

Benha Faculty of Engineering
Mechanical Engineering Department

M1382 : Computer Aided Design CAD
First Semester 2018, Y3

Lecture No. 02

Presented by:
Mahmoud Magdy



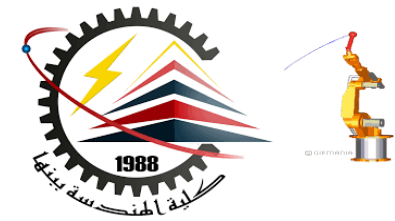


Week	Topics
1	Introduction
2	Introduction to CAD (Solid Modeling)
3	Part modeling
4	Finite element analysis (FEA)
5	Parts assembly using SolidWorks
6	Basic concepts of engineering drafting
7	Linear Static Analysis
8	Adaptive Analysis and Mesh Control
9	Modal Analysis
10	Design Optimization
11	Case study 1
12	Case study 2
13	Co-simulation SolidWorks and ADMS software
14	Project Discussion



Basics and Modeling Fundamentals

Ref : SolidWorks Teacher Guide

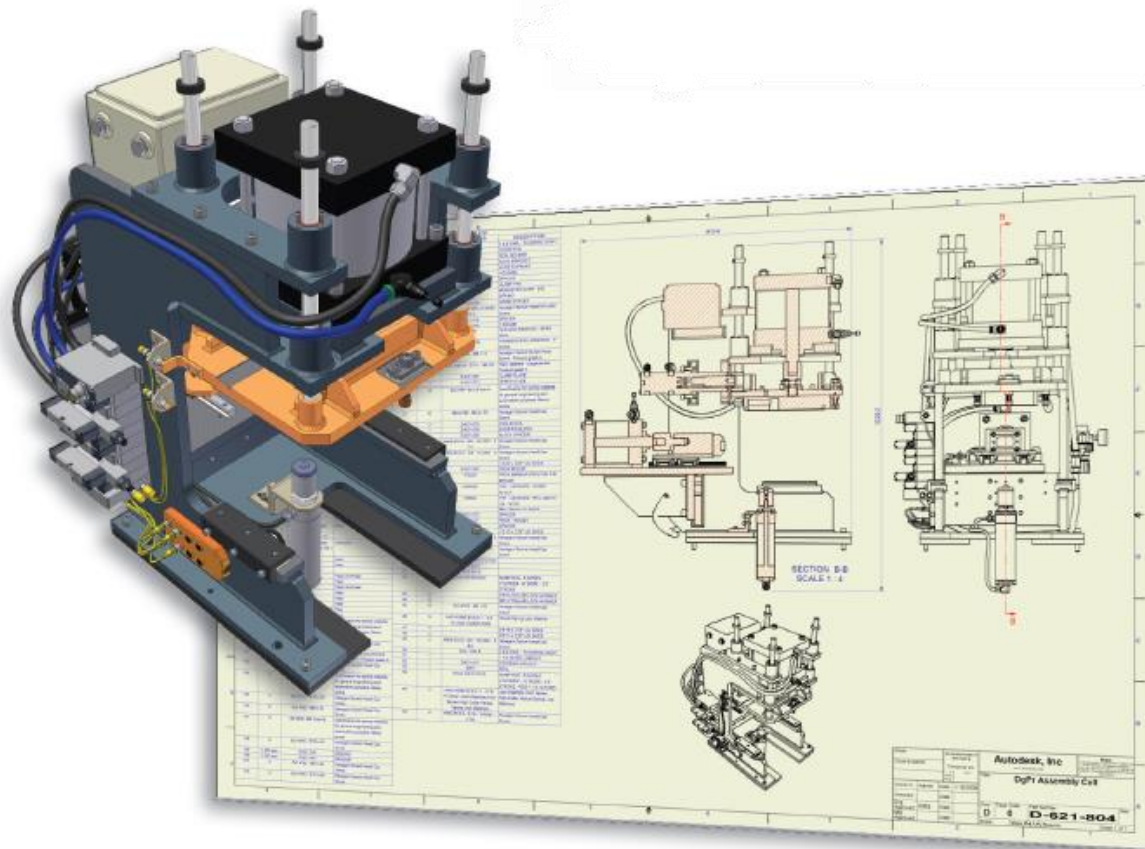


SolidWorks

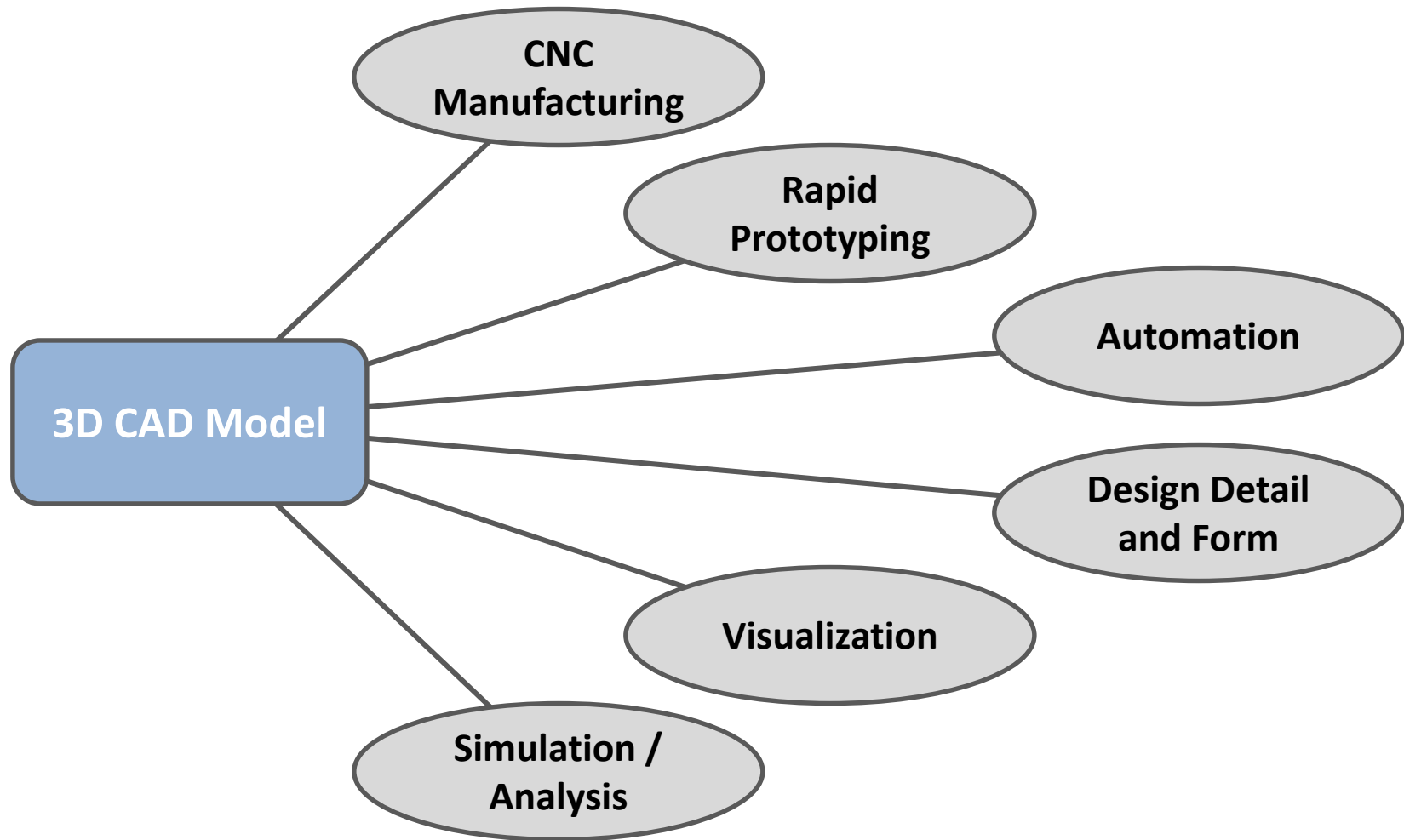
- SolidWorks is a 3D solid modeling package which allows users to develop full solid models in a simulated environment for both design and analysis.

What is 3D Modeling

In SolidWorks, you sketch ideas and experiment with different designs to create **3D models** of the **real physical object**.



3D Design Use



The SolidWorks Model

- **The SolidWorks model is made up of:**

- **Parts**



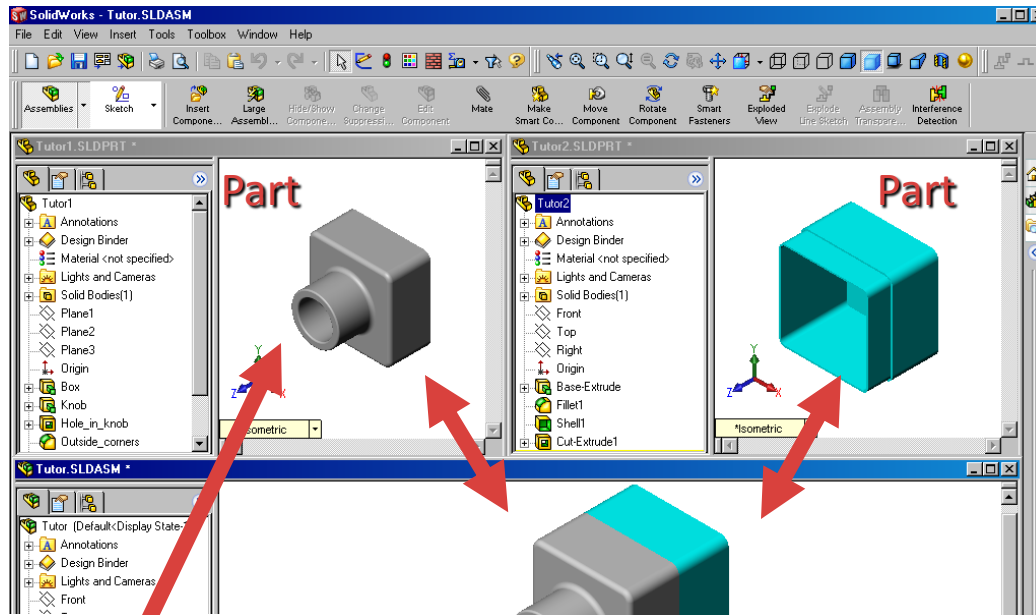
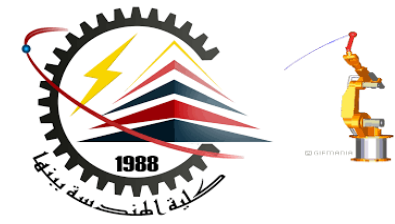
- **Assemblies**



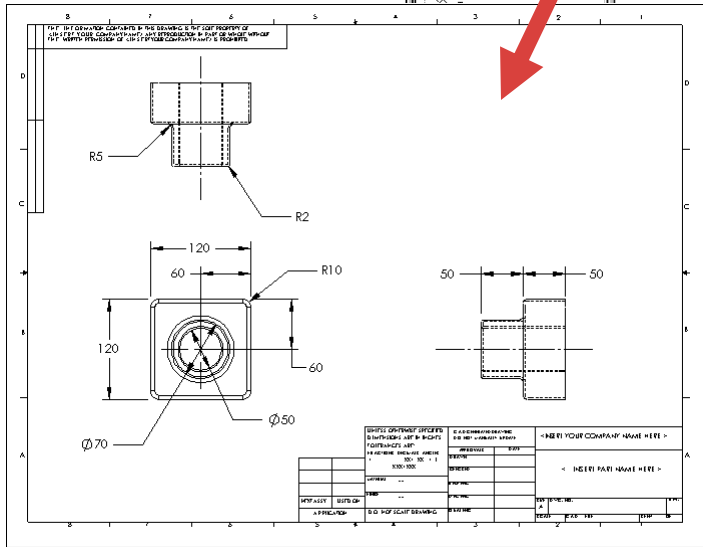
- **Drawings**



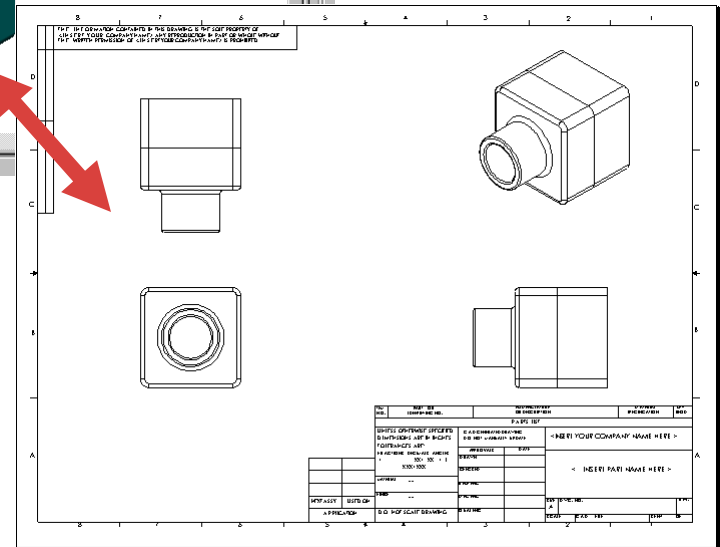
The SolidWorks Model



Drawing



Drawing





SolidWorks Components - **PARTS**

Before we begin looking at the software, it is important to understand the different components that make up a SolidWorks model.

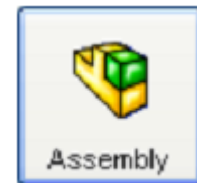


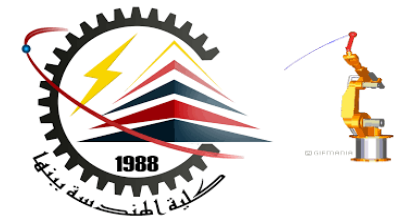
- ✓ The first, and most basic element of a SolidWorks model is **a Part**.
- ✓ Parts consist of primitive geometry and features such as **extrudes, revolutions, lofts, sweeps**, etc.
- ✓ Parts will be the **building blocks** for all of the models that you will create



SolidWorks Components - **Assemblies**

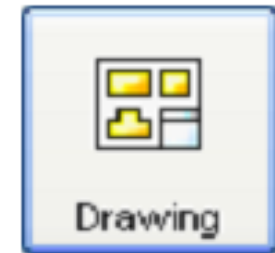
- The second component is the assembly. Assemblies are **collections** of parts which are assembled in a particular fashion using mates (constraints).
- Any complex model will usually consist of one, or many assemblies.



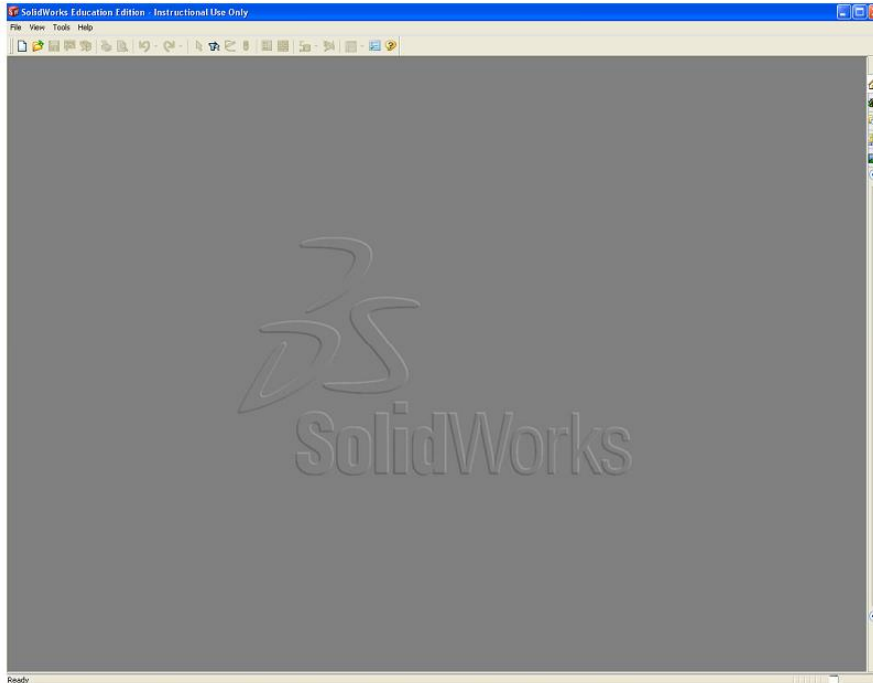


SolidWorks Components - **DRAWINGS**

- The third, and final component in SolidWorks is the Drawing.
- A drawing is the typical way to represent a 3D model such that any engineer (or manufacturer) can recreate your part.

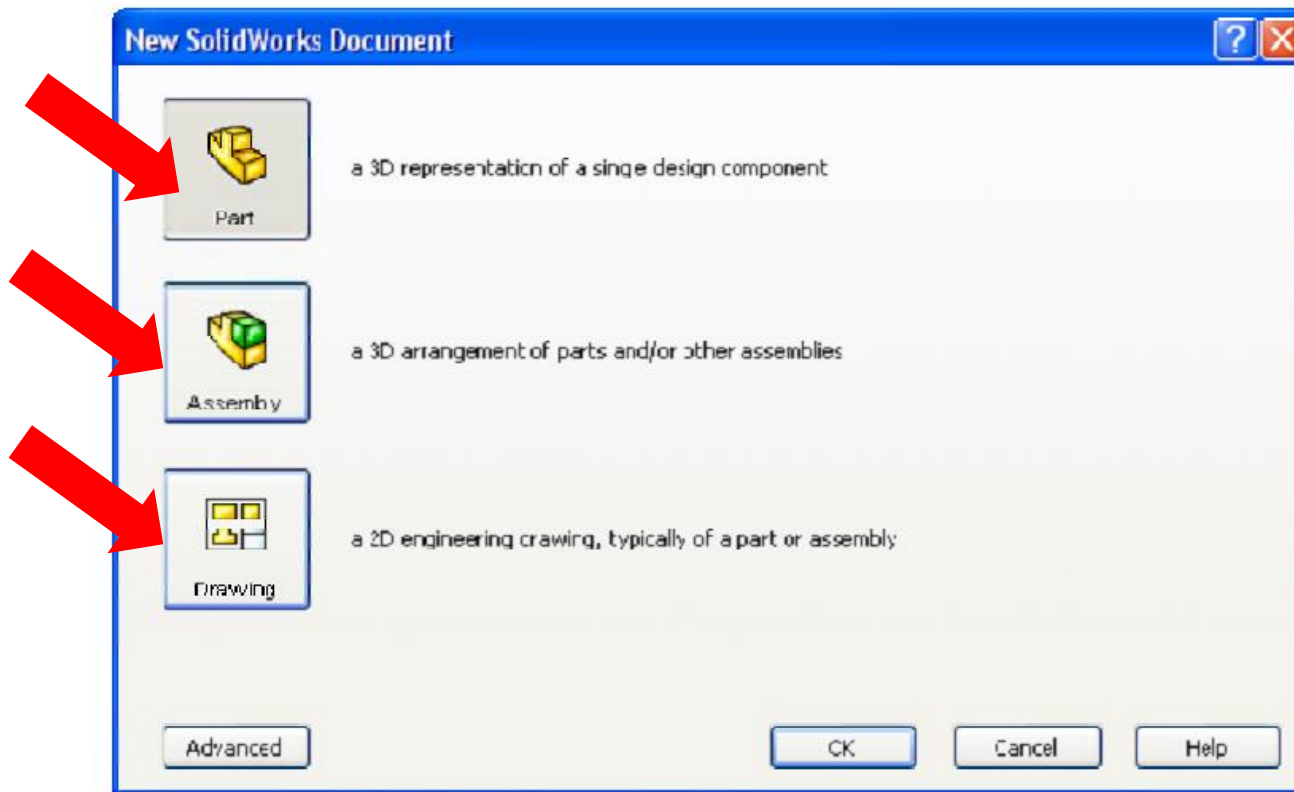


SolidWorks – Let's Begin

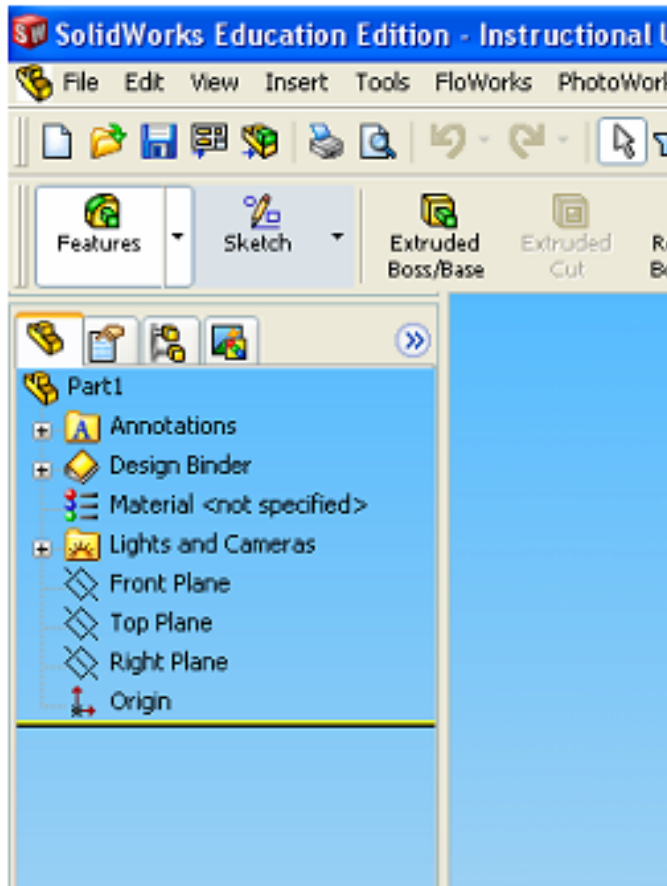


- By default, no file is opened automatically when you start the program.
- To create a new file, **click on File** > New or click the New File icon in the main toolbar.
- This will open the New SolidWorks Document wizard.

SolidWorks Tour

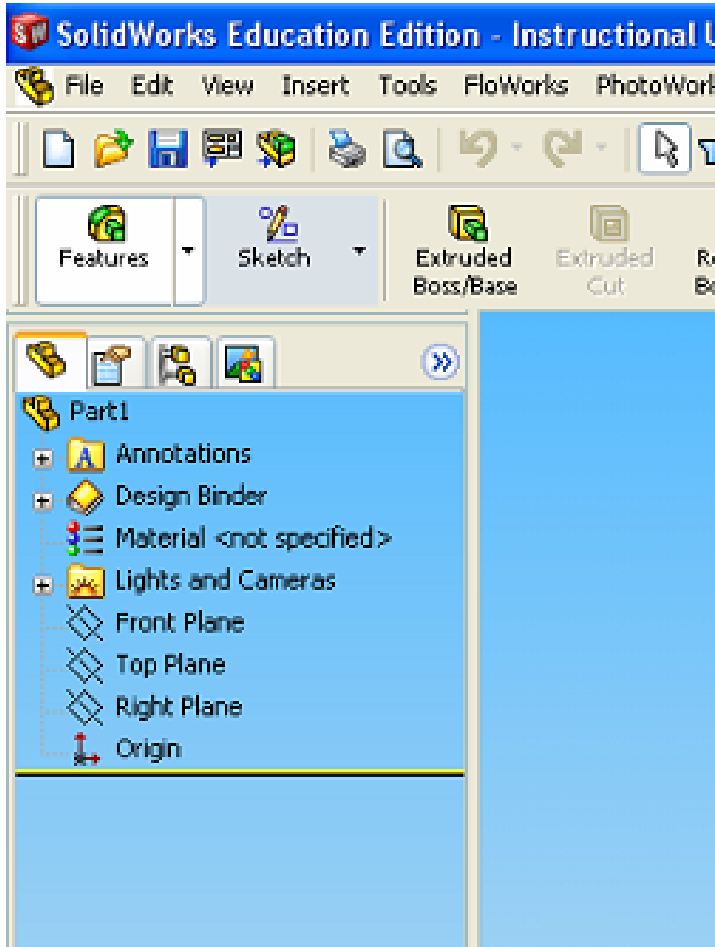


SolidWorks Tour



- Let's begin by creating a new part. To do this, **click on Part, then OK**
- Once you do this, you will be brought into the modeling view which should open several toolbars and panes

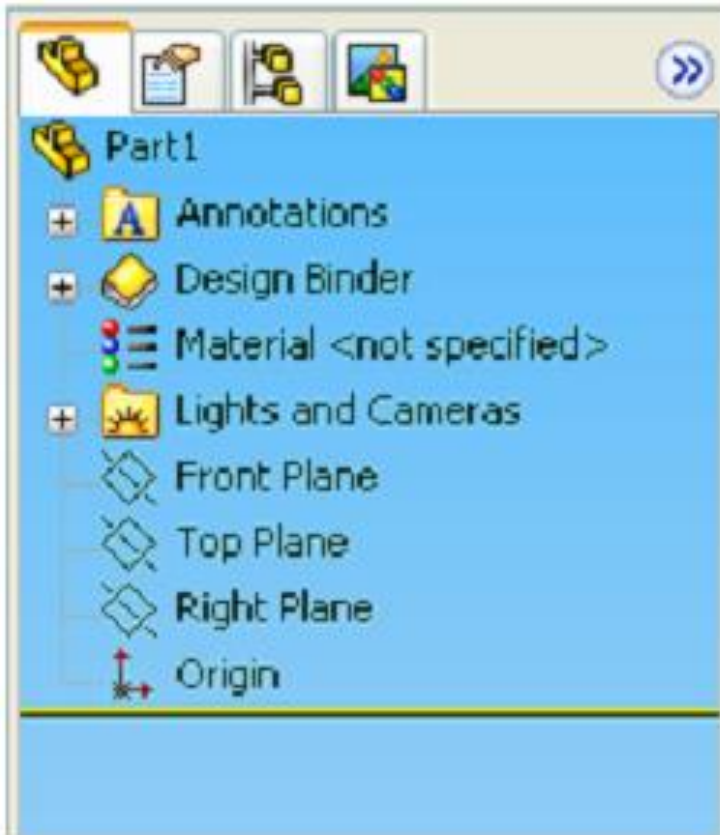
SolidWorks Tour



There are several important parts of the screen that needs to be identified before we continue.

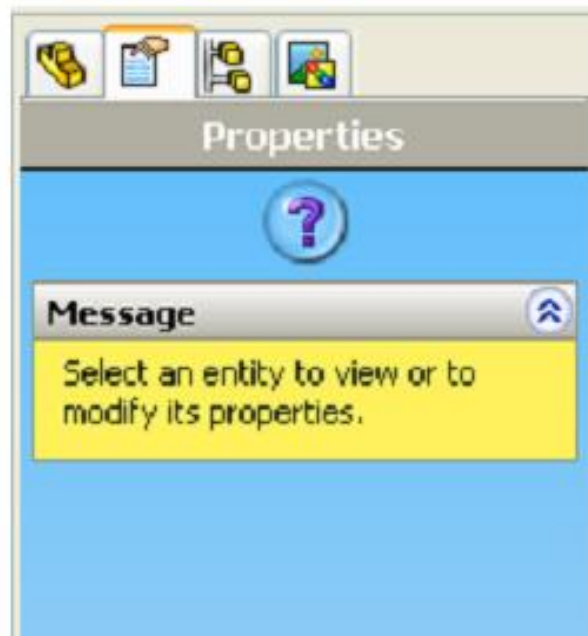
First, the left side of the screen consists of several tabbed panes that provide very important information regarding your model.

SolidWorks Tour



- The first tab, called the Feature Manager, lists all features that have been created within your model.
- This tab is extremely important as it will be from here that you select and change features once they have been created.

SolidWorks Tour



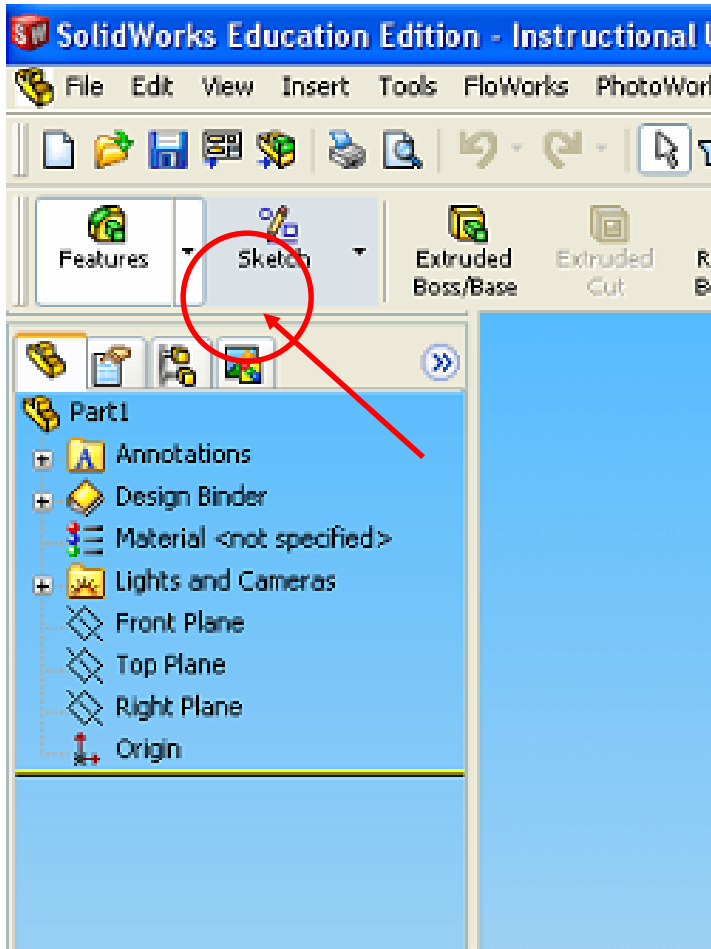
- The second tab, called the Property Manager, allows you to adjust the properties of various entities either during construction, or once it has been created.
- Note that generally you will not need to manually change the tab on the manager window

SolidWorks Tour



- The third tab is called the Configuration Manager and is used to set up different view configurations such as exploded views or 3D section views.
- Usually this will be used once the part has been created and you wish to set up specific configurations for visualization.

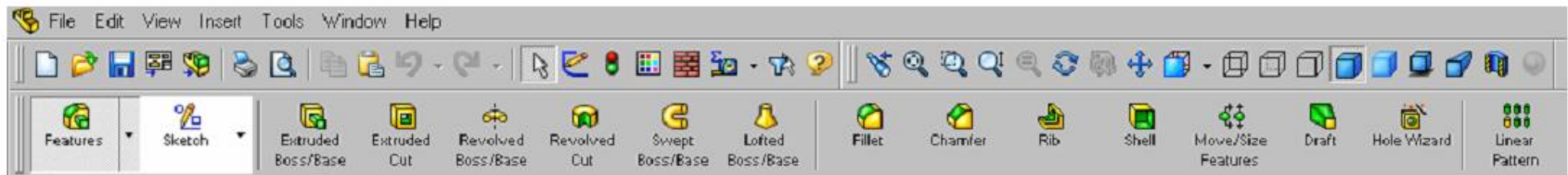
SolidWorks Tour



- There may also be other tabs visible in the manager window.
- Generally any time you load an additional SolidWorks module (such as PhotoWorks, COSMOS Motion, COSMOS Works, etc.) it will create a new tab in this window.

SolidWorks Tour

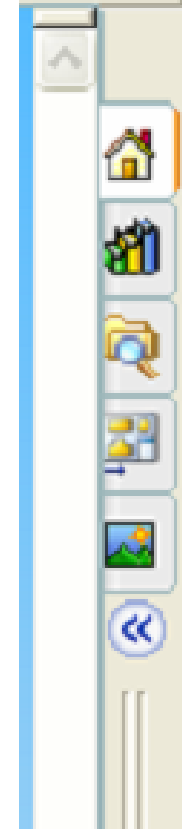
- The next important feature of the interface is the dynamic Toolbar



- The dynamic Toolbar provides access to the most relevant, and frequently use commands in SolidWorks

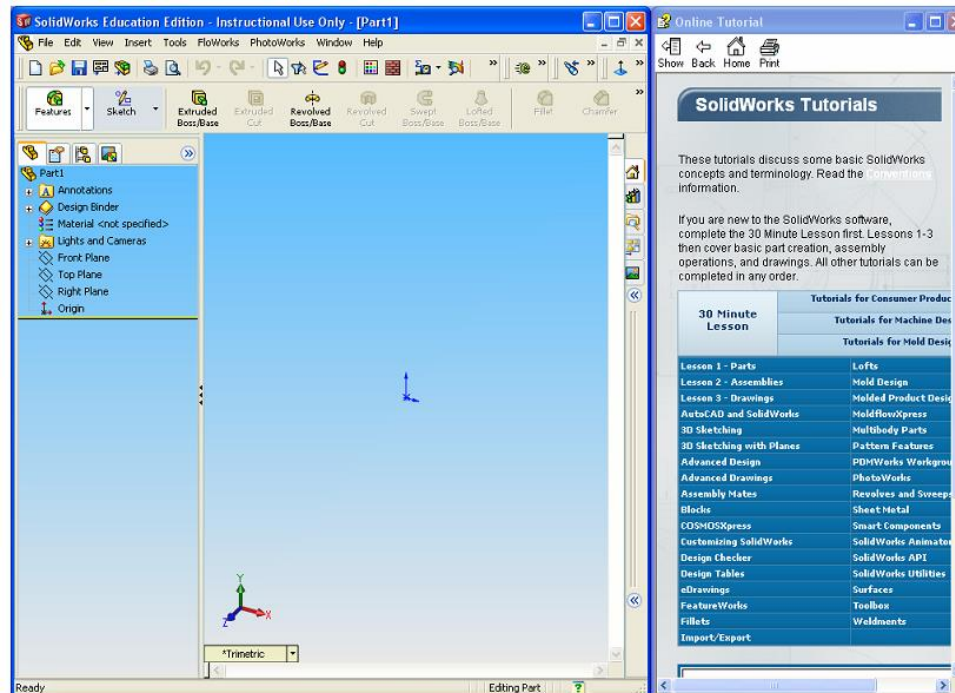
SolidWorks Tour

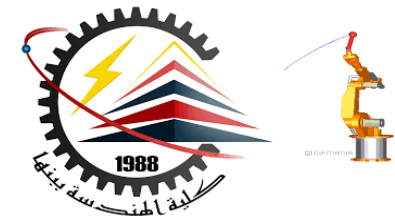
- The last part of the interface which should be noted is the Task Pane on the right side of the screen.
- Using the Task Pane you can view content specific tasks such as importing standard geometry, file explorer, view palette, as well as any plug-in specific information.



SolidWorks Tour

- The last thing that needs to be shown is how to open the SolidWorks tutorials.
- They can be accessed by going to Help > SolidWorks Tutorials.
- The tutorials are very helpful and cover from the most basic features to more advanced analysis and assemblies





SolidWorks Exercise

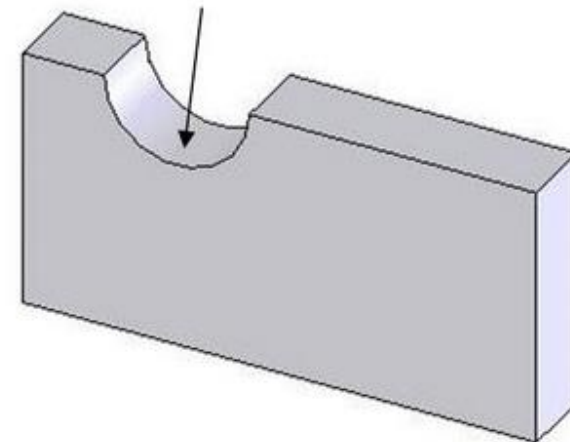
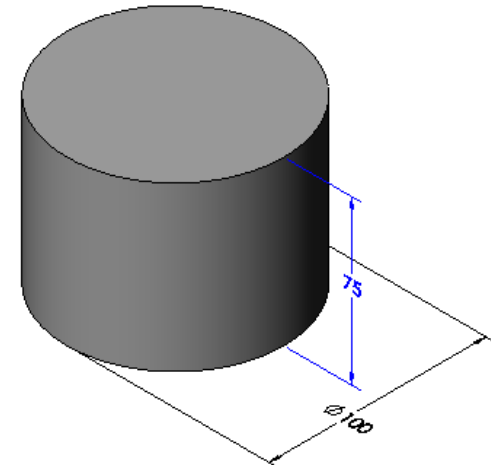
Now that we have explored the interface of SolidWorks, lets create a simple part step-by-step.

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.**

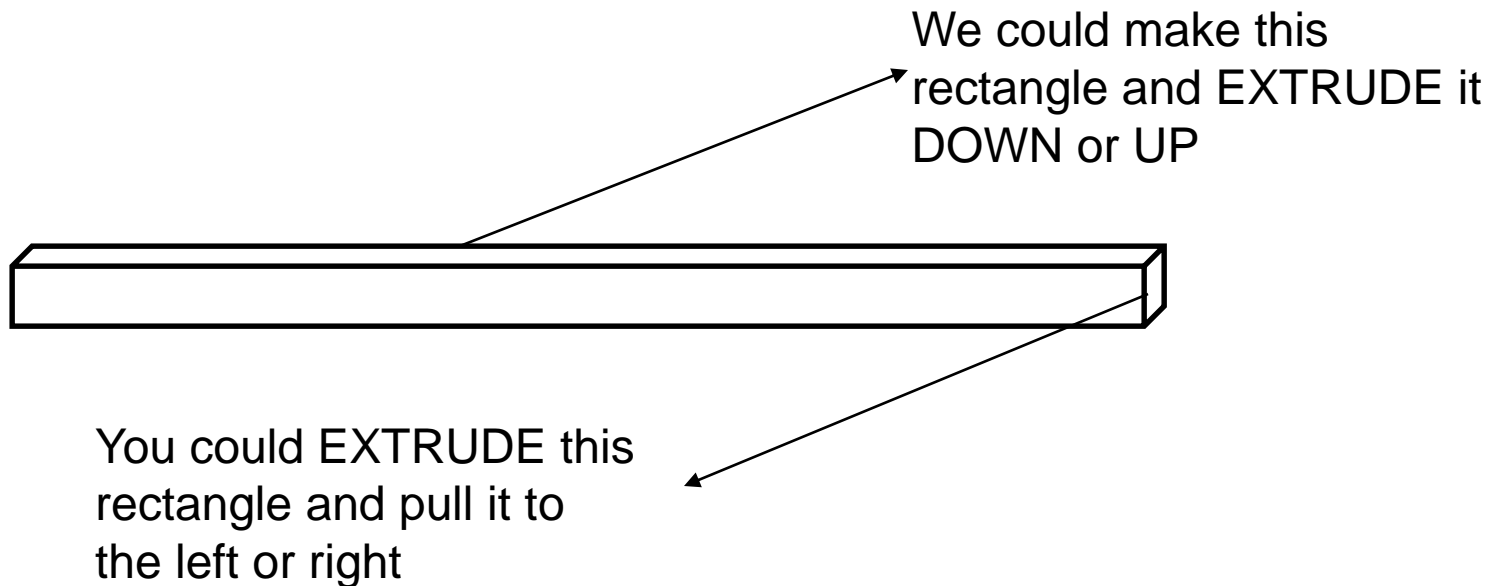
You can extrude to make **a SOLID** or you can extrude to make a **CUT**





SolidWorks Exercise

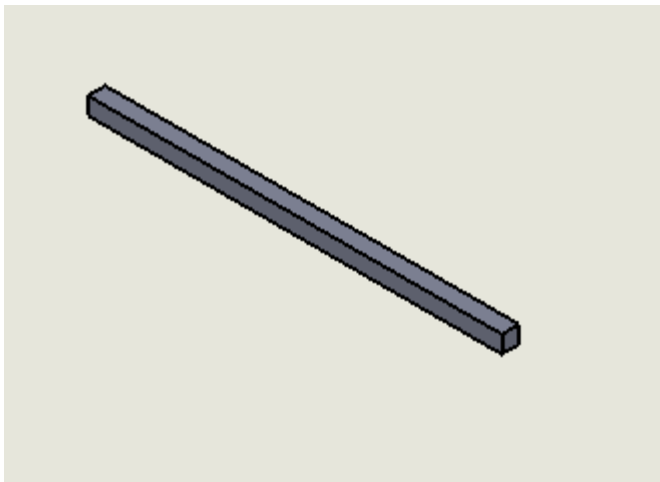
There are MANY ways to EXTRUDE a surface



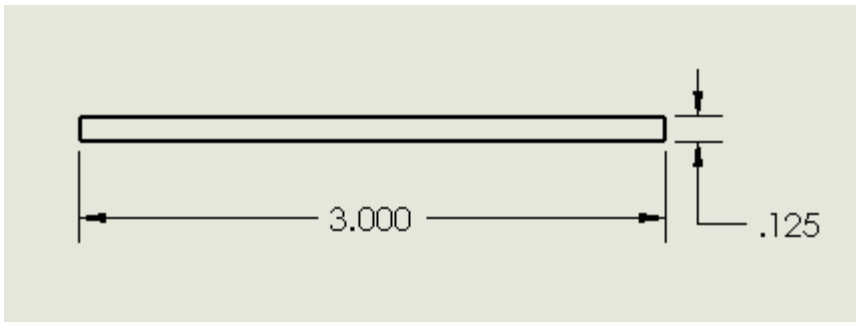
Let's begin. Choose FILE, then NEW PART

SolidWorks Exercise

We wish to model the following part:

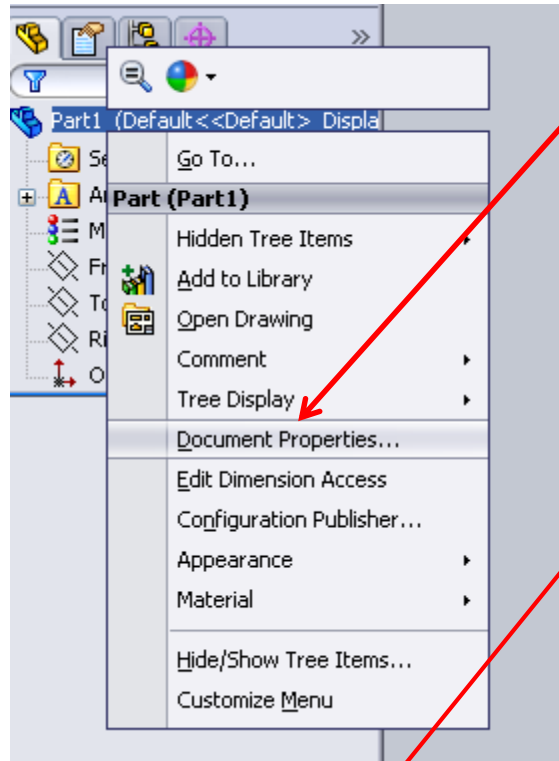
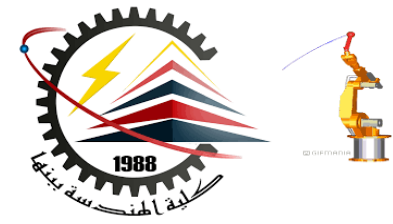


This is a basic 3 inch bar used often time to mount our wheels to our motors. It is obviously 3 inches long, yet 0.125 inches in width.

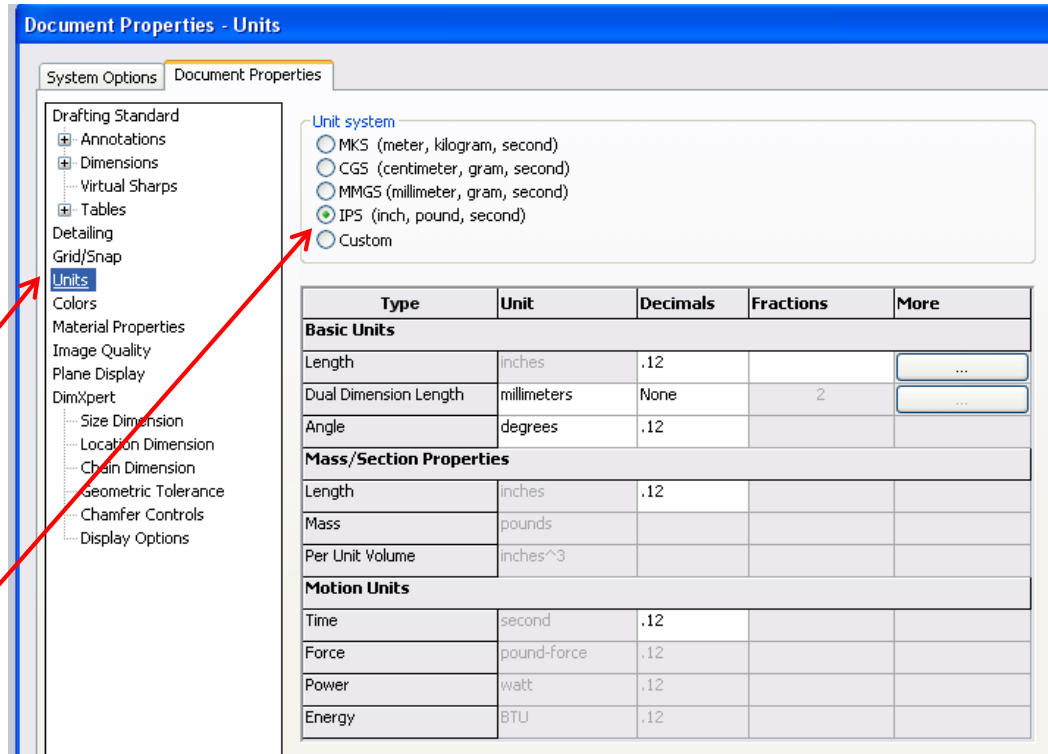


How do we make sure we use the correct UNITS?

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.

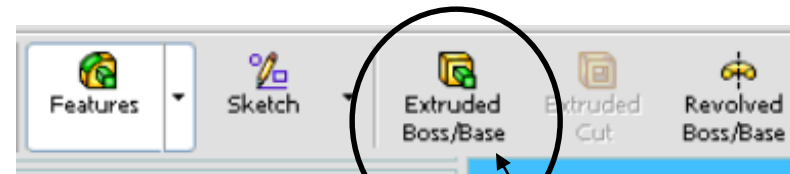
Then choose OK

SolidWorks Exercise

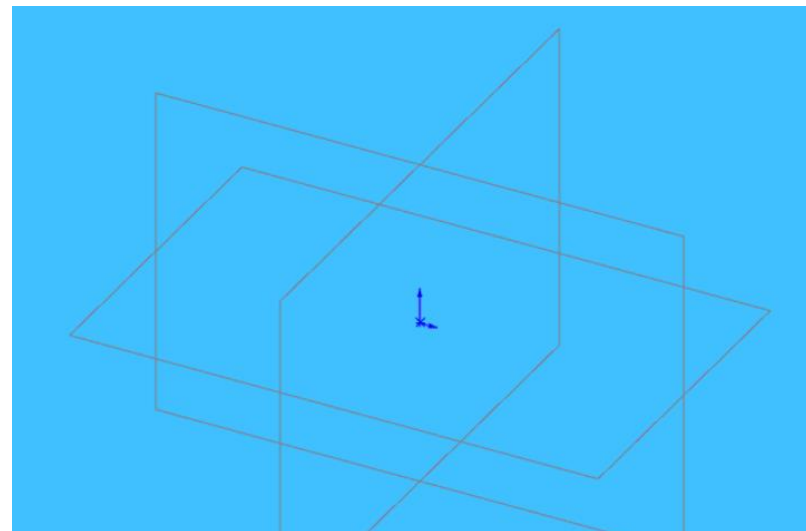
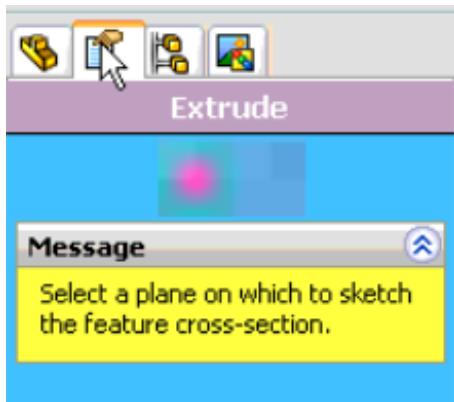


SolidWorks Exercise

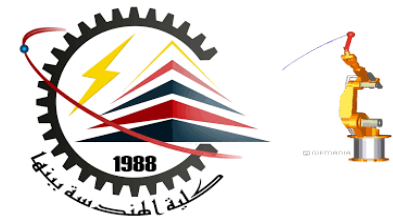
Let's begin by selecting
EXTRUDED Boss Base



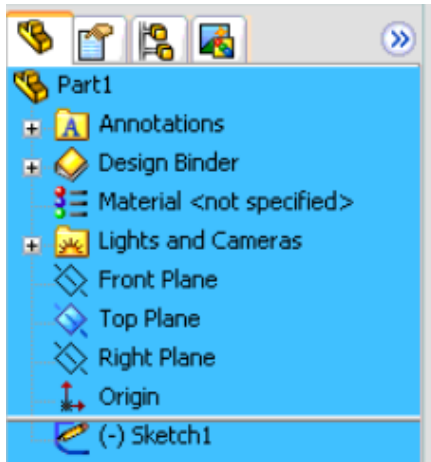
You should notice that your tab will change to property manager asking you to select a plane from the view.



SolidWorks Exercise

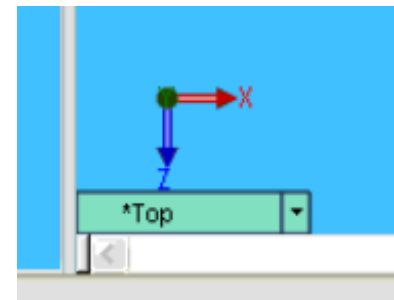


Go ahead and select the horizontal plane shown.

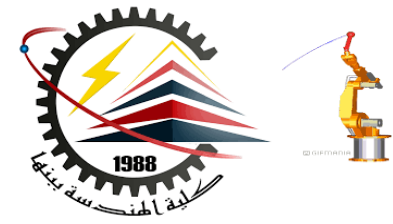


You should notice several things happen.

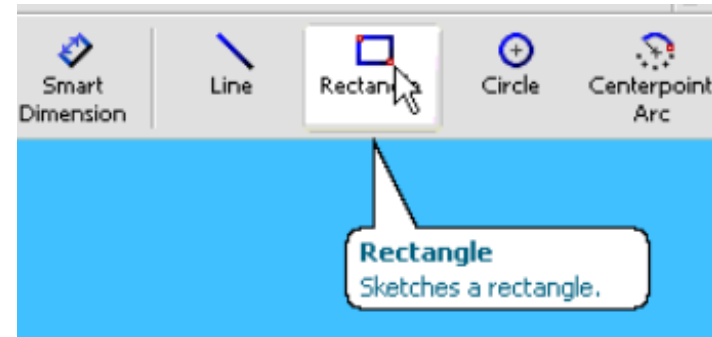
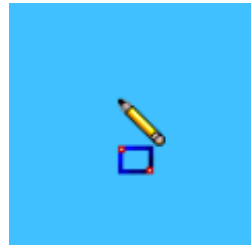
1. You switch to feature manager
2. You have “sketch” buttons on the dynamic toolbar
3. You view below is shown as TOP VIEW! You are looking down on top of the part.



SolidWorks Exercise

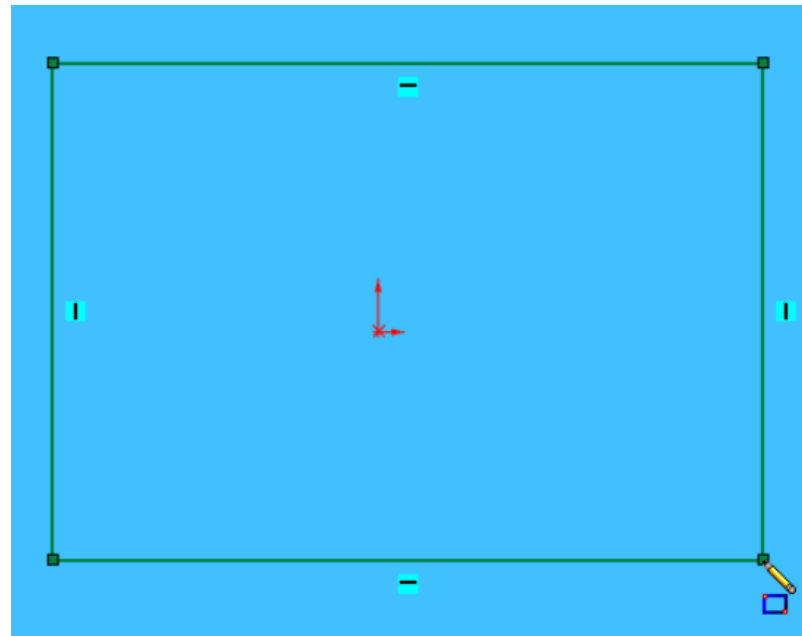


Click on Rectangle. You should see your cursor change to a rectangle with a pencil which means you are sketching.

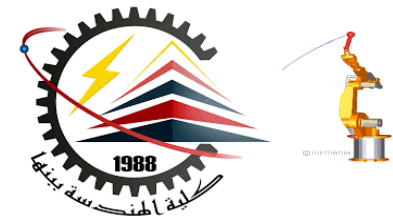


Make a rectangle by dragging the mouse from one corner of the screen to the other. The size does not matter at this point.

Click the GREEN CHECK in FEATURE MANAGER when complete.

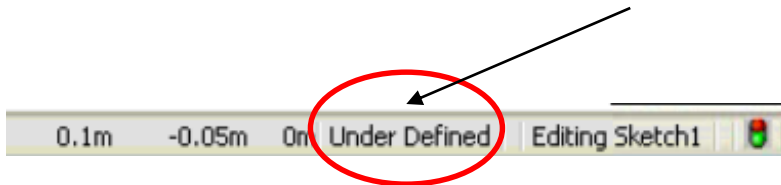
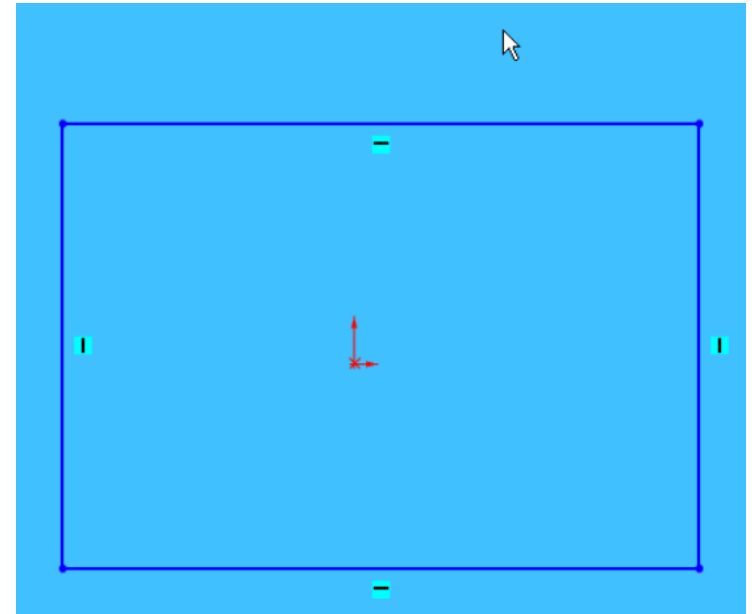


SolidWorks Exercise



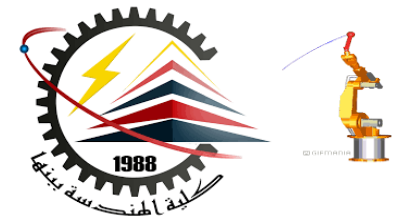
If you hit escape, it will get you out of the rectangle TOOL and back to a normal cursor.

We need DIMENSIONS to our rectangle however.

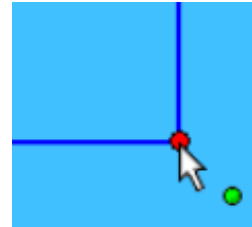


At the bottom right of your screen you will see the figure above. Obviously, we are in the middle of our sketch but it also says, “Under Defined”. This means that there are parts of the sketch that are not defined according to location.

SolidWorks Exercise

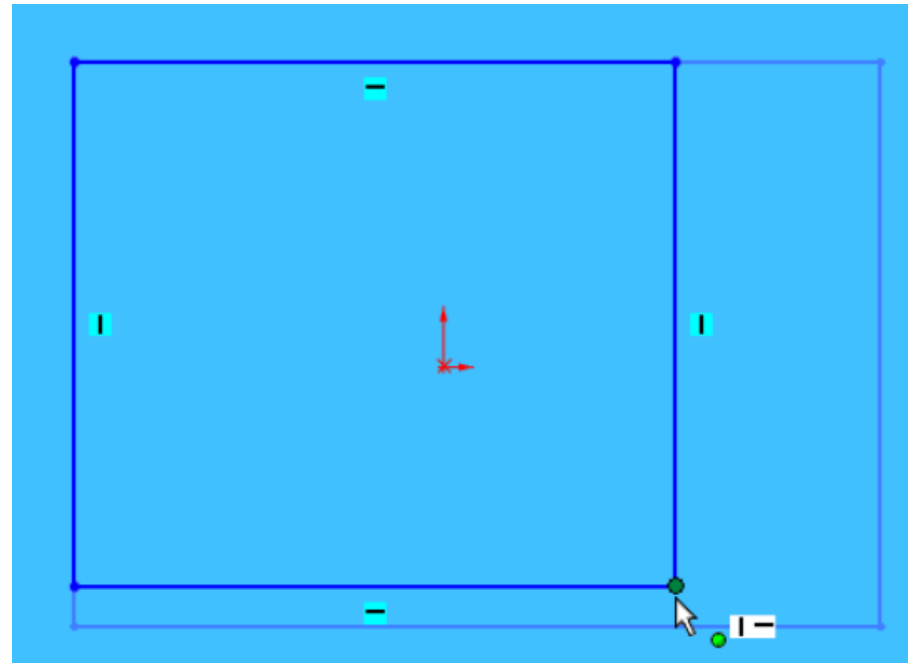


Click the bottom right point on the rectangle.

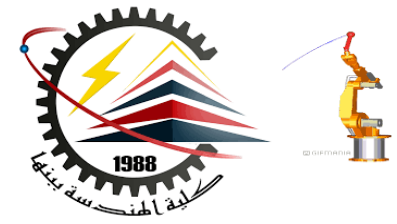


You should be able to move this point around as its location is NOT defined.

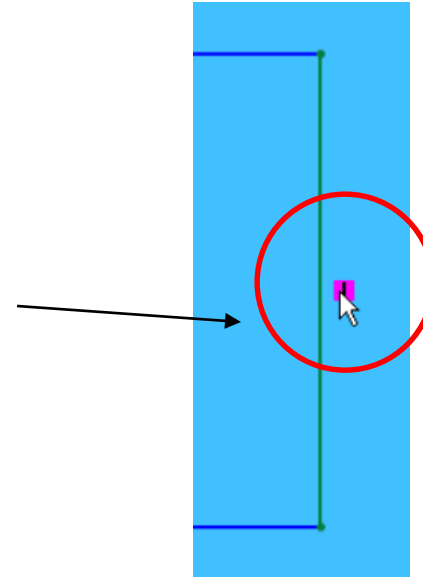
This point has degrees of freedom and is NOT constrained.



SolidWorks Exercise

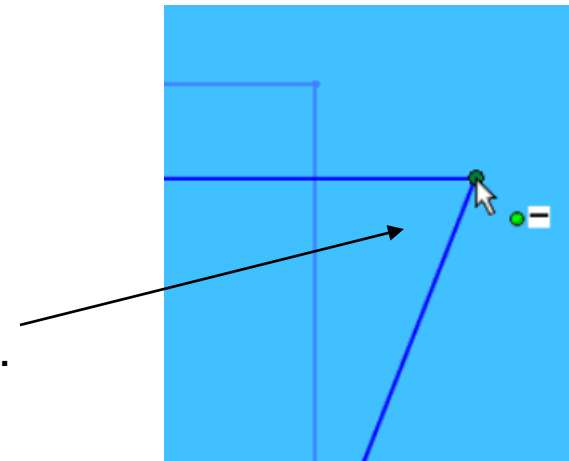


Click on the small box next to the far right vertical line. Notice this box turns pink when clicked and that it has a vertical line in it. This means that this line is vertical and when you click on it you can move it left or right. The vertical lines can thus be moved up or down.

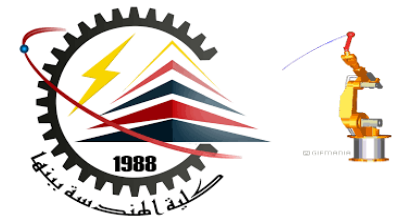


Press the DELETE button, then click on the top right point. Notice the line is not defined as a vertical line anymore and the degrees of freedom are extended.

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

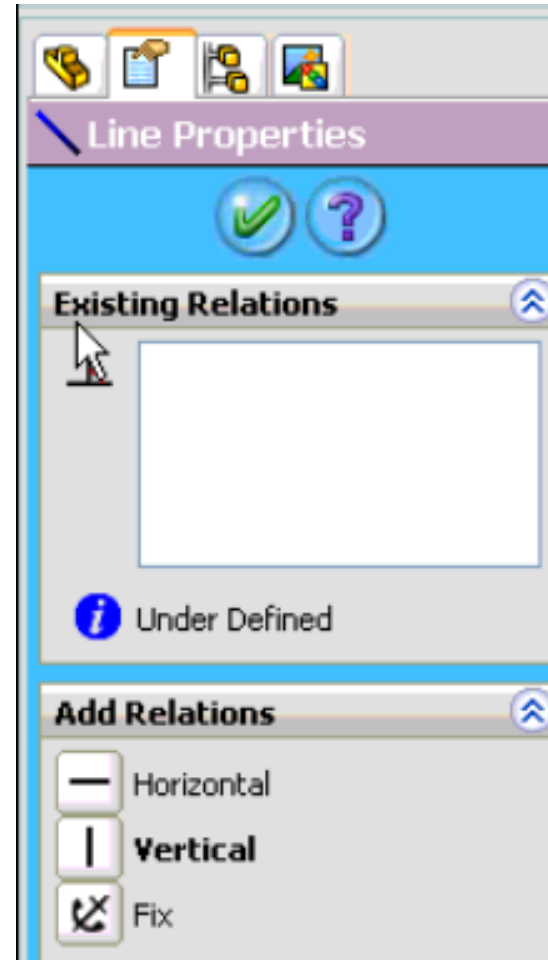


SolidWorks Exercise

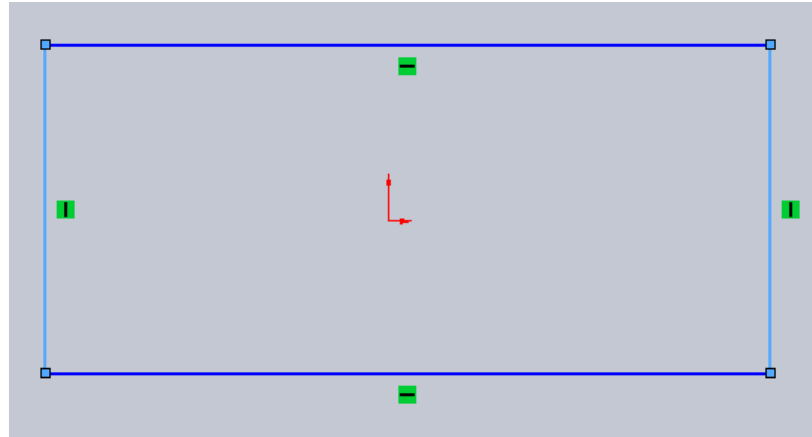
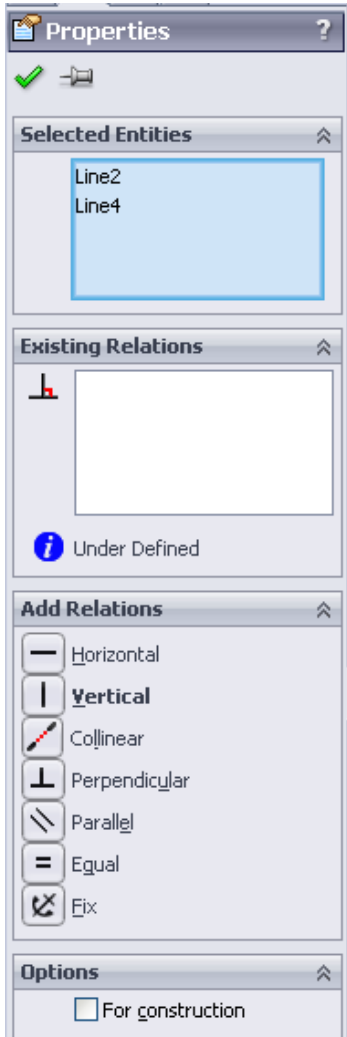
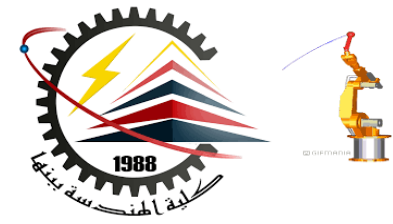


You can add relations by clicking on a line, turns green.

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.



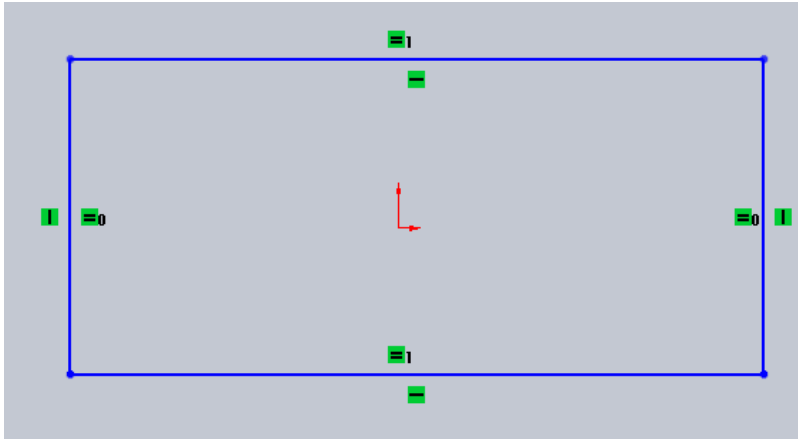
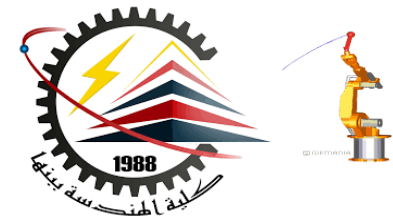
SolidWorks Exercise



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.

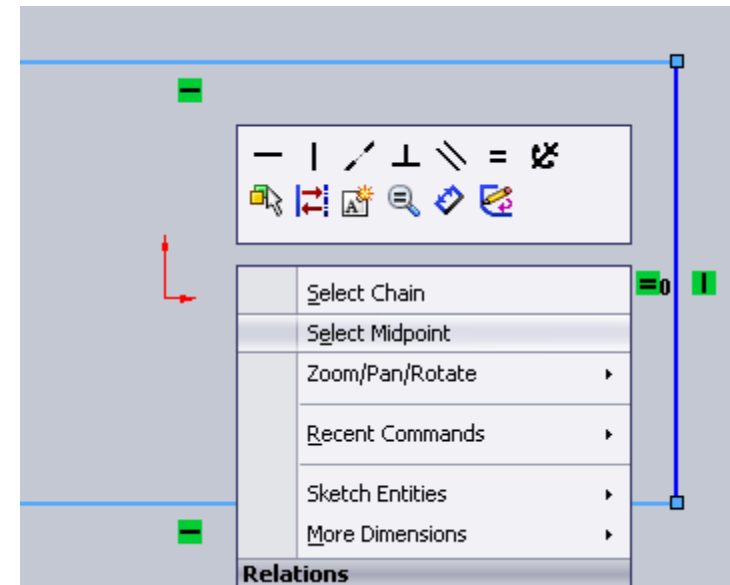
SolidWorks Exercise



Unfortunately, our design is still blue which means we are still **UNDER DEFINED**.

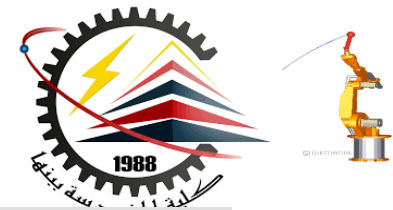
RIGHT CLICK on the bottom horizontal line and choose **SELECT MIDPOINT**. Hold down **CTRL** and select the **ORIGIN**. Add a vertical relation.

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.

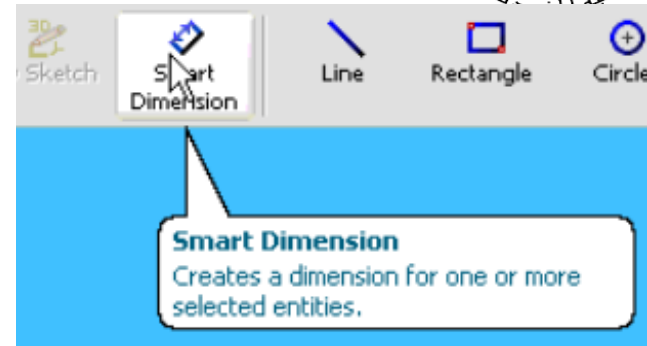


Do the same thing for the sides by adding a horizontal relation to the origin.

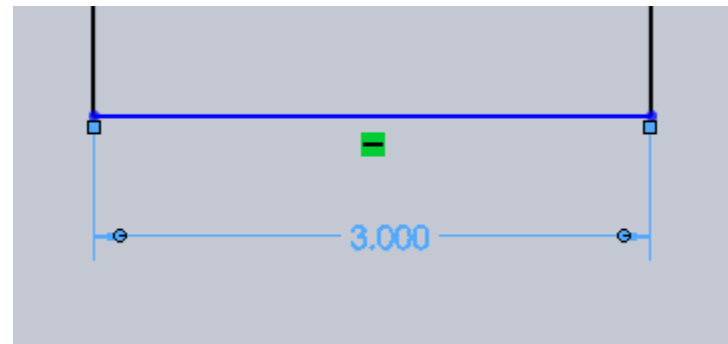
SolidWorks Exercise



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

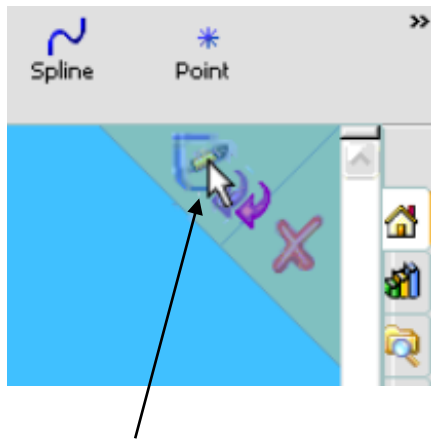
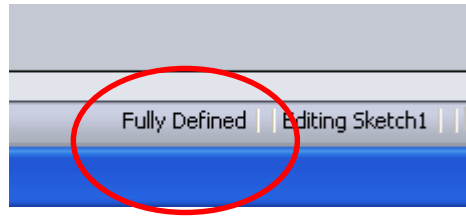
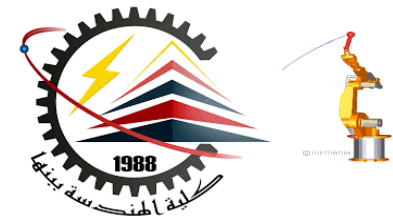


Click the bottom horizontal line of our rectangle and drag the dimension down.



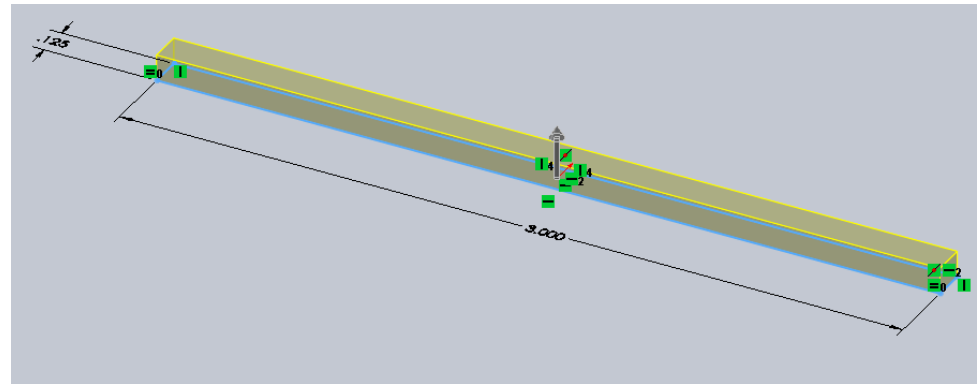
You can change the length of the line by using the slider bar or you can simply enter in 3 inches in the box shown. Enter the value 3 then click the green check. If the box is way too big or small, press “ f ” on the keyboard to fit.

SolidWorks Exercise



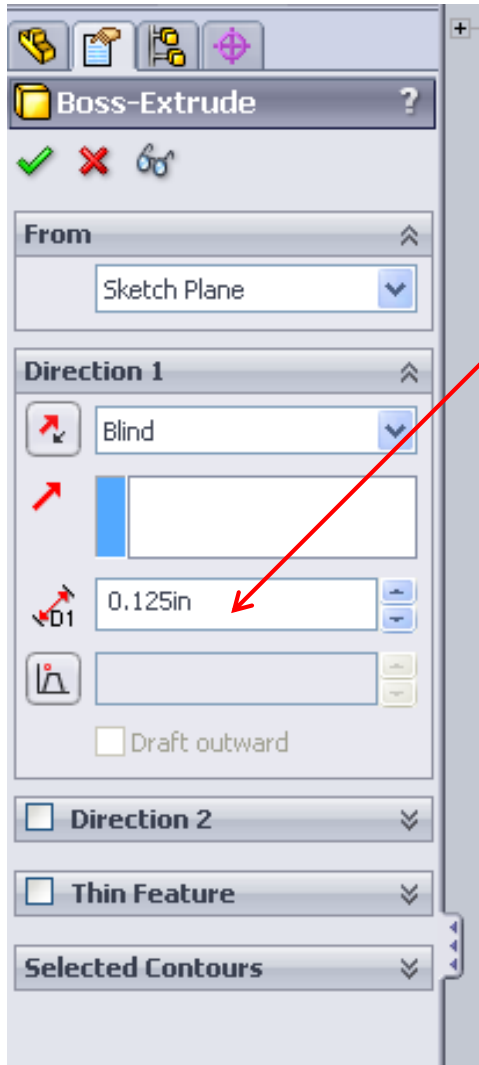
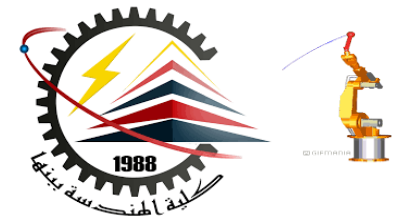
Click in the top right corner to EXIT the SKETCH.

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.

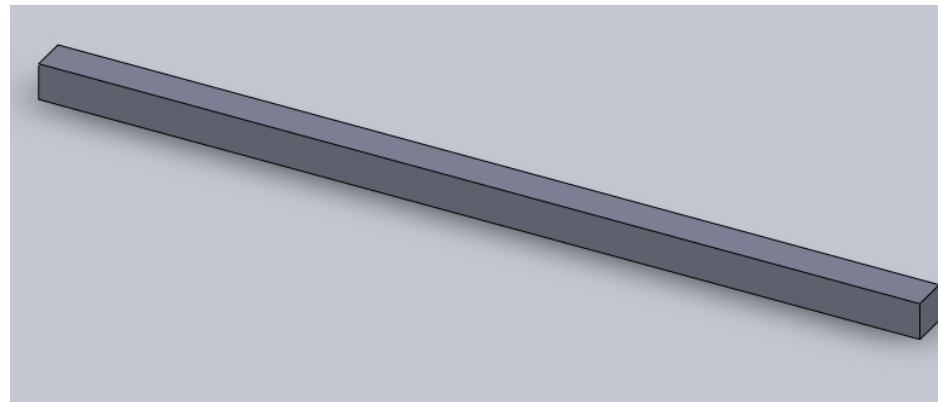


If you hit EXTRUDE earlier, the part will automatically become 3D. If not, click on the part in feature manager then click EXTRUDE BOSS/BASE at the top.

SolidWorks Exercise



After EXTRUDE was chosen, 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.

Thank You for Attention !!

Any Questions

