left or right

Let's begin. Choose FILE, then NEW PART

SolidWorks Exercise

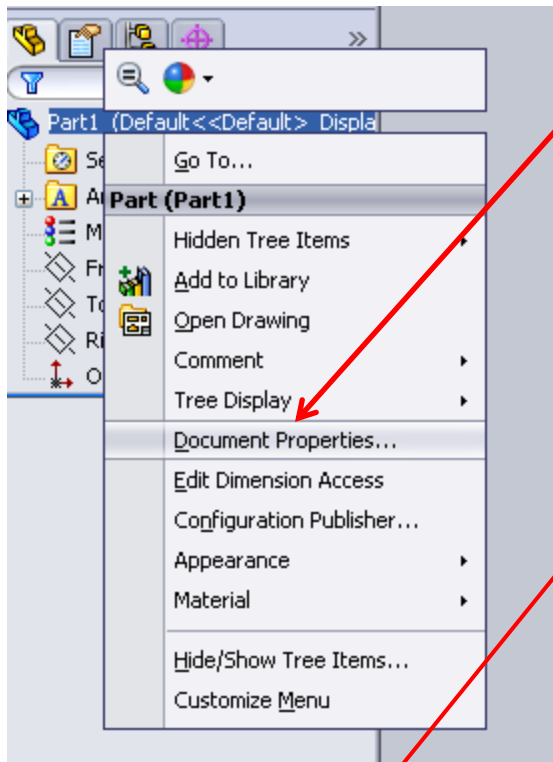## We wish to model the following part:

This is a basic 3 inch bar used often time to mount our wheels to our motors. It is obviously 3 inches long, yet 0.125 inches in width.
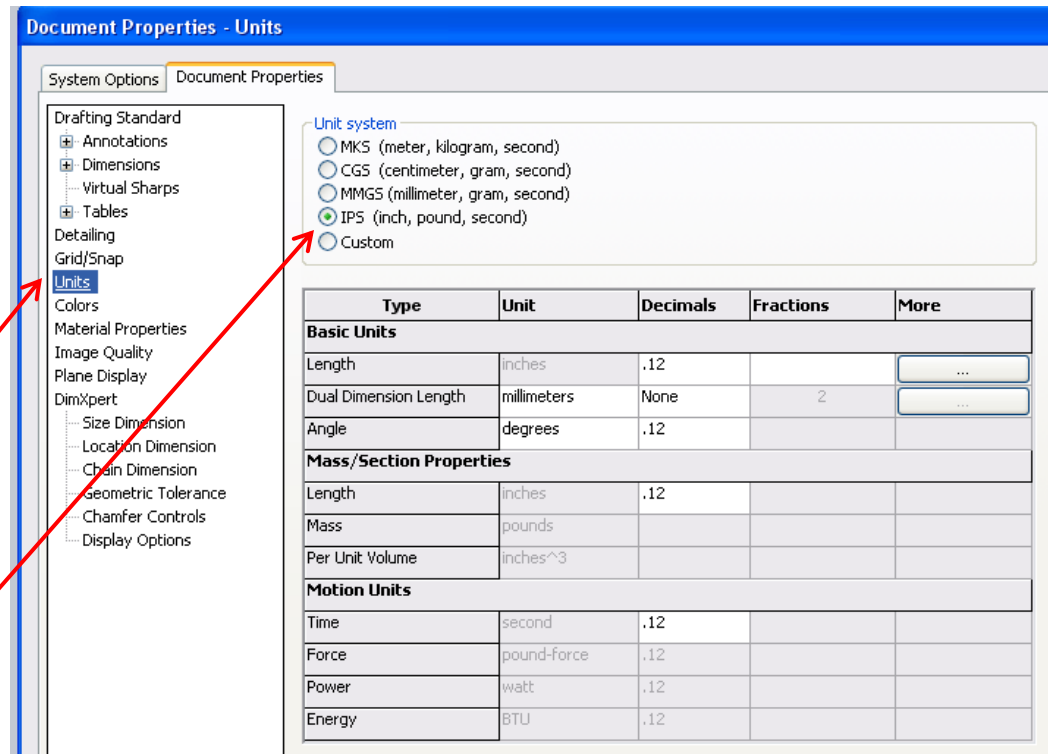

How do we make sure we use the correct UNITS?

SolidWorks Exercise

In feature manager, RIGHT CLICK on PART1, then choose DOCUMENT PROPERTIES. Always remember to do this FIRST when making a new part.
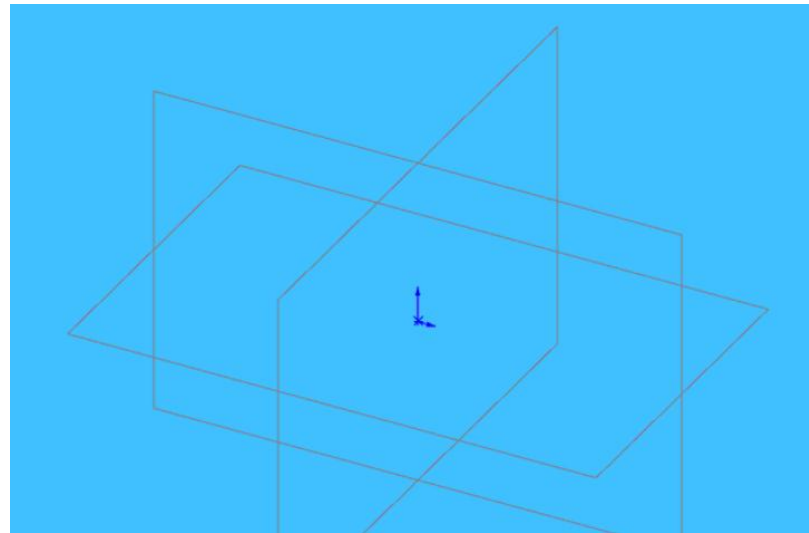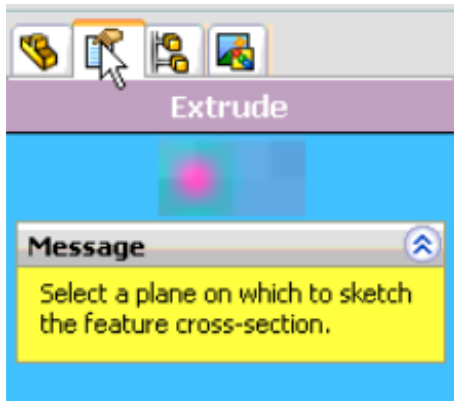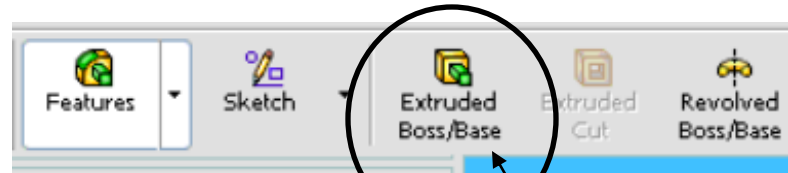

Click on UNITS, then choose IPS for inches.

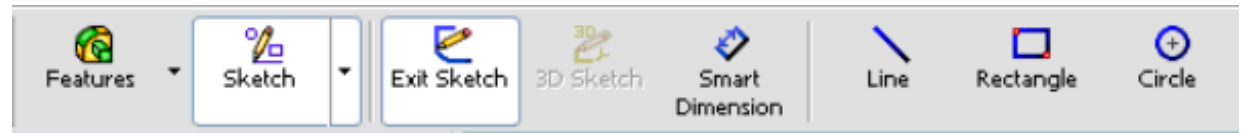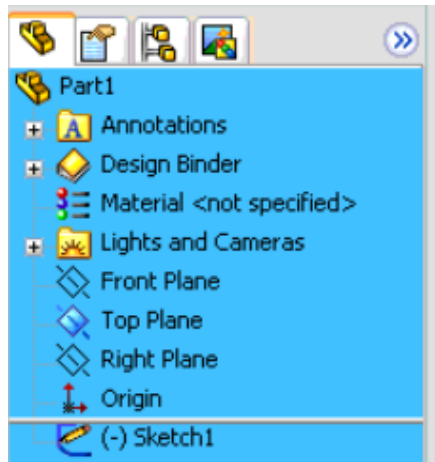Then choose OK

SolidWorks Exercise

Let's begin by selecting EXTRUDED Boss Base

You should notice that your tab will change to property manager asking you to select a plane from the view.

Extrude

Message

Select a plane on which to sketch the feature cross-section.

SolidWorks Exercise

Go ahead and select the horizontal plane shown.

Part1
- Annotations
- Design Binder
- Material <not specified>
- Lights and Cameras
- Front Plane
- Top Plane
- Right Plane
- Origin
- (-) Sketch1

Features | Sketch | Exit Sketch | 3D Sketch | Smart Dimension | Line | Rectangle | Circle

You should notice several things happen.
1. You switch to feature manager
2. You have "sketch" buttons on the dynamic toolbar
3. You view below is shown as TOP VIEW! You are looking down on top of the part.
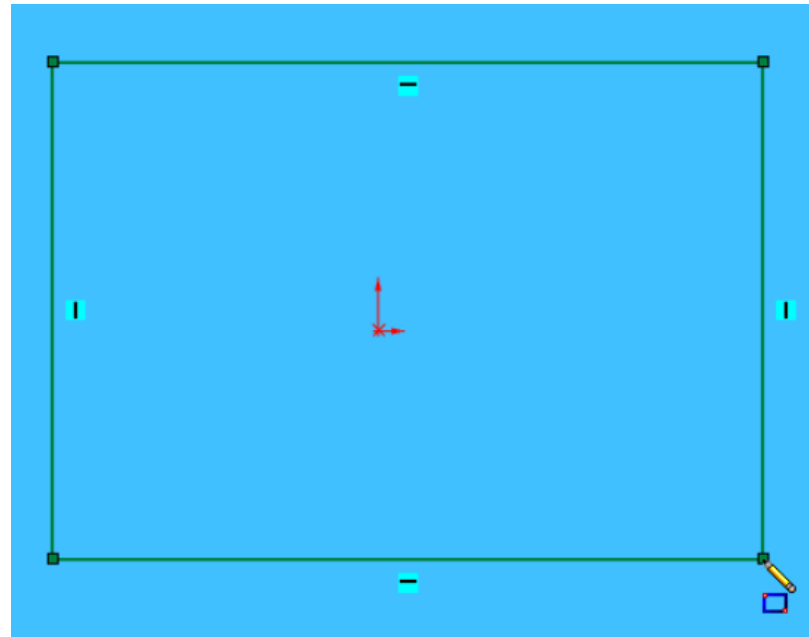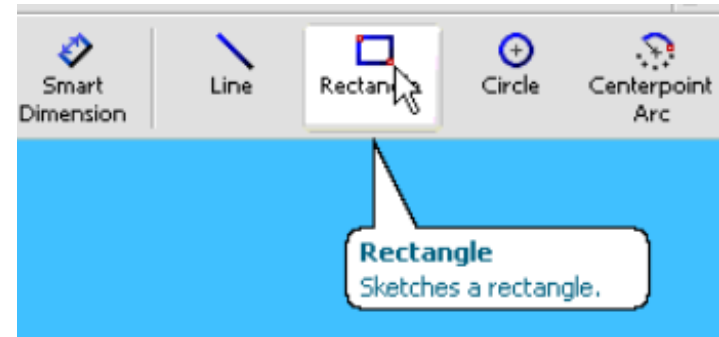
*Top
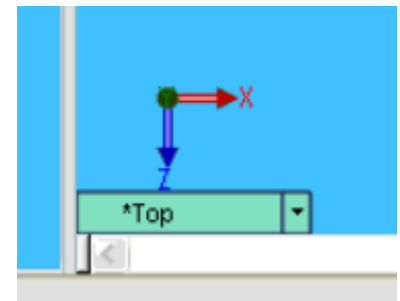
SolidWorks Exercise

Click on Rectangle. You should see your cursor change to a rectangle with a pencil which means you are sketching.

Make a rectangle by dragging the mouse from one corner of the screen to the other. The size does not matter at this point.
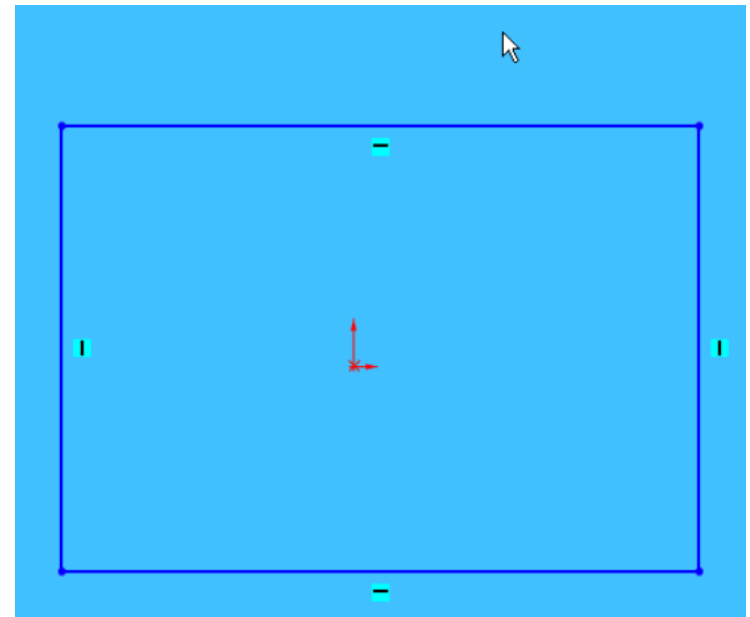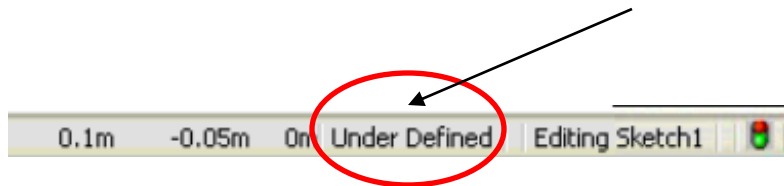
Click the GREEN CHECK in FEATURE MANAGER when complete.

SolidWorks Exercise

If you hit escape, it will get you out of the rectangle TOOL and back to a normal cursor.

We need DIMENSIONS to our rectangle however.

| 0.1m | -0.05m | On | Under Defined | Editing Sketch1 | |

At the bottom right of your screen you will see the figure above. Obviously, we are in the middle of our sketch but it also says, "Under Defined". This means that there are parts of the sketch that are not defined according to location.
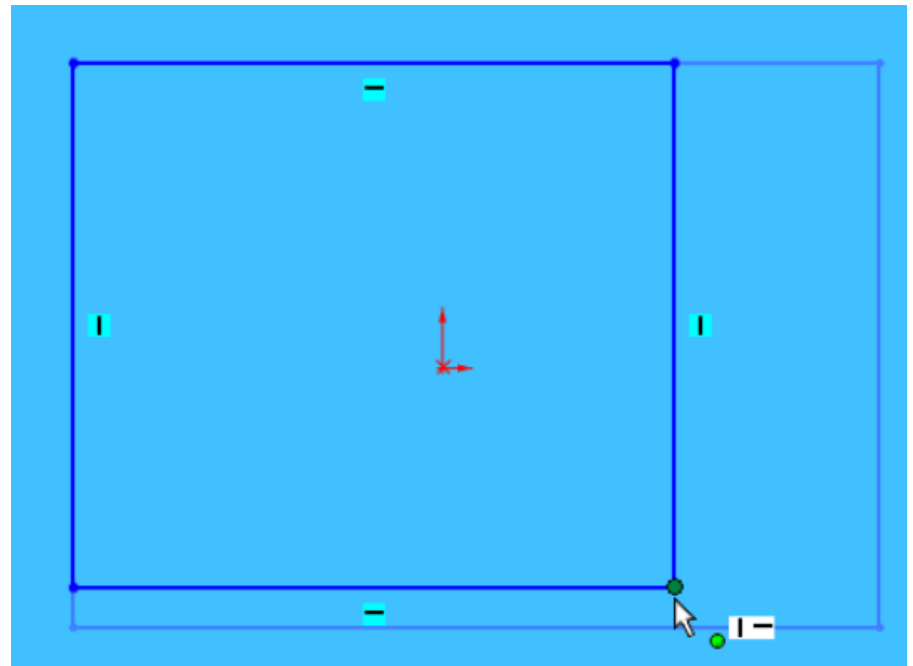
SolidWorks Exercise

Click the bottom right point on the rectangle.
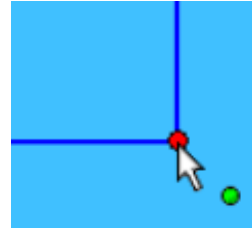
You should be able to move this point around as its location is NOT defined.

This point has degrees of freedom and is NOT constrained.

SolidWorks Exercise

Let's save our work!

Click on the small box next to the far right vertical line. Notice this box turns pink when clicked and that it has a vertical line in it. This means that this line is vertical and when you click on it you can move it left or right. The vertical lines can thus be moved up or down.

Press the DELETE button, then click on the top right point. Notice the line is not defined as a vertical line anymore and the degrees of freedom are extended.

The horizontal line still moves up or down, but notice the vertical line can be moved in any direction as you deleted the relationship.
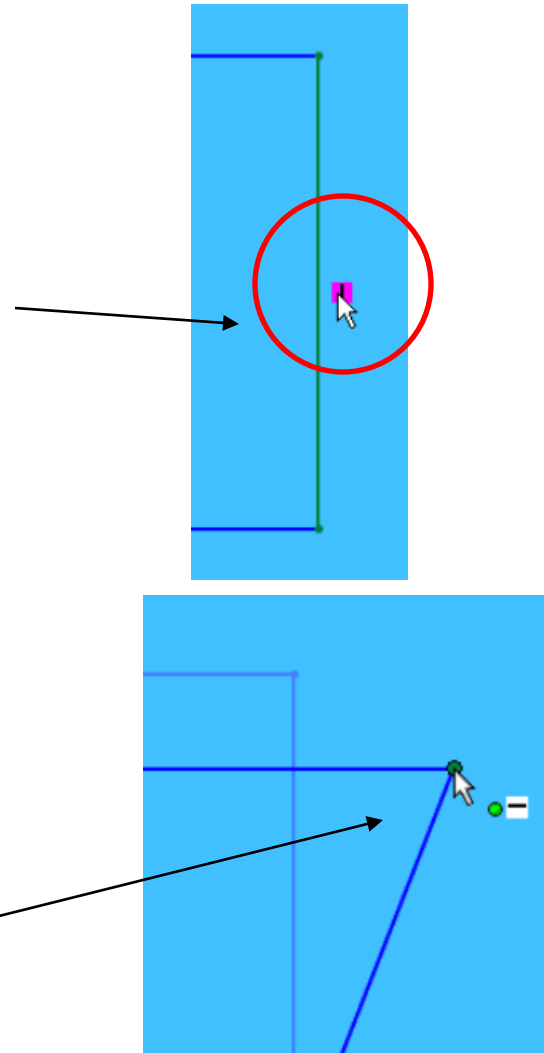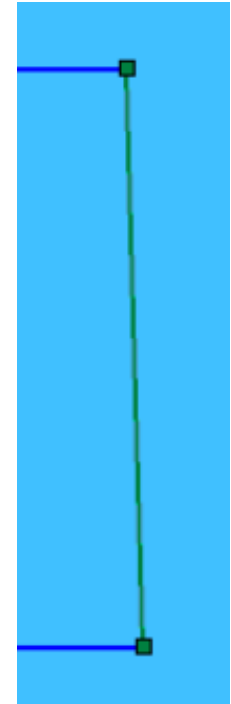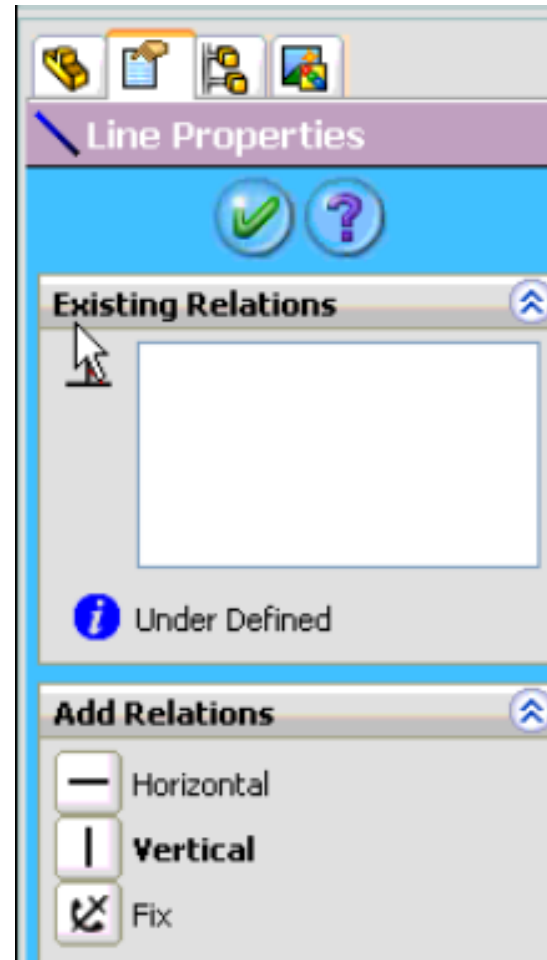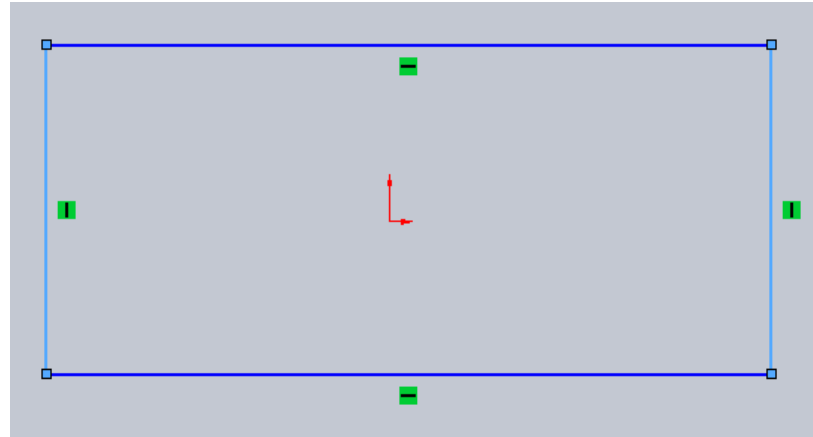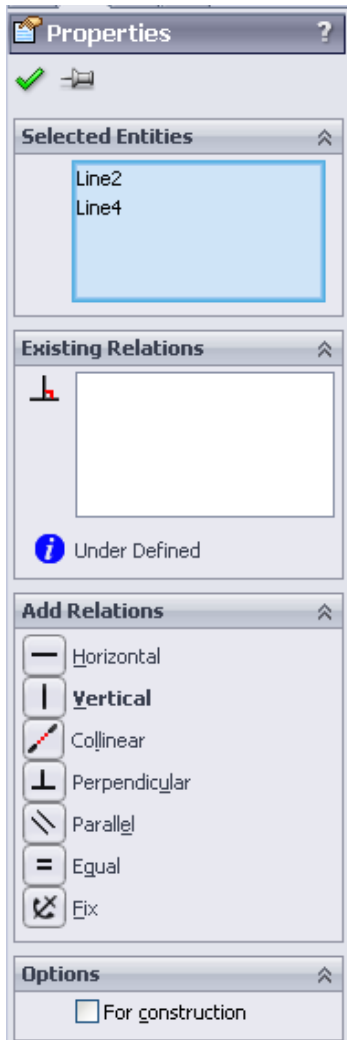
SolidWorks Exercise

You can add relations by clicking on a line, turns green.

Line properties will appear and you can click on the VERTICAL button on the ADD RELATIONS windows to constrain the line to just the vertical direction.
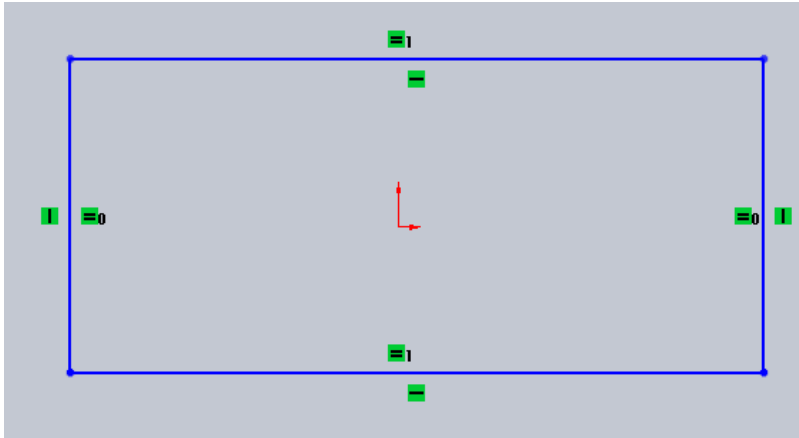
SolidWorks Exercise

Click on the RIGHT vertical line, then hold down the CTRL button. Click the left vertical line. You should see a list of available relations that you can add to BOTH line. Let's choose EQUAL and VERTICAL. Click the green check when finished.
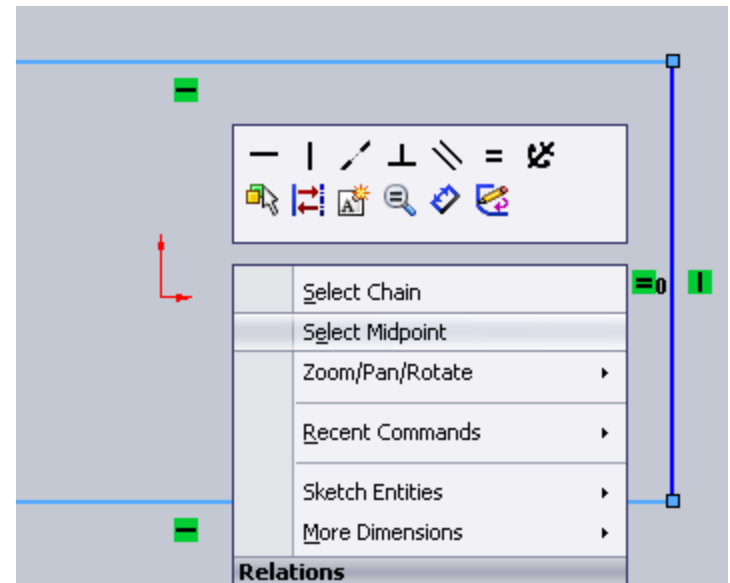
Do the same thing for the top and bottom horizontal line, except add a horizontal relation.

SolidWorks Exercise

You should see tiny green boxes next to the lines. If you place your mouse over them it will tell you what relation that element has.

Unfortunately, our design is still blue which means we are still UNDER DEFINED. In fact, we need to fix our design to the origin in some way. Depending on what design aspect come later it is often desired to fix the origin at the CENTER of the design. RIGHT CLICK on the bottom horizontal line and choose SELECT MIDPOINT. Hold down CTRL and select the ORIGIN. Add a vertical relation.
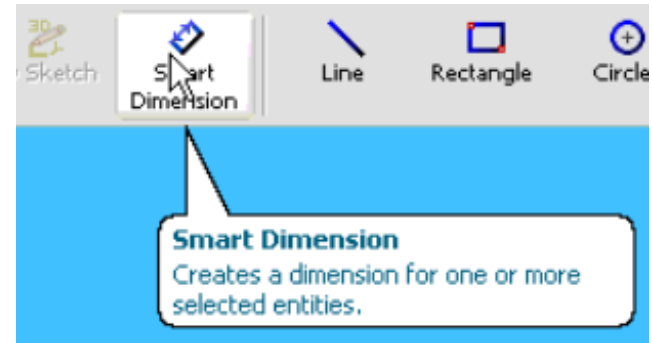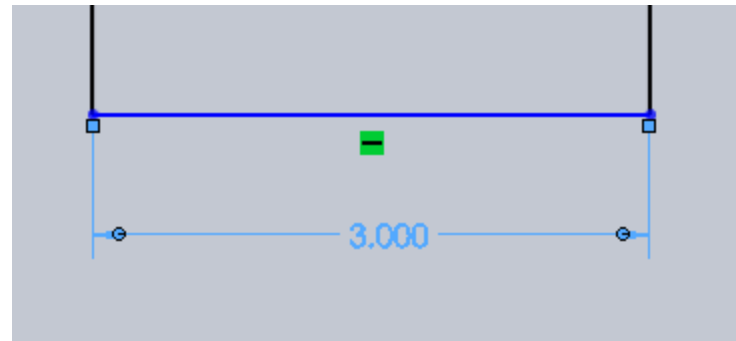
Do the same thing for the sides by adding a horizontal relation to the origin.

SolidWorks Exercise

Now we want to add specific dimensions to our drawing. Choose SMART DIMENSION on the dynamic toolbar.

Smart Dimension
Creates a dimension for one or more selected entities.

Click the bottom horizontal line of our rectangle and drag the dimension down.

You can change the length of the line by using the slider bar or you can simply enter in 3 inches in the box shown. Enter the value 3 then click the green check. If the box is way too big or small, press "f" on the keyboard to fit.
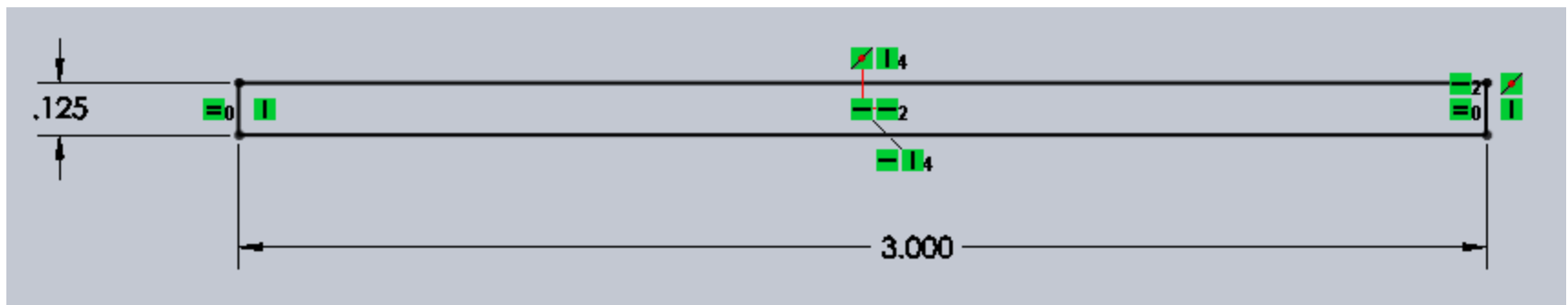
SolidWorks Exercise

Now using Smart Dimension again lets dimension the left vertical line..
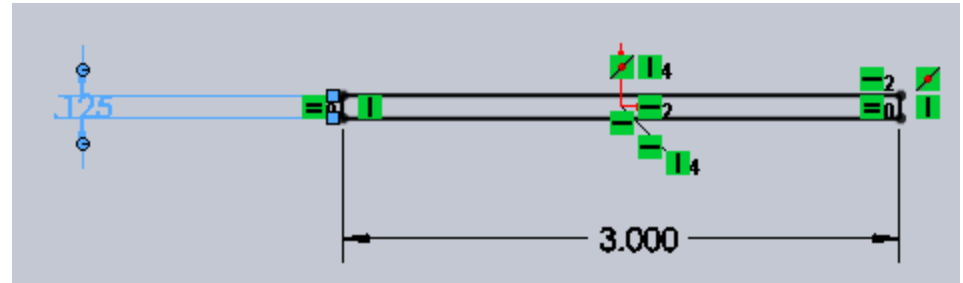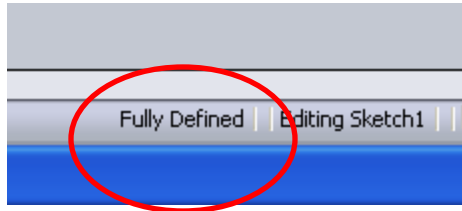

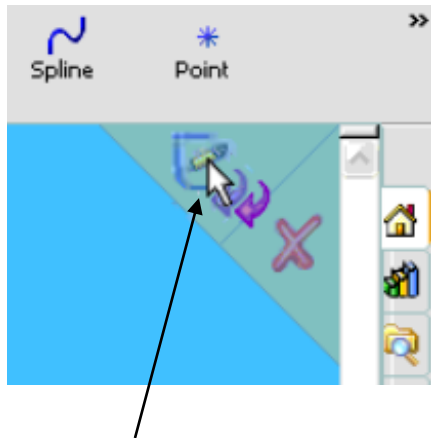Modify the Length to be 0.125 units as shown in the original drawing.

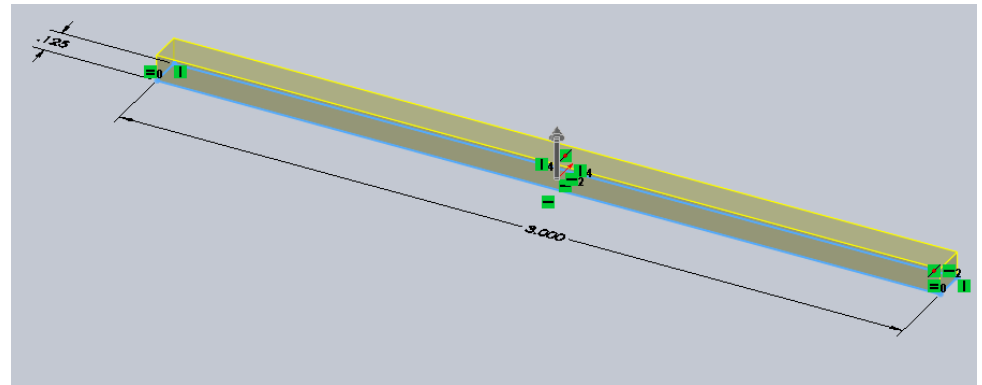Hit "f" on the keyboard to view the design better

SolidWorks Exercise

Notice that after you added relations and added dimensions the design color turned black. This means it is FULLY DEFINED. ONLY after seeing this can we then EXTRUDE.

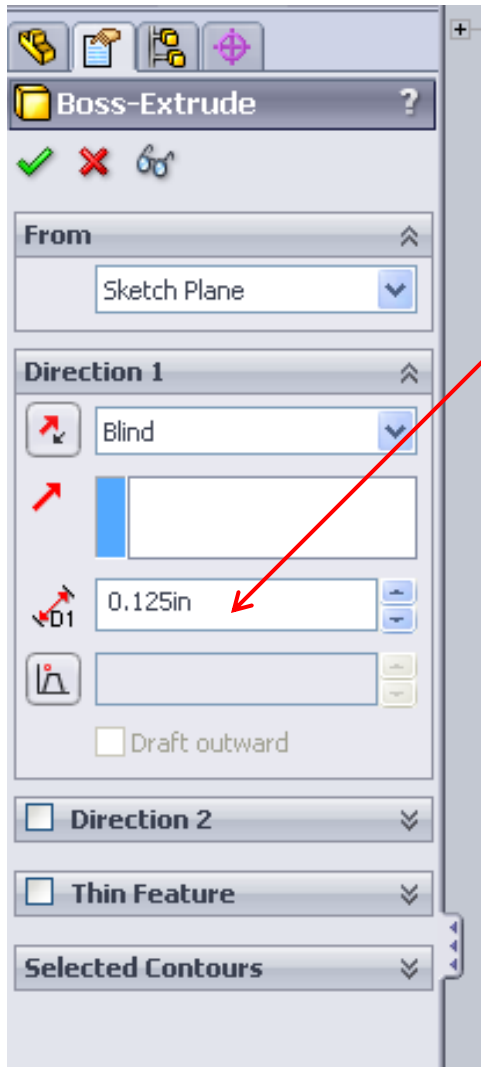Fully Defined | Editing Sketch1

Spline   Point

Click in the top right corner to EXIT the SKETCH.

If you hit EXTRUDE earlier, the part will automatically become 3D. If not, click on the part in feature manager then click EXTRUDE BOSS/BASE at the top.

SolidWorks Exercise

After EXTRUDE was chose, the PROPERTY MANAGER should have opened. In the BOX, "D1" it is asking for a width distance. Since the width and thickness are the same we will enter 0.125 in. Click the green check to finish and view the finished part.
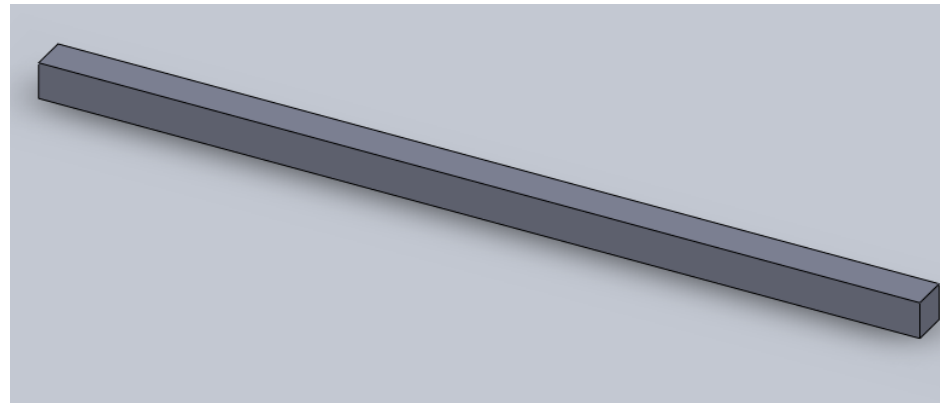
SolidWorks Exercise

- To ZOOM you can use the mouse wheel or hit the Shift button and move mouse

- To ROTATE, press in the mouse wheel and hold.

- To PAN you use the Ctrl button and move mouse

- To fit a picture to a window simply press the "f" letter key on the keyboard to FIT.

Mechanical Drawings

So far we have been dealing with creating parts and assemblies in SolidWorks, however, when you go to get a part machined, you will need to create a mechanical drawing of each of your parts (and assemblies).

Mechanical drawings are important because they allow those who are technically trained to reconstruct your 3D geometry from 2D drawings.
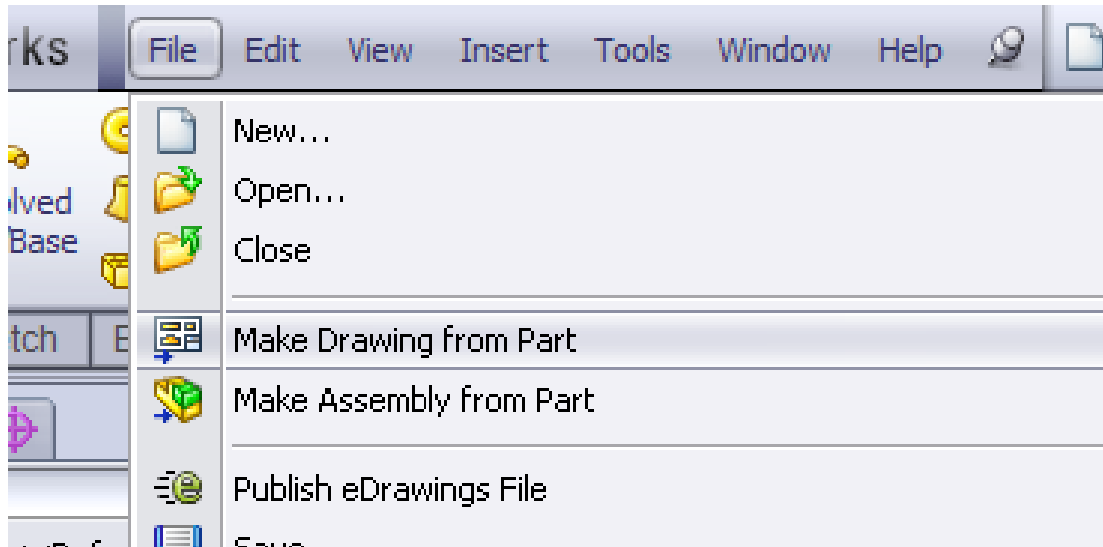
Drawings in Solidworks

Fortunately, SolidWorks makes it very easy for us to create drawings from a part or assembly file.

In fact, if built properly, SolidWorks will also dimension the entire part and assembly for us…something that saves a lot of time!
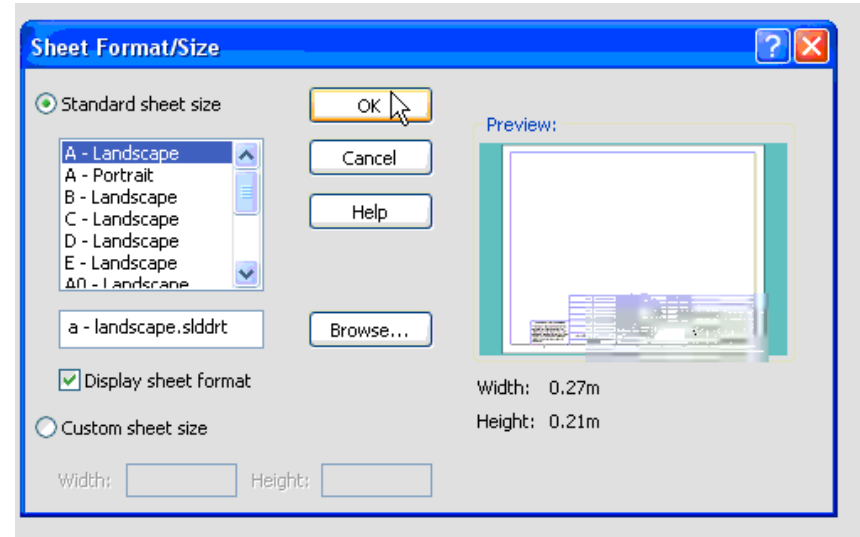
Drawings in Solidworks

Once we open or finish the part, we choose "make drawing from part" from the file menu at the top which will automatically create a drawing file from our part:

Let's do this! If it asks you to save then save your file.

Drawing Format

At first you see several different formats that are set up for you. Choose A-Landscape.

Drawings in Solidworks

However, before we can begin placing views, it is important to set our projection style to Third Angle (in order to have the projections behave as we expect)

To do this, right click anywhere on the sheet and click on Properties (or you can right click on the sheet in the Feature Manager)

Drawings in Solidworks

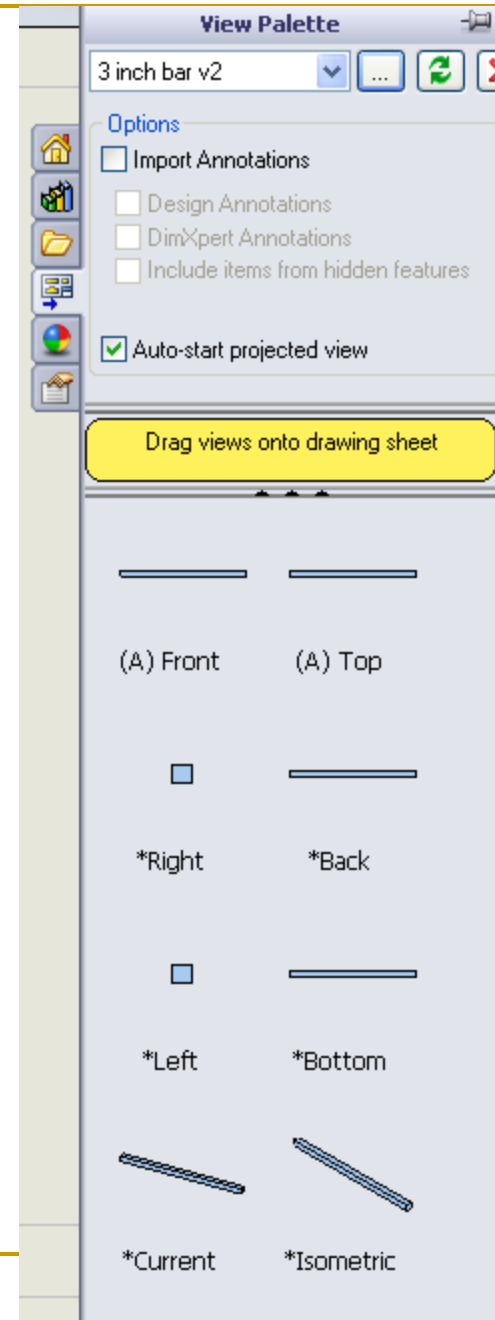This will open the Sheet Properties window:
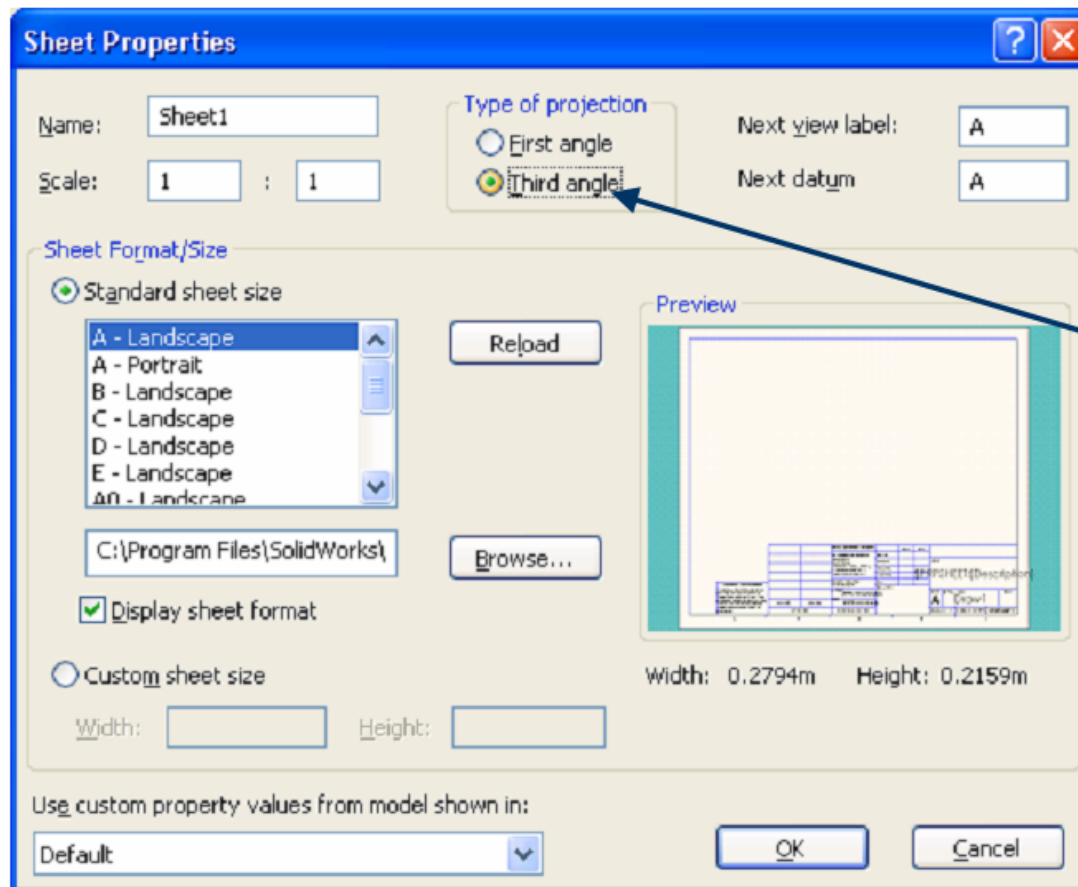
Make sure Third angle is selected for Type of projection.
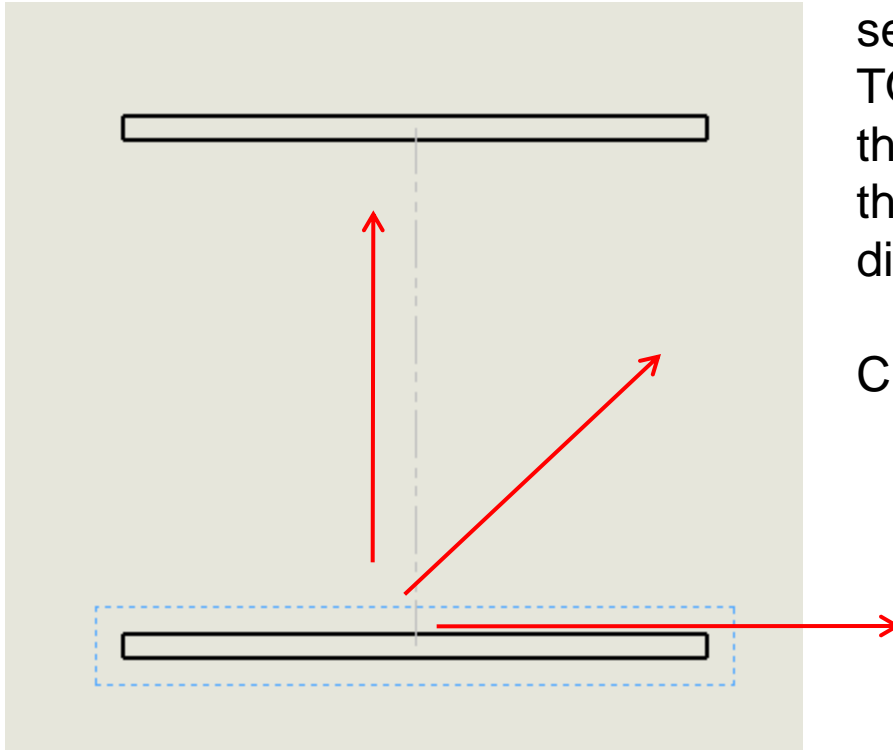
Drawings in Solidworks

Open up the palette on the RIGHT side menu

Click and HOLD the view you want and drag it into the drawing field.

Choose the FRONT view and drag it to the bottom right of the drawing field.

View Palette

3 inch bar v2

Options
- ☐ Import Annotations
  - ☐ Design Annotations
  - ☐ DimXpert Annotations
  - ☐ Include items from hidden features

☑ Auto-start projected view

Drag views onto drawing sheet

(A) Front   (A) Top

*Right   *Back

*Left   *Bottom
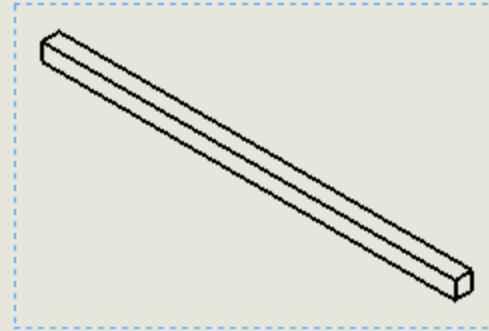
*Current   *Isometric
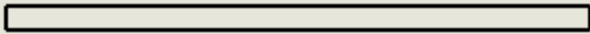
Drawings in Solidworks

As soon as the front view is in position move the mouse UPWARD. You will see another design, this is actually the TOP view. Click to place the top view then return to the front view and move the mouse sideways. Repeat, then go diagonal.
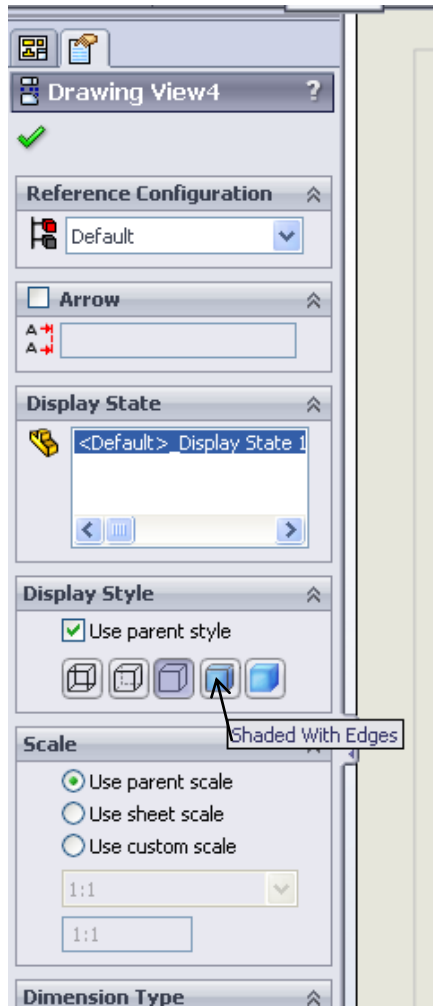
Click the green check when finished.
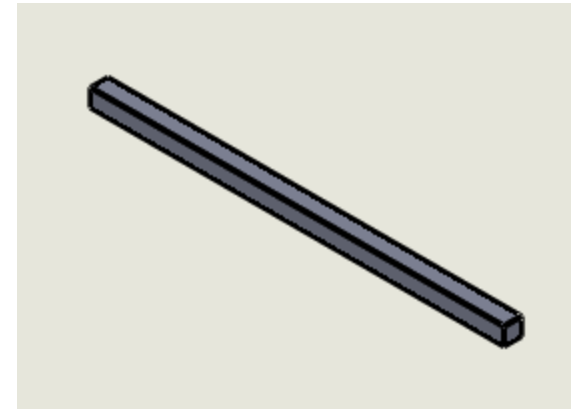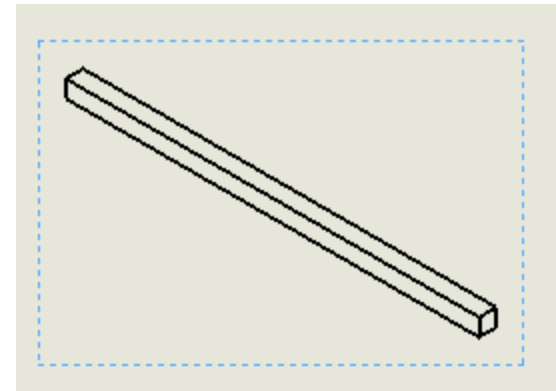
Drawings in Solidworks

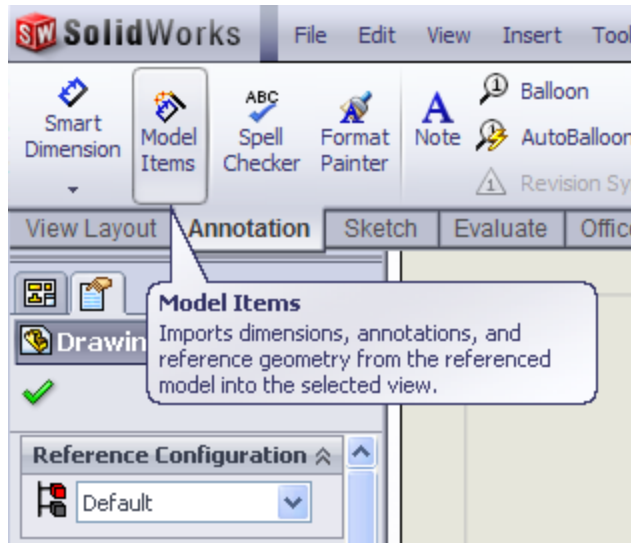You should now see 4 views (front, top, side, & isometric).

Drawings in Solidworks

Click on the ISOMETRIC view, then choose SHADED WITH EDGES under DISPLAY STYLE on the left. In drawings, we always shade in the ISOMETRIC view
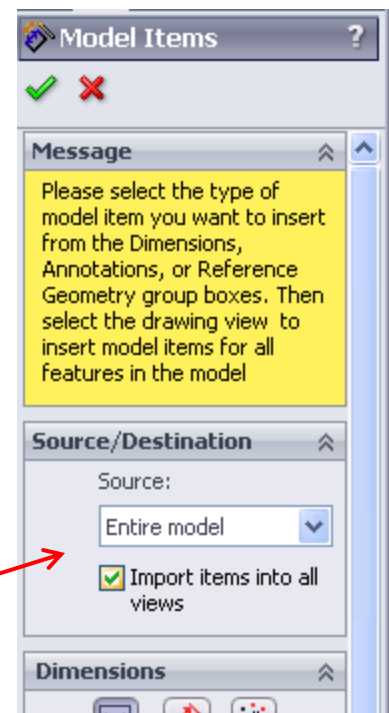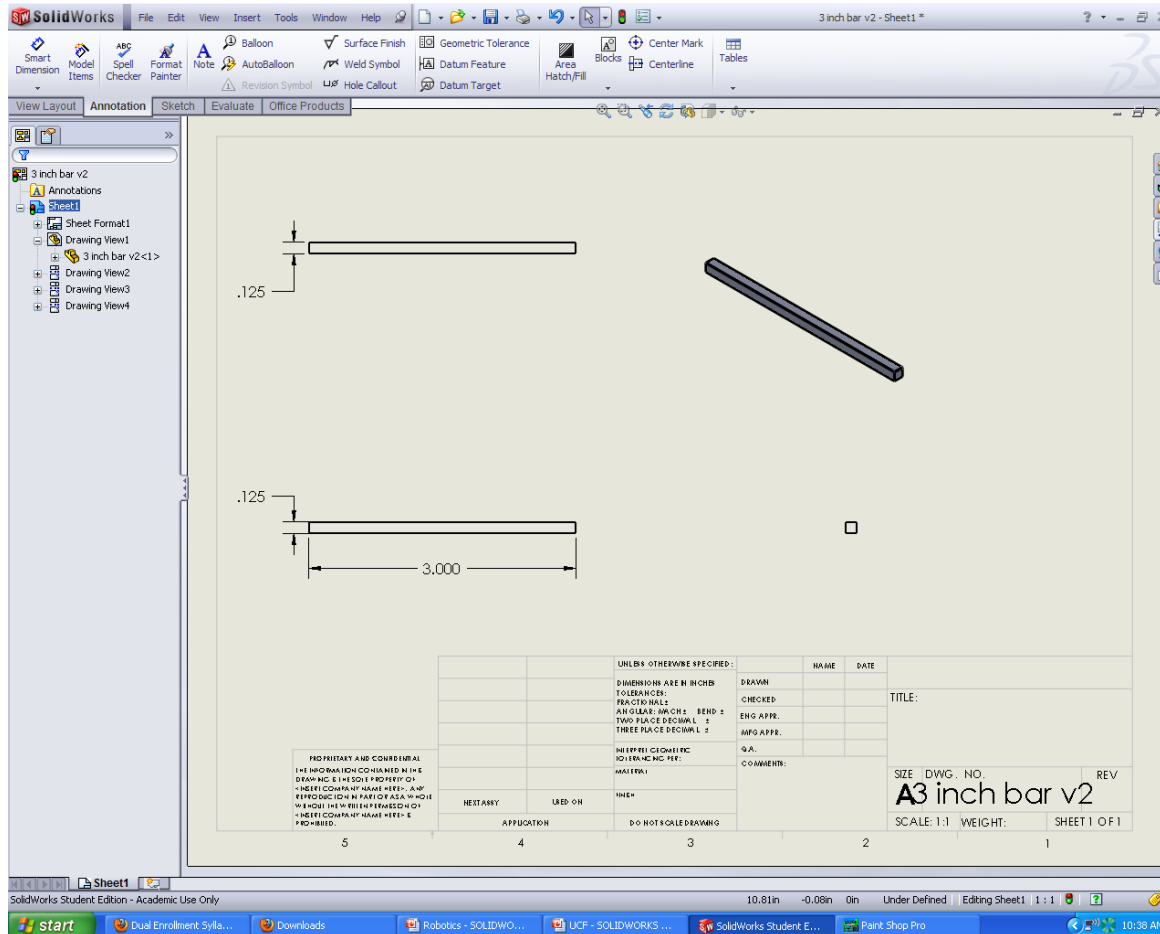
Drawings in Solidworks

Click on the FRONT VIEW, then within the ANNOTATIONS tab at the top click on MODEL ITEMS.

Under source/destination choose , "entire model" from the drop down menu. Then check the box, import items into all views. Dimensions will then be added to the drawing. If you don't see dimensions on each view, try adding them separately.

Drawings in Solidworks

Save the drawing, then print it out. Write your name in the title block in the bottom right.